Published by :
http://www.ijert.org

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 5 Issue 05, May-2016

Stress Analysis for Wing Attachment Brackets

Tarun Kumar B. Jain
B. Tech Student
Department of Aeronautical Sciences,
Hindustan University,
Chennai, India.

Boopathi Raja G
B. Tech Student
Department of Aeronautical Sciences,
Hindustan University,
Chennai, India.

Meenakshi Sundaram
Associate Professor
Department of Aeronautical Sciences,
Hindustan University,
Chennai, India.

***Abstract*—In this paper we describe how to design and analyze 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.**

Keywords—Brackets, Stress analysis, Modal analysis, Static load, Finite elelment method, use of brackets in aircraft, CATIA, ANSYS and Wing attachment Brackets.

I. INTRODUCTION

An aircraft is a complex structure, but a very efficient man-made flying machine. Aircraft are generally built-up from the basic components of wings, fuselage, tail units and control surfaces. The load-bearing members of these main sections, those subjected to major forces, are called the airframe.
Brackets are connector type elements widely used as structural supports to carry hydraulic and electrical lines used in engines, wings and landing gear links. Failure of brackets may lead to the catastrophic failure of the whole structure. Finite element analysis studies and experimental data help the designer to safeguard the structure from catastrophic failure. I-Bracket and Z-Bracket are considered for our project to analyze stress and natural frequencies that may cause resonant response under right excitation forces.

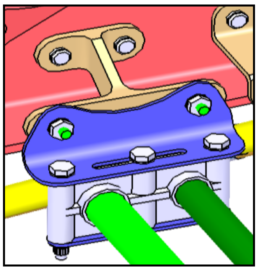

Fig. 1.1 I-Bracket

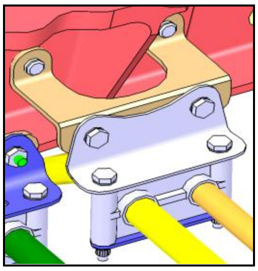

Fig. 1.2 Z-Bracket

II. LITERATURE SURVEY

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" [1]: 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, Prof. D.S. Chavan, Prof. M.V. Kavade, "Static Structural and Modal Analysis of Engine Bracket Using Finite Element Analysis" [2]: 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 will prove as a hindrance in vibration characteristics of the bracket.

Shashikumar.C, Nagesh.N, Ganesh, "Design and Analysis of Wing fuselage attachment bracket for fighter aircraft" [3]: Design and analysis of wing-fuselage bracket for fighter aircraft is carried out to predict the fatigue life. In a metallic structure fatigue manifests itself in the form of a crack which propagates. Fatigue cracks will appear at the location of high tensile stress. As a result maximum stress concentrations were found at one of the lug hole.

Harish E.R.M., Mahesha.K, Sartaj Patel, "Stress Analysis for Wing Attachment Bracket of a six seater Transport Airframe Structure" [4]: 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. PROPOSED METHODOLOGY

At first the theoretical study of bracket is done. The overall purpose of wing attachment bracket is to support the hydraulic and electrical lines. The key areas for modification are identified. The main task in this study is to find the equivalent stress, total deformation and shear stress on brackets by optimizing it for various loading conditions. The 3-dimensional model is prepared using Catia V5 for Brackets. Different type of materials are assigned and analysis is carried out using finite element analysis software named Ansys Inc. Finally, Natural frequencies for brackets at various modes were found which gave right excitation values that may cause resonance.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 5 Issue 05, May-2016

IV. SCOPE OF WORK

This paper explains the ability of the material used for brackets with fasteners to withstand the applied tensile loads about the C.G. of the fasteners at system side. The envelope load due to hydraulic and electrical unit is transformed to design load of about 100 N. Different lightweight materials, which reduces weight of the aircraft is used. Location of stresses and natural frequency of brackets is analyzed.

V. OBJECTIVES

To carry out stress analysis of the wing attachment brackets using Catia V5 and Ansys software. Comparing values of total deformation, equivalent-stress, elastic strain and shear stress. We can further plot the total deformation-load graph and stress-strain curve at different tensile loading conditions to compare brackets. Modal analysis of brackets at different modes of frequency is also carried out to avoid resonant response under excitation conditions.

