Dynamic Analysis and Fatigue Life Estimation of Wing Fuselage Attachment Bracket of an Airframe Structure

N. Bhaskara Rao[1] K. Sambasiva Rao[2]
[1]M. Tech. Student [2]Assistant Professor
[1,2]Department of Mechanical Engineering
[1,2]Swamy Vivekananda Engineering College, Bobbili, India

Abstract— Brackets are connector type elements widely used as structural supports for pin connections in airframe structure. In this project a detailed Finite element analysis of the fuselage attachment under the worst loading condition was carried out. During the part of project a dynamic and fatigue analysis of bracket was carried out using finite element analysis package. Then the 3-D model of bracket built in NX CAD is imported into ANSYS using the parasolid format. From the analysis results mode shapes and frequencies are documented by using FEA software. Harmonic analysis is also carried out to plot the frequency Vs amplitude graphs.

Key words: Fatigue Life Estimation of Wing Fuselage Attachment Bracket, Airframe Structure

I. Introduction

Brackets are the primary structural elements in airframe structure that are widely used in connecting different components of the airframe. For example aircraft engine-pylon support fittings, wing fuselage attachment, and landing gear links are some of the typical applications where attachment lugs of various configurations can be found. The catastrophic failure occurring may lead to lug joint bracket of the aircraft structure [6, 7]. Therefore, Finite element analysis (static) and experimental (numerical) data helps the designer to life of the structure from catastrophic failure [8]. Therefore, it is important to establish design criteria and analysis methods to ensure the damage tolerance of aircraft attachment brackets.

A. Material Specification

Selection of aircraft materials depends upon key material properties that are pertinent to maintenance cost and structural performance are Density, Stiffness, Strength, Durability, Damage tolerance, Corrosion. A combination of various materials is often necessary i.e. alloys are used. Commonly 2 alloys are used steel alloys and Aluminum alloys.

B. Manufacturing Process

The process of converting raw materials, components, or parts into finished goods that meet a customer’s expectations or specifications. Manufacturing Processes are Forging and Casting.

II. Literature

Stress analysis is carried out for the given geometry of the wing-fuselage attachment bracket of a six seater transport airframe structure. Finite element method is used for the stress analysis[1]. Rarely an aircraft will fail due to a static overload during its service life. The aircraft needs to execute complicated maneuvers while fighting with enemies. Complicated maneuvers will require instant change in acceleration [2]. The combination of high level of acceleration and complicated maneuvers will introduce high magnitude of loads on the wings [3]. The bending moment will be Maximum at the root of the wing which caused highest stress at this location [4]. Wings are attached to the fuselage structure through wing-fuselage attachment brackets. The bending moment and shear loads from the wing are transferred to the fuselage through the attachment joints [5].

In this project bending load transfer joint is considered for the analysis. First one needs to ensure the static load carrying capability of the wing-fuselage attachment bracket. For the continued airworthiness of an aircraft during its entire economic service life, fatigue and damage tolerance design, analysis, testing and service experience correlation play a pivotal role.

III. Problem definition and methodology

The analysis is performed in both static and dynamic condition. Harmonic analysis is also carried out to plot the frequency Vs amplitude graphs. In this project we shall also find out the life, safety factor and damage of the bracket. Finally design optimization of the bracket shall be done to increase the life of the bracket component.

A. Problem Definition

To design of a wing fuselage attachment bracket against fatigue failure. Linear static and dynamic stress analysis of the wing fuselage lug attachment bracket. Calculation of the fatigue life is done w.r.t crack initiation in the wing fuselage lug attachment bracket.

B. Methodology

The load calculations are done for wing fuselage attachment bracket. Create a 3D model using NX-CAD software. Import the 3D model into Ansys to perform static and dynamic stress analysis. In dynamic analysis, initially modal analysis is done to calculate the natural frequencies. Harmonic analysis is performed to check the structure behaviour at different frequencies due to applied load. Fatigue life calculation is done to estimate the life of the bracket. Optimization of the bracket is performed to increase the life of the bracket.

