Experimental and finite element analysis of pole mounting bracket for performance and mass optimization

1Uday Kumar Singh, 2Abhishek David
1M.Tech scholar, 2Assistant Professor Mechanical Engineering
1Oriental Institute of Science and Technology, Bhopal, India,
2Oriental Institute of Science and Technology, Bhopal, India

***Abstract* - Two pole mount brackets for 50kg RRU is modelled and analyzed for wind speed 67 m/s when RRU is mounted at the height of 40m from ground level. Software used for modelling is SOLIDWORKS. Experimental and FEA analysis was executed for Model 1 bracket for testing the sustainability of bracket for wind load. Based on these results model 2 bracket is designed for better performance, material savings and low stress concentration area for same loading conditions. All loading conditions are calculated based on standard TIA 222-G. FEA analysis is performed with the help of ANSYS software. The results obtained after performing the analysis shows the difference in the structural behavior of the bracket. Since RRU is very much complex in shape it is replace by a same weight of dummy. An average dummy size is considered and analysed. After testing both brackets withstand all static but model 2 is much more efficient in terms of weight and maximum stress observed in body is also less in model 2 bracket. Bolt pretension used for bracket-to-bracket clamping and bracket to dummy clamping is 20NM.Materail used for bracket is high tensile steel.**

***keywords* - Telecommunication tower, FEA, Pole mount, Wind load testing, Bracket and ANSYS**

I. Introduction

A telecommunication tower is a complex structure and remote radio unit (RRU) is a component is a remote radio transceiver that connects to an operator radio control panel via electrical or wireless interface. When used to describe aircraft radio cockpit radio systems, the control panel is often called the radio head. Like RRU there are also some components which are available on telecommunication tower such as antenna, receiver, microwave, etc. Load of these components are transferred on brackets which clamped in between of RRU and pole. Finite element analysis and experimental studies data help the designer to predict the structure failure and it helps in reduction of prototype failure. Two brackets are considered for our study to analyse stress under right excitation forces. All the tests that are going to be performed on bracket will experience real life load as per predefined standard.

Figure 01: RRU & Telecommunication tower setup in real life

In mechanical engineering a bracket is a intermediate component for fixing one part to another. Pole mount brackets were modeled to securely fasten your plastic or metal enclosures or box structures to a pole. The pole mount brackets are a set of two rails or a plate with a pole clamp and needed screws. Pole mount bracket is made of galvanized high tensile steel, which is guarantees a long time of usage.

II. LITERATURE REVIEW

Tarun Kumar et al. [1] had describe how to design and analyse the brackets at various tensile loading conditions acting at Centre of Gravity point of fasteners. Material used for brackets and fasteners are Al alloy 7075-T6 and Ti-6Al-4V respectively. This is done to ensure structural stability. The equivalent stress is compared to yield strength of the material used. The design and analysis of the brackets and fasteners is carried out using CATIA V5 and ANSYS software. Finite element method is used for the stress analysis. Calculation of equivalent stress, shear stress and total deformation will be considered under this study. Finally, we've carried out modal analysis of brackets to know the right excitation conditions which may cause resonant response.

Priyanka S. Dahaleaet al. [2] in this paper it presents experimental and Finite Element examination of a run of the mill motor mounting bracket. It likewise introduced the Modal Analysis in FEA to decide the frequency band and check the bracket for security. The exploratory examinations for co-connection to build up variety of rate and in this manner decide the idea of Boundary conditions to be utilized in FEA for progressively precise investigation. The Automobile motor case framework may encounter undesirable vibrations brought about by impedance between the street and the motor. Motor bracket has been planned as a structure to help motor. Because of vibrations of motor the openings on the motor Bracket get extended which prompts the failure of bracket. The outcomes got for the static basic and modular investigation have demonstrated that the subsequent limit condition for example one fixed and one pressure support are increasingly precise with less level of deviation 18.6 % with exploratory outcomes, which can be utilized for additional investigation for wellbeing of Engine mounting bracket.

Mohammed Khaja Nizamuddin et al. [3] This paper manages the topology optimization of motor mounting bracket of 'Chevrolet beat' utilizing the instruments CATIA V5R20 for demonstrating and Hyper works for limited component investigation. The utilization of motor mounts is the best answer for dampening the impact of vibrations and transmitting powers between the motor and the car body structure. The principle goal of the work is to limit the heaviness of the motor mounting bracket by thinking about the structure and material format. For various material format and various structures, the anxieties and loads are registered and contrasted with show up at the best model under recommended conditions. It is seen that the most extreme worry for all plans didn't surpass a definitive rigidity for the relating materials. In light of the examination of the weight decrease accomplished in the three streamlined structures, it tends to be reasoned that the most elevated weight decrease has been acquired in plan.