VI. GEOMETRICAL CONFIGURATION

The geometrical dimensions of the brackets and fasteners are shown in the figure.

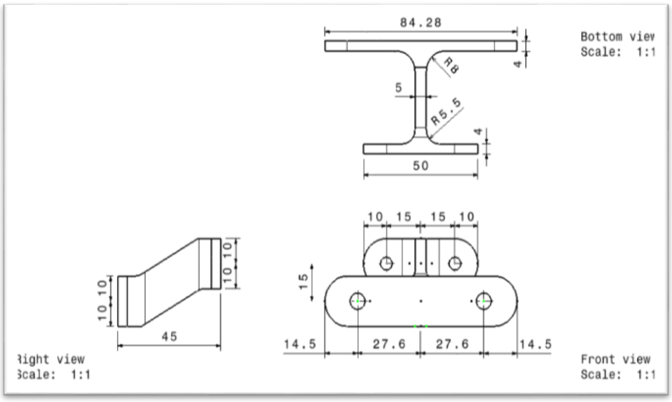

Fig. 6.1 Dimensions of I-Bracket

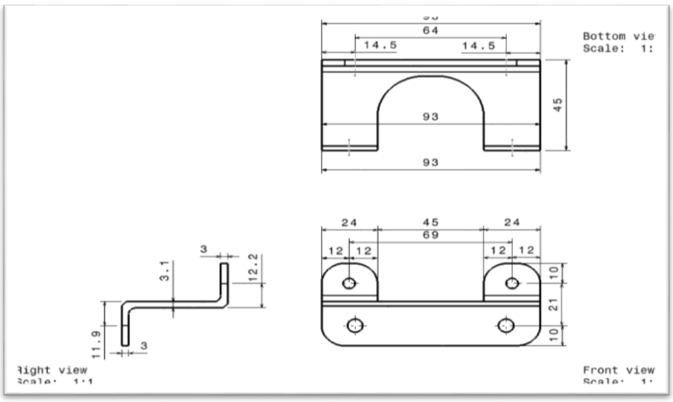

Fig. 6.2 Dimensions of Z-Bracket

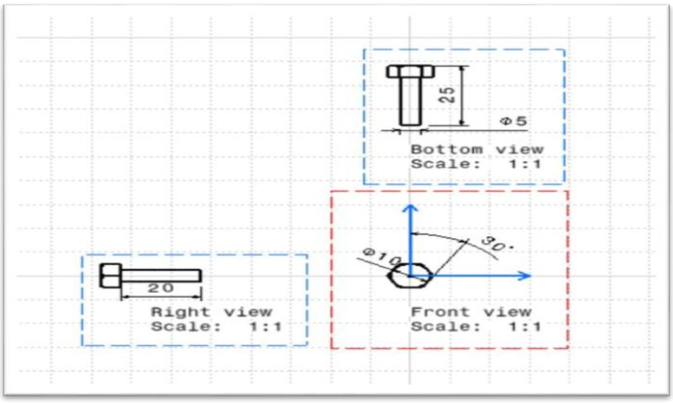

Fig. 6.3 Dimensions of Fasteners

VII. MATERIAL SPECIFICATION

The material considered for brackets is Al alloy 7075-T6, the following mechanical properties are:

1. Density, $\rho = 2810\ kg/m^3$
2. Young's Modulus, E = 71.7 GPa
3. Poisson's Ratio, $\mu = 0.33$
4. Ultimate Strength, $\sigma_u = 572$ MPa
5. Yield Strength, $\sigma_y = 503$ MPa

The material considered for fasteners is Ti-6Al-4V, the following mechanical properties are:

1. Density, $\rho = 4430\ kg/m^3$
2. Young's Modulus, E = 113.8 GPa
3. Poisson's Ratio, $\mu = 0.34$
4. Ultimate Strength, $\sigma_u = 950$ MPa
5. Yield Strength, $\sigma_y = 880$ MPa

VIII. FINITE ELEMENT ANALYSIS

The finite element analysis (FEA) is an implementation of FEM to solve a certain type of problem. The finite element method (FEM) is a numerical technique for solving problems which are described by partial differential equations or can be formulated as functional minimization. A domain of interest is represented as an assembly of finite elements. Approximating functions in finite elements are determined in terms of nodal values. A continuous physical problem is transformed into a discretized finite element problem with unknown nodal values. The software used for the analysis of brackets is Ansys.