IV. Modeling procedure

CAD involves creating computer models defined by geometrical parameters which can be readily altered by changing relevant parameters. CAD systems enable designers to view objects under a wide variety of representations and to test these objects by simulating real world conditions. The NX software integrates knowledge-based principles, industrial design, geometric modeling, advanced analysis, graphic simulation, and concurrent engineering. The software has powerful hybrid modeling capabilities by integrating constraint-based feature modeling and explicit geometric modeling.

A. Input for the Wing Fuselage Bracket

A 2D drawing is used to design a 3D model for our component using Unigraphics NX 7.5 CAD software.

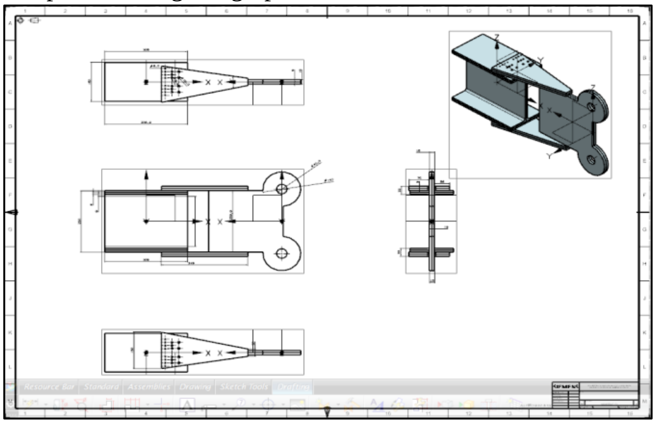

Fig. 1: 2D drawing of Fuselage Attachment Bracket

B. Development of 3D Modelling

1) Sketching

Below is the sketch required to obtain the 3D model of the fuselage attachment bracket from the above 2D drawing input.

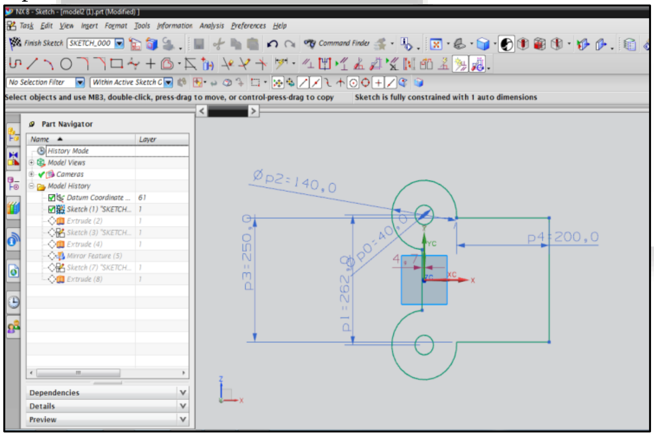

Fig. 2: 2D sketch of fuselage attachment bracket.

2) Extrude of Basic Sketch

Extrude command is used to create a body by sweeping a 2D or 3D section of curves, Edges, sketches in a specified Direction.

