Structural Optimization of Mounting Bracket Actuator of a Typical Solid Rocket Motor

[1]V.Ganesan, [2]M.S.Prabhu , [3]S.Tamilselvan
[1 2 3] Assistant Professor, Department of Aeronautical Engineering, Hindusthan Institute of Technology

Abstract—The manuscript deals with the study of effect of shape and physical behavior of materials on the mounting bracket of rocket nozzle. The procedure of our project involves analysis for evaluation of the structural response under loading, and modifications are based on this evaluation. The modelling of the bracket for the required dimensions is carried out in CATIA. Loads are applied on the bracket. The actuator is attached to the nozzle near the complex ring through the use of a specially designed bracket known as actuator bracket. The numerical simulation for various boundary conditions and dimensions were carried out in ANSYS. The shear stress and bending stress for various conditions were simulated and the results were studied. The optimal values were obtained for bracket with 13mm thickness which exhibited minimum stress values.

***Index Terms*—Rocket nozzle-Analysis-CATIA-various boundary conditions-shear stress-bending stress-minimum stress value**

INTRODUCTION

The actuator is attached to the nozzle near the complex ring through the use of a specially designed bracket known actuator bracket. The bracket is bolted to the aft exist of core and compliance ring using MP35N bolts. As the bracket compliance ring and aft exist cone section constructed of structural steel. In the least-weight and performance design of aircraft and aerospace structures, sizing and shape optimizations are two traditional techniques and have been widely employed. Topology optimization has been developed remarkably over the last several decades in both theoretical studies and practical applications. By redistributing the material layout and accordingly the load carrying paths, topology optimization has been recognized as one of the most promising techniques in the design of aircraft and aerospace structures.

OBJECTIVE

To design and analyze the stress and to reduce the deflection of a mounting bracket actuator used in a nozzle of a typical rocket motor. The high stress and the bending stress can cause wear of a material. To increase the life of a material, the shear stress, bending stress and tensile stress are to be reduced to behind maximum value and minimum value. After topology optimization, to remove the material where we have minimum stresses and deflection of a model.

ANSYS WORKBENCH

ANSYS Workbench is a new generation solution from ANSYS that provides powerful methods for interacting with the ANSYS solver functionality. This environment provides a unique integration with CAD systems, and your design processes, enabling the best CAE results. ANSYS Workbench is a project-management tool. It can be considered as the top-level interface linking all our software tools. Workbench handles the passing of data between ANSYS Geometry / Mesh / Solver / Post processing tools. This greatly helps project management. You do not need worry about the individual files on disk (geometry, mesh etc.). Graphically, you can see at-a-glance how a project has been built. Because Workbench can manage the individual applications AND pass data between them, it is easy to automatically perform design studies (parametric analyses) for design optimization.

OPTIMIZATION

Meanwhile, plenty of technical difficulties highlighted in the rapid development of aeronautics and aerospace structural engineering promote the progress of topology optimization theories in turn. These notable achievements continue to motivate further studies on the applications of topology optimization in designing complicated engineering structures. topology optimization intends to find an optimal structural configuration within a given design domain for specified objectives, constraints, loads and boundary conditions. Note that the key advantage of topology optimization over shape or sizing optimizations lies in the fact that no specified initial structural topology needs to be presumed a priori. In order to obtain reasonable and practical structural configuration, lots of conditions need to be carefully defined before the topology optimization design of engineering structures. Firstly, due to the complexity of the aircraft structure system, the definition of the design domain is the fundamental factor for topology optimization. The designable and non-designable domains shall be assigned in prior according to the requirements of functional devices, aerodynamic surfaces, manufacturing restrictions etc. In this case, the body of the pylon is assigned as the design domain which is discretized into over 240,000 refined solid finite elements. Two hanging positions the beam tip and lower lugs are assigned as non-designable components which are to be fixed during the optimization procedure. The other two important factors are the definitions of the load cases and boundary conditions, which are the determinants of the load carrying path. For an aircraft structure, tremendous numbers of load cases exist due to different flight attitudes and payloads. As a result, it is essentially important to identify the critical load cases. Here, considering the envelope of the loads, 24 load cases including the structural self-weight, inertial forces, the thrust and weight of the turbine engine under three different attack angles and four different roll angles are chosen for topology optimization. The external loads are then precisely distributed to the two hanging positions. Finally, we come to the boundary conditions. As the designed structures are separated from the global airframe, the boundary conditions are defined at the separation surfaces. In many cases when the fixations are not stiff enough, elastic boundary conditions are needed by using numbers of springs on different DOFs.