A. Meshing of Brackets

Element is an entity into which the system under study is divided. An element shape is specified by nodes. There are many types of element shapes that are further divided into various classes depending on their uses. A volume element has the shape of a hexahedron (8 nodes), wedge (6 nodes), tetrahedron (4 nodes) or a pyramid (5 nodes). In this paper using FEM approach, hexahedron element type meshing is performed.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 5 Issue 05, May-2016

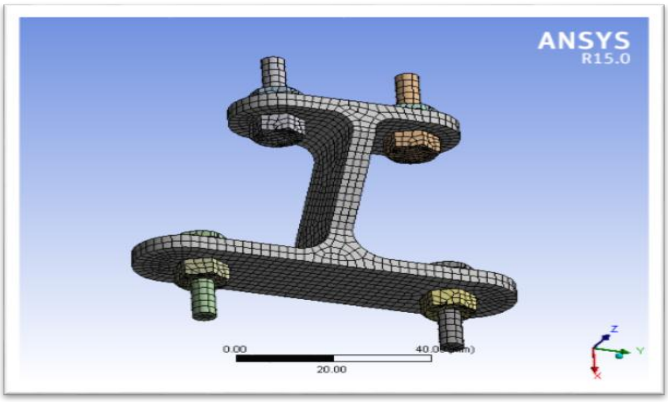

Fig. 8.1 Meshed I-Bracket

Statistics of meshed I-Bracket:
Mesh type: Hex dominant (8 Nodal element)
No. of Nodes: 33588
No. of Elements: 10090

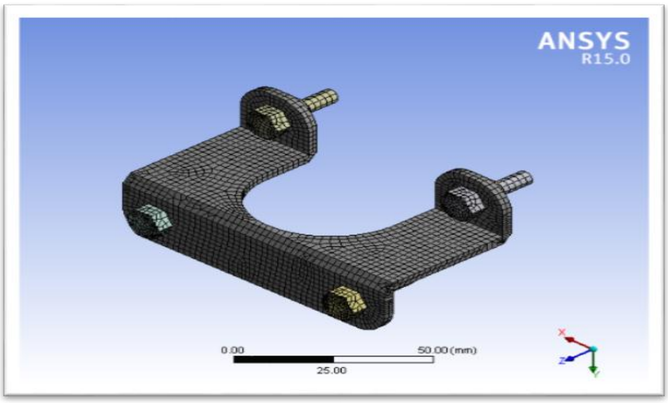

Fig. 8.2 Meshed Z-Bracket

Statistics of meshed Z-Bracket:
Mesh type: Hex dominant (8 Nodal element)
No. of Nodes: 40098
No. of Elements: 11667

B. *Stress Analysis*

The context of this paper is pertained to hydraulic and electrical systems, which are two areas where the routing of air ducts and electrical wires/cable bundles, respectively need to be securely clamped to the structural elements like brackets. Pressure load due to hydraulic unit and electric cables/wires is transformed to brackets as design load acting at C.G. of the fasteners at system side. Other end of the brackets considered to be fixed which can be attached to the spar. Hence when geometry has multiple parts it has to be defined with proper connections between each parts. Only after the creation of these connections the load applied to one part can be transferred to equally to all parts. In the bracket assembly it has 9 parts, of two different materials has been used so each has to be defined with appropriate contacts. Contact type can be bonded or frictional type with frictional coefficient of about 0.15.

TABLE 8.1 DESIGN LOAD: SYSTEM SIDE

F_x (N)	**F_y (N)**	**F_z (N)**	**M_x (Nmm)**	**M_y (Nmm)**	**M_z (Nmm)**
100	100	100	0	0	0

The transformation from envelope load to design load is not required.