B.K.Sriranga et al. [4] Civil transport aircraft is used for analysis. At the time of flight at maximum lift, the wings will undergo highest bending. Fuselage and wings are attached through wing-fuselage brackets. Bending load joints are used for analysis. In this thesis they used two geometries which consists of I-spar of Aluminium Alloy 2024-T351 and Lug-joint of Alloy steel heat treated AISI-4340 as material. They carried out stress analysis and identified the maximum tensile stress position at lug holes and validation is carried out by considering plate with circular hole.

Umesh S. Ghorpade et al. [5] The process of optimization of natural frequency of engine mount bracket is done to for noise reduction using different lightweight materials for structural stability, which reduces weight of aircraft. The results from thesis concluded that low natural frequency would prove as a hindrance in vibration characteristics of the bracket.

Harish E.R.M. et al. [6] Stress analysis of the wing fuselage lug attachment bracket is carried out to ensure static load carrying capability. Finite element method is used to find maximum tensile stress at one of the rivet hole of I-spar plate.

III. OBJECTIVE

The main objective of the present work to investigate the effects on both bracket for same loading conditions in experimental and loading setup. There are following objective have been set for this work:

1. Prepare CAD model of bracket 1 and test setup.
2. Calculation of wind load for Dummy.
3. Solution and observation of results after FEA analysis.
4. Generation of experimental setup.
5. Observation of results after Experimental setup.
6. Preparation of CAD model of bracket 2 and test setup
7. Solution and observation of results after FEA analysis for bracket 2.
8. Comparison of both brackets

IV. METHODOLOGY

At first the theoretical study required for designing of bracket is done and then further executed. The overall purpose of bracket attachment is to withstand wind loads for predefined height. The purpose of this study is to find the equivalent stress and deformation of brackets by application of loading conditions on both brackets. The 3-dimensional model is prepared using SOLIDWORKS for Brackets. Loadings are assigned and analysis is carried out using finite element analysis software named Ansys Inc.

Governing equation for linear analysis

For a linear static structural analysis, the global displacement vector {x} is solved for in the matrix equation below:

$$[K]\{X\} = \{F\}$$

Assumptions made for linear static structural analysis are:

- [K], which is the global stiffness matrix, is constant
  - Linear elastic material behavior is assumed.
  - Small deflection theory is used.
- {F}, which is the global load vector, is statically applied
  - No time-varying forces are considered
  - No damping effects.

It is important to remember these assumptions related to linear static analysis. Nonlinear static and dynamic analyses are covered in other training courses.

Wind load calculation for dummy as per TIA 222-G:

dummy Size considered for loading is 398mm x 162mm x 573mm.

The weight of the ERS unit is 50 kg.

Projected area for wind in X direction (Filter 1) = 0.398 *0.573 m = 0.228m^2

Wind load for dummy:

$q_{Ultimate}$ = $0.613 * k_z * k_{zt} * K_{d\_Ultimate} * V_{Ultimate}{}^{2} * I_{Ultimate}$

= $0.613*1.34*1*0.95*67*67*1$

= $3503\ N/m^2$

Ultimate pressure load = $n*q_{Ultimate}*G_h$

= $1.6*3503*1.1\ N/m^2$

= $6165\ N/m^2$

Drag coefficient = 1.2

Total ultimate load = (Ultimate pressure load* Total projected area* Drag coefficient) N

= (6165*0.228*1.2) N

= **1687 N**

CAD models and test setup of brackets: Three-dimensional CAD model of pole mount bracket is created using SOLIDWORKS software, A three dimensional views of pole mount for both models of bracket is shown below:

Meshing: After completing the CAD geometry of model of pole mount bracket is imported in ANSYS mechanical for further finite element analysis where next step is meshing. Meshing is a critical operation in finite element analysis in this process CAD geometry is divided into large numbers of small pieces called mesh. Total number of nodes and total number of Elements for both setups are mentioned in table 01. Types of elements used for meshing are tetrahedral and hexahedron.

Table 01: Number of node and elements

Model	**No. of elements**	**No. of nodes**
Bracket 1	188335	297701
Bracket 2	106057	253906

Bracket and experimental test setup: Experimental test setup is generated for validation of FEA results that whether the bracket is able to sustain calculated amount of load or not in real life. Picture of bracket front pole mount and back pole mount along with hardware assembly on pole is shown in figure 06. Complete test setup for wind load testing is shown in figure 07 below:

Boundary conditions are assigned to create a virtual environment of the real life working of the system. The boundary conditions for simulation are explained below:

1. Define the working materials as high tensile steel with density 7850 kgm^3, young's modulus 2e+05MPa and poison ratio 0.3.
2. Set the contact for assembly setup.
3. Fix the bottom face of pole.
4. Apply pretension of 20NM for all clamping regions.
5. Enable gravity for system in downward direction.
6. Apply wind load.
7. Solve the simulation and observe all results.

V. RESULTS AND DISCUSSION

FEA analysis of model 1 bracket:

This analysis shows that whether pole mount bracket can sustain or not for stipulated weight force 50 kg as well as the wind force at the wind speed 67 m/s when RRU is mounted at a height of 40m.Yield stress of material selected for bracket is 350Mpa.Wind load that is applied on projected face is 1687N.Weight of front pole bracket and back pole bracket is observed as 1.37 kg and 1.12 kg respectively. Stress plot and deformation scale is shown below for better visualization of results.