The mounting bracket, which is optimized and analyzed, is being used in aerospace launching electronic goods like batteries sensors in automobile for fixing the body to the chassis and for fixing the auxiliaries. The criterion for design of the mounting bracket it depends on the loading also. If any eccentric loading acts on the brackets the design criteria based on the shear and flexural moment. If the loads act transverse, the design is based on shear and bending stress. These mounting brackets are fixed to the structure bolts are pins. Aerospace launching vehicles, for mounting batteries of different weights and size. The problem is to optimize the minimum weight, as the aerospace vehicle is needed to design for minimum weight. In the launching vehicles mounting brackets are mostly used for supporting parts and structures.

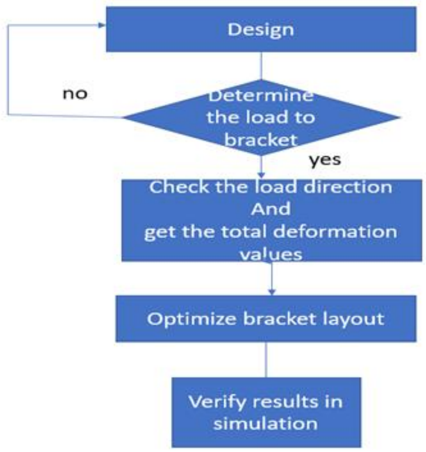
LOADS ACTING ON BRACKETS

Brackets are meant for carrying loads, support structures, bearing that support the shafts. Mounting Brackets are used in various fields such as Aerospace Industries, applications, etc. In the present work we deal with the Mounting brackets used in Aerospace industries, which are used to carry loads of Batteries, Electronic Goods, etc. As Minimum weight is a critical factor in Aerospace Industries we have optimized the weight of the Bracket by reducing the volume of the Bracket by giving a number of cut-outs and by changing materials of the Bracket such as Al Alloy and Graphite Epoxy Composite. These Mounting Brackets are found in large numbers in Aerospace vehicles, as we are able to reduce the weight of a single Mounting Bracket, hence we can reduce the weight to a large extent in the Vehicle, which is the key factor in Aerospace Vehicles. number in Aerospace vehicles, as we are able to reduce the weight of a single Mounting Bracket, hence we can reduce the weight to a large extent in the Vehicle, which is the key factor in Aerospace Vehicles. The shapes of the mounting brackets are according to the application; they may be hanging type or supporting type. In dimensionality, these mounting. The Present work has been undertaken to Design and analysis of the Mounting Bracket to reduce the weight to a large extent by maintaining High Factor of Safety.

METHODOLOGY

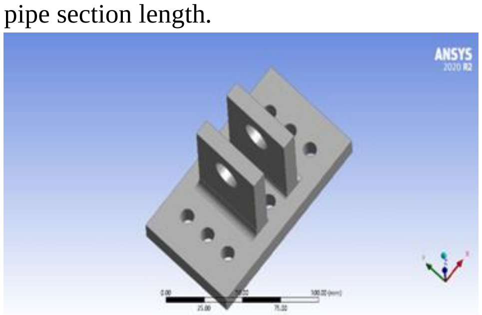
A mounting bracket assembly comprises a flexible body including at least one top member and a flexible angled bottom member connected to the top member.

The flexible body defines a beam insertion aperture between the top member and the bottom member. The mounting bracket assembly further comprises at least one clamp attached to the top member. The mounting bracket assembly may further comprise an integral grounding device disposed adjacent the top member to electrically ground the electricity generating device. The flow chart of methodology as shown as fig.