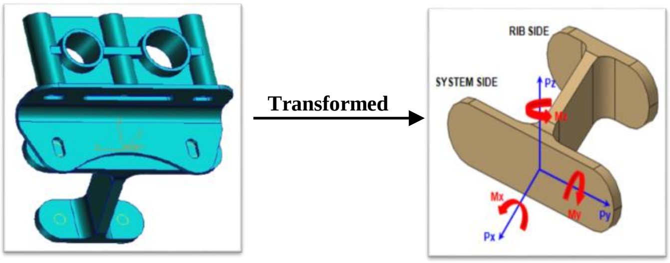

Fig.8.3 Envelope Load

Fig. 8.4 Design Load

C. *Loads and Boundary Conditions:*

The rib side of the brackets is fixed and the system side load is applied at C.G. of the fasteners. The loads and boundary conditions along with the finite element model are shown in the figure.

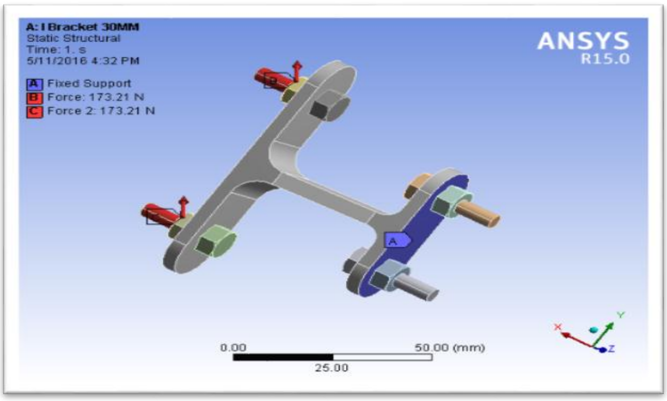

Fig. 8.5 Boundary Conditions for I-Bracket

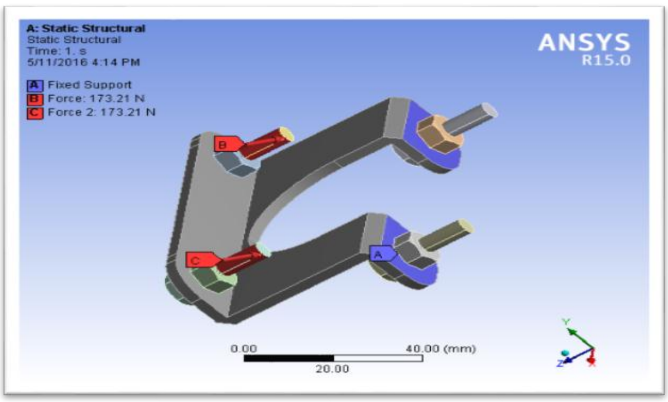

Fig. 8.6 Boundary Conditions for Z-Bracket

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 5 Issue 05, May-2016

D. Comparative data sheets:

On application of various tensile loads at system side fasteners, analyzed results are tabulated to plot graphs. The results taken are total deformation, equivalent stress, elastic strain and shear stress.

TABLE 8.2 ANALYZED RESULTS OF I-BRACKET

S. No.	Load, P (N)	Total Deformation, dl (mm)	Equivalent Stress, σ_v (MPa)	Equivalent Elastic Strain, ε x 10^{-3} (mm/mm)	Shear Stress, τ (MPa)
1	25	0.23	95.43	1.43	24.83
2	50	0.47	127.1	1.77	25.44
3	75	0.71	190.65	2.66	33.58
4	100	0.95	254.2	3.55	44.82
5	125	1.19	317.76	4.44	50.06

TABLE 8.3 ANALYZED RESULTS OF Z-BRACKET

S. No.	Load, P (N)	Total Deformation, dl (mm)	Equivalent Stress, σ_v (MPa)	Equivalent Elastic Strain, ε x 10^{-3} (mm/mm)	Shear Stress, τ (MPa)
1	25	0.09	43.26	0.77	18.43
2	50	0.19	86.53	1.55	36.86
3	75	0.28	129.81	2.32	55.3
4	100	0.38	173.08	3.1	73.74
5	125	0.47	216.35	3.87	92.17