Experimental testing of model 1 bracket:

This test shows that whether pole mount bracket can sustain or not for stipulated weight force 50 kg as well as the wind force at the wind speed 67 m/s when RRU is mounted at a height of 40m in real life or not. Yield stress of material selected for bracket is 350Mpa.Wind load that is applied in form of normal load with the help of test setup as shown in figure 07 which is equal to 172 kg which is equivalent to 1678 N. No damage is observed during testing. Testing pictures and bracket after test setup is shown below for better visualization of results.

FEA analysis of model 2 bracket:

This analysis shows that whether pole mount bracket can sustain or not for stipulated weight force 50 kg as well as the wind force at the wind speed 67 m/s when RRU is mounted at a height of 40m.Yield stress of material selected for bracket is 350Mpa.Wind load that is applied on projected face is 1687N.Weight of front pole bracket and back pole bracket is observed as 1.15 kg and 1 kg respectively. Stress plot and deformation scale is shown below for better visualization of results.

VI. CONCLUSIONS

After analyzing the data obtained in the study following conclusion is drawn:

- In bracket model 1 higher stresses are observed that is 881 MPa which is around the modelled bolts & model constraint location, these stresses are due to model, contact position and area concentration rather than stresses that can be expected at real/practical use. Therefore, these are non-dimensioning stresses. It occurs in a small region in the supporting clamps. No other stresses above yield limit are observed and therefore the structure can be approved with wind load of 1687 N on RRU along with equipment load of 50kg.
- In bracket model 2 higher stresses are observed that is 677 MPa which is around the modelled bolts & model constraint location, these stresses are due to model, contact position and area concentration rather than stresses that can be expected at real/practical use. Therefore, these are non-dimensioning stresses. It occurs in a small region in the supporting clamps. No other stresses above yield limit are observed and therefore the structure can be approved with wind load of 1687 N on RRU along with equipment load of 50kg.
- No Physical damage is observed in brackets, no component loosens and no break is observed after application of 177kg loads on projected wind load area during experimental testing.
- Proposed model bracket is comparatively better in term of stress concentration over the bracket below 350 Mpa.
- Weight of front pole mount bracket of model 2 is 16% less in comparison to front pole mount bracket of model 1.
- Weight of back pole mount bracket of model 2 is 11% less in comparison to back pole mount bracket of model 1.

REFERENCES

[1] Tarun Kumar, Boopathi Raja and G Meenakshi Sundaram, "Stress Analysis for Wing Attachment Brackets" IJERT, ISSN: 2278-0181, Vol. 5 Issue 05, May-2016

[2] Priyanka S. Dahalea, Pallavi D. Pawarb, Vinaay G. Patilc, "Assessment of Boundary Conditions for FEA of Mounting Bracket Using Co-relation with Experimental Results." Materials Today: Proceedings 5 (2018) 13290–13300Bacharoudis E.C., Filios A.E., Mentoz M.D. and Margaris D.P., (2009), Parametric study of a Centrifugal Pump Impeller by Varying the Outlet Blade Angle, The Open Mechancial Engineering Journal, Vol. 2, pp. 75-83.

[3] S. Kirthanaa, Mohammed Khaja Nizamuddin,"Finite Element Analysis and Topology Optimization of Engine Mounting Bracket." Materials Today: Proceedings 5 (2018) 19277–19283
[4] B.K. Sriranga, Dr.C.N. Chandrappa, R. Kumar and Dr.P.K. Dash, "Stress analysis of wing-fuselage lug attachment bracket of a transport aircraft," ICCOMIM, July 2012.
[5] Umesh S. Ghorpade, Prof. D.S. Chavan, Prof. M.V. Kavade, "Static structural and modal analysis of engine bracket using finite element analysis," IJERT, ISSN: 2278-0181, Vol. 1 Issue 10, December 2012.
[6] Harish E.R.M., Mahesha.K, Sartaj Patel, "Stress analysis for wing attachment bracket of a six seater transport airframe structure," IJIRSET, ISSN: 2319-8753, Vol. 2 Issue 7, July 2013.
[7] R.K. Bansal, "Strength of materials," 4th Edition.
[8] Chandrapatla, "Introduction to FEA," 3rd Edition.
[9] Stephen P. Timoshenko, "Theory of elastic stability".
[10] https://www.towernx.com/downloads/TIA-222-G_Explained.pdf
[11] https://kupdf.net/download/ansi-tia-eia-222-g_58b5025c6454a7c247b1e922_pdf
[12] https://etechcomponents.com
[13] https://en.wikipedia.org/wiki/Remote_radio_head
[14] https://bmsce.ac.in/Content/ME/MFELAB_manual_Jan2019_Updated_28_1_2019.pdf
[15] http://research.me.udel.edu/~lwang/teaching/MEx81/ansys56manual.pdf