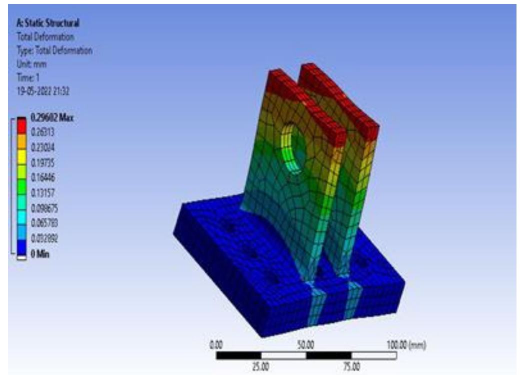
OPTIMIZATION

Based on the identified input and output parameters, parameter correlations can be determined which identifies strong and weak correlations between sets of input and output parameters. ANSYS design exploration also offers response surface generation and response surface optimization which is particularly suitable for cases with larger DoE sets. In this case, a direct optimization component (shown in project setup image above) is used for optimizing the pipe section length.



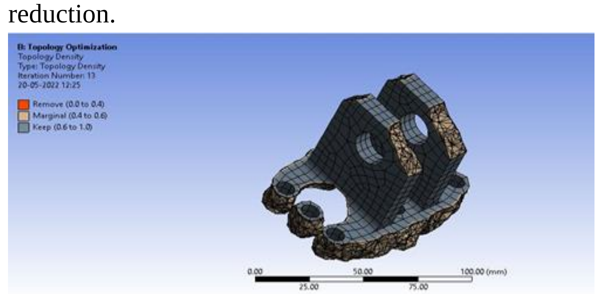
TOPOLOGY OPTIMIZATION

Topology optimization is a mathematical method that optimizes material layout within a given design space, for a given set of loads, boundary conditions and constraints with the goal of maximizing the performance of the system. Topology optimization is different from shape optimization and sizing optimization in the sense that the design can attain any shape within the design space, instead of dealing with predefined configurations.



WHY TOPOLOGY OPTIMIZATION

Topology optimization is done for the purpose of reducing the stress and deflection induced in the bracket. The bracket is modelled and meshed and all the other steps are done and the topology optimization tool is selected from the tree and the result is attached to the tool and the process is done and the rough surface is obtained. Finite element methods along with the experimental techniques are used. The weight Is the most important criteria while considering the actuator mounting bracket, so reduce the weight of the mounting bracket is necessary for the topology optimization method is used for weight reduction.



TOPOLOGY OPTIMIZATION OF BRACKET

The optimization produces considering porosity as a design variable have resulted in a configuration with large void spaces within the carrier material. These configurations are mechanically unstable and since in the real reactor such carrier material cannot be packed at fixed locations, the material might change locations due to the fluid flow.

SPACECLAIM

Space Claim's 3D direct modelling technology is primarily expressed through its user interface in four tools: pull, move, fill, and combine:

Pull contains most creation features found in traditional CAD systems, determining its behavior through users' selection and through the use of secondary tool guides. For example, using the Pull tool on a face by default offsets the face, but using the Pull tool on an edge rounds the edge

Move repositions components and geometry, and can also be used to create patterns (often called arrays).

Fill primarily removes geometry from a part by extending geometry to fill in the surrounding area. Popular uses include deleting rounds and holes from a model. Space Claim Engineer also includes more specialized tools for model preparation. Combine performs Boolean and splitting operations, such as merging parts and subtracting parts from each other.

AFTER OPTIMIZATION

The goal of topological optimization is to find the best use of material for a body such that an objective criterion (i.e. global stiffness, natural frequency, etc.) attains a maximum or minimum value subject to given constraints (i.e. volume reduction). In this work, maximization of static stiffness has been considered. This can also be stated as the problem of minimization of compliance of the structure. Lesser compliance means lesser work is done by the load on the structure, which results in lesser energy is stored in the structure which in turn, means that the structure is stiffer.

MESHING

ANSYS, Inc. offers a wide spectrum of meshing tools for the creation of meshes for all of the physics it supports. Each tool addresses a specific set of needs and capabilities. All are built around the idea of providing a robust and powerful meshing solution for users that reduces the time required to create a mesh while still producing a mesh that will generate accurate results in a timely manner. Element size of meshing is 4.959mm.