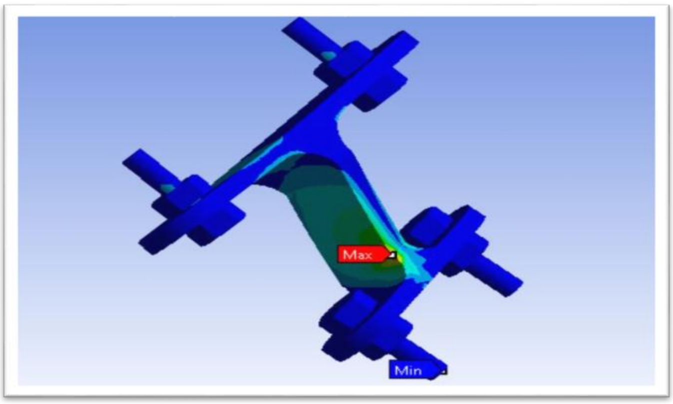

Fig. 8.7 Equivalent Stresses on I-Bracket

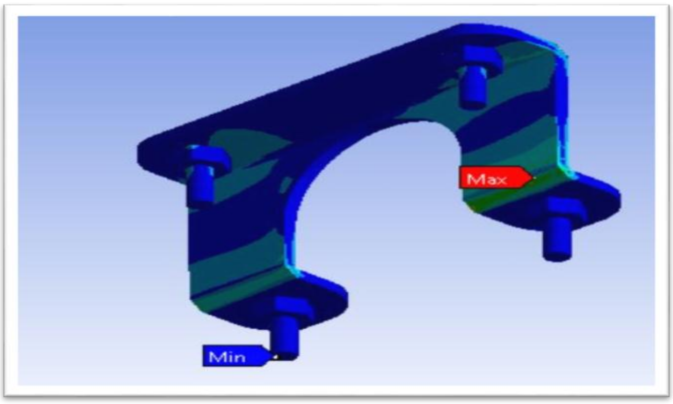

Fig 8.8 Equivalent Stresses on Z-Bracket

IX. MODAL ANALYSIS OF BRACKETS

Natural frequencies of systems are those frequencies at which the resonant response occurs under the right excitation conditions. Knowledge of these critical dynamic frequencies is an essential step in the design and evaluation of a system subjected to dynamic loading. Different modes of frequency can be estimated by modal analysis using Ansys software.

TABLE 9.1 NATURAL FREQUENCIES

Brackets	Mode 1 (Hz)	Mode 2 (Hz)	Mode 3 (Hz)
I-Bracket	689.29	1215.1	2422.6
Z-Bracket	584.32	1169.7	2400.1

X. RESULTS AND DISCUSSIONS

Stress analysis of the wing attachment bracket is carried out in this paper using FEM approach. Maximum tensile stress is about 317.76 MPa in I bracket and about 216.35 MPa in Z bracket observed at 125 N, ultimate load near fixed end. Maximum tensile stress is below the yield strength of the material used for brackets. Shear stress due to various tensile loads on fasteners is also found out.

XI. CONCLUSIONS

The maximum stress values obtained is within the yield strength of the material. Several iterations are carried out to obtain a mesh independent value for maximum stress. The point of maximum stress is the possible location of crack initiation in the structure due to fatigue loading. Various modes of frequency gives right excitation values that may induce high amplitude of vibrations causing resonance.

ACKNOWLEDGEMENT

The analytical work was conducted at P3 Consulting Engineering Private Limited, Bangalore. The author gratefully acknowledge their support. The author would also like to thank Dr. R. Ashokan, Senior Professor, Department of Aeronautical Sciences, Hindustan University, Chennai for providing expert guidance and advice for the analytical work.

REFERENCES

[1] 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.
[2] 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.
[3] 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.
[4] Shashikumar.C, Nagesh.N, Ganesh, “Design and analysis of wing fuselage attachment bracket for fighter aircraft,” IJERGS, ISSN: 2091-2730, Vol. 4 Issue 1, Jan-Feb 2016.
[5] R.K. Bansal, “Strength of materials,” 4th Edition.
[6] Chandrapatla, “Introduction to FEA,” 3rd Edition.
[7] E.F. Bruhn, “Analysis and design of flight vehicle structures,” 1973.
[8] Michael Chun-Yung Niu, “Airframe stress analysis and sizing” 2nd Edition.
[9] Stephen P. Timoshenko, “Theory of elastic stability”.