BEARING LOAD

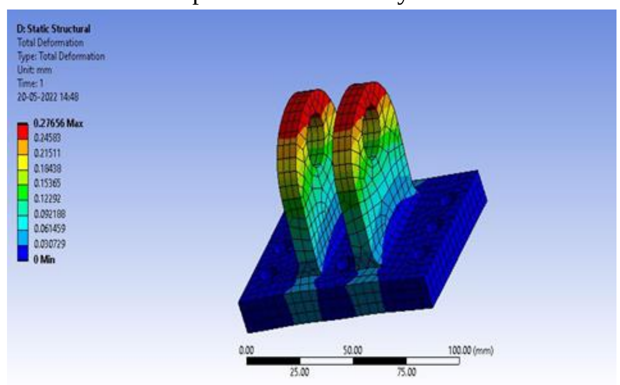
Bearing load have two components Radial load act perpendicular to the axis of rotation, while axial (thrust) load acts parallel to the axis of rotation. In ANSYS, the bearing load boundary condition analyzes radial force only. It will be applied on the interior face of a cylinder in the radial direction.

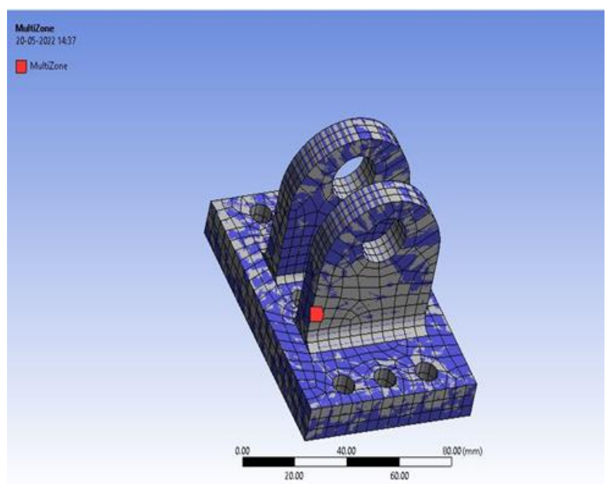
SOLUTION

Select the solution link from the tree diagram and select whatever solution you want, in our case it is the deformation, equivalent stress and strain. Finally click on solve option given on top at the title bar and let the program solve the given condition. Finally, the results will be generated

DEFORMATION

Total Deformation in finite element method the total deformation and directional deformation are general terms irrespective of software being used. Directional deformation may be place because the displacement of the system in a very particular axis or user defined direction. Total deformation is that the vector sum of all directional displacements of the systems.





CALCULATIONS

LOAD & STRESS CALCULATIONS FOR THE BOLTS

Since the load W=50000N is inclined at an angle of 15 to the axis of the bolts, resolving it into horizontal and vertical components.

W_V = 50000 X sin 15 = 12940.95N

W_H= 50000 X cos 15 = 48296.29N

Due to WH component which acts parallel to the bolt axis

Direct tensile load on each bolt

W_{t1} = 12940.95/9 = 1437.88 N

Due to Wv component which acts perpendicular to the bolt axis

Direct shear load on each bolt

Ws = 48296.29/9 = 5366.25 N

Turning moment about the edge of the bracket

Tm = 48296.29 X 0.07

= 3380.7403 N-m

Due to this turning moment, bracket will tilt about the edge EE

Let W= load on each bolt per mm distance from edge 'E' due to the turning effect and L1, L2, L3 are the distance of the respective bolts from the tilting edge.

Total moment of the load on the bolts about the tilting edge

$3380.7403 = 3W (L1)^2 + 3W (L2)^2 + 3W (L3)^2$

$= 3W(17.5)^2 + 3W (35)^2 + 3 W (52.5)^2$

= 918.75W + 3675W + 8268.75W

= 12862.5W

W=0.26N/mm

Heavily loaded bolts are those which are at a greatest distance from the tilting edge. So bolts which are at 'L3' distance from the tilting edge are heavily loaded. Maximum tensile load on each of these bolt due to turning moment

$Wt_2 = W.L_3$

= 0.262 X 52.5

= 13.755 N

Net tensile load on each of these bolt

$W_t = W_{t1} + W_{t2}$

= 1437.88 +13.755

= 1451.635 N

Since these bolts are subjected to tensile load 1451 N and shear load of 5366.25 N then

Equivalent tensile load is

$W_{te} = 1/2[W_t + \sqrt{W_{t2} + 4_{Ws2}}]$

= ½ [1451.63 + √1451.632 + 4 * 5366.252]

W_{te} = 6140.92 N

Equivalent shear load

$W_{se} = 1/2 [\sqrt{W_{t2} + 4* W_{s2}}]$

= 1/2 [√1451.632 + 4* 5366.252]

W_{se} = 5415.11 N

Considering pre tensioning due to torque

Bolt stress for M8*1,12.9 class

= 0.6*108+0.44(6140.92/51.14)

= 64.8+52.83

= 117.63N/mm^2

F.O.S on strength = 108/117.63

= 0.31

STRESS CALCULATIONS FOR THE BRACKET

Due to the horizontal component Wv, this acts perpendicular to the bolt axis

Direct tensile stress on each arm at the base

σt1 (BASE) = 12940.95/(50*10)

= 25.8 N/mm^2

Direct shear stress on each bracket arm at the base

σs(base) = 48296.29/(50*10)

= 96.5N/mm^2

Direct tensile stress on each arm at the bearing point

σt (Bearing) = 12940.95/(18*10)

= 71.894 N/mm^2

Direct shear stress on each arm at bearing point

σs (bearing) = 48296.29/(18*10)

= 268.312 N/mm^2

Maximum tensile stress = 1/2 [σt + √σt2 + 4*τs2]

= 1/2 [701+ √7012+ 4*(96.5)2]

= 714.04 N/mm2

Maximum shear stress = 1/2 [√σt2 + 4*τs^2]

= 1/2 [7012 + 4*(96.5)^2]

= 308.87 N/mm^2

CONCLUSION

This thesis concludes the following on the design and analysis of the structural optimization of mounting bracket actuators of typical solid rocket motor. The experimental setup to mounting bracket model has been designed in CATIA Meshing and loads are applied for the mounting bracket model in ANSYS. Topology optimization is done and the geometry is modified in ANSYS space claim. The analytical and theoretical test values are verified.

REFERENCES

[1]Development of efficient finite elements for structural integrity analysis of solid rocket motor propellant grains. Author: R. Marimuthu.

[2]Conceptual design and shape optimization of a pintle nozzle for controllable thrust propulsion and steering.Author: Vignesh Saravanam.

[3]Design, Analysis and test of a high powered Model Rocket.Author: John J Blandino.

[4]Design and structural Analysis of solid rocket motor casing hardware used in aerospace application.Author: Dinesh Kumar. B.

[5]Typical Solid Propellant Rocket motors.Author: B.T.C. Zandbergen.

[6] Fitzerald JE, Hufferd WL. "Handbook for the Engineering structural analysis ofsolid propellants". CPIA Publication May 1971;214.

[7] NASA-SP-8073Solid propellant grain structural integrity analysis. Cleveland,Ohio: National Aeronautics and Space Administration, Lewis Research centre;June 1973.

[8] Renganathan K, Nageswara Rao B, Jana MK. Analytical solution for a multi-layerthick cylindrical shell under internal pressure and thermal loads applicable tosolid propellant rocket motors. ForshungimIngenierwesen2000;66:57e66.

[9] Renganathan K, Nageswara Rao B, Jana MK. Analytical solution for a multilayer thick cylindrical shell subjected to axial inertia applicable for slump-estimates of solid propellant rocket motor grains. Trends in Applied Science Research 2006;1(2):123e31.

[10] Chen JT, Leu SY. Finite element analysis, design and experiment on solid propellant motors with stress reliever. Finite Elements in Analysis and Design1998.

[11] Chyuan SW. Nonlinear thermoviscoelastic analysis of solid propellant grains subjected to temperature loading. Finite Elements in Analysis and Design.

[12] M.S.Prabhu, Optimization of Unmanned Aerial Vehicle Wing structure. IJISET - International Journal of Innovative Science, Engineering & Technology, Vol. 2 Issue 3, March 2015.