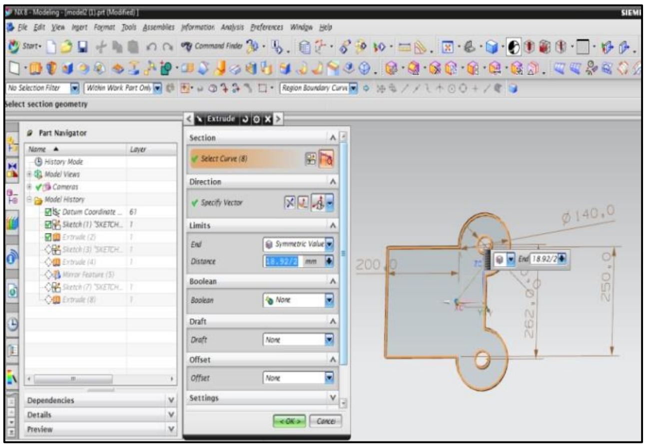

Fig. 3: Extruded sketch of fuselage attachment bracket

3) Sketch created on base body

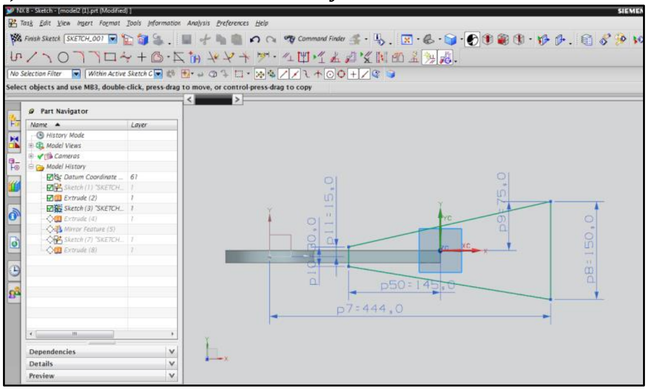

Fig. 4: Creating of sketch on fuselage attachment bracket

4) Extrude of profile curve

Extrude command is used to create a body by sweeping a 2D or 3D section of curves, Edges, sketches in a specified Direction.

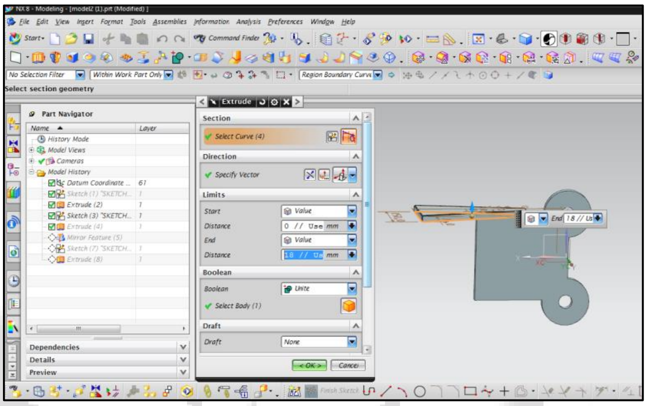

Fig. 5: Extrude of sketch on fuselage attachment bracket

5) Extruded bracket

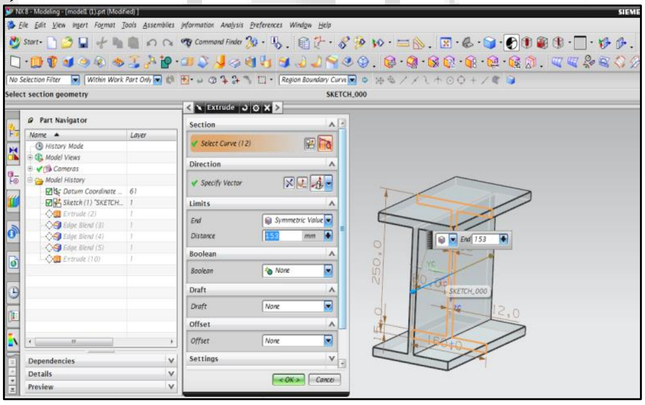

Fig.6: Extruded sketch of fuselage attachment bracket

V. Analysis

A. Structural Analysis of Wing Fuselage Bracket

3D model of the fuselage bracket was developed in NX-CAD from the papers. The model was then converted into a parasolid to import into ANSYS. A Finite Element model was developed with shell and beam elements. Initially load calculations for the wing fuselage bracket are done for 6 cases of 'g' values for FOS of 1.5.The loads calculated are shown in the below table. Static analysis was done by applying the calculated loads for two different materials (Steel Alloy and AISI-4340 Aluminum Alloy-2024-T351). From the analysis the displacement and maximum principle stresses are calculated and tabulated.

S. No	Parameter	Steel Alloy_AISI-4340	Aluminum Alloy-2024-T351
1	Young's Modulus(N/mm^2)	203000	72400
2	Poisson's ratio	0.33	0.33
3	Ultimate Tensile Strength (N/mm^2)	1835	503
4	Yield Stress, бy(N/mm^2)	1600	472

Table 1: Result

B. *Load Calculations*

S. No	Parameter (g)	Load Value (N)
1	1g	12899.25 N
2	2g	25798.5 N
3	3g	38697.75 N
4	4g	51597 N
5	5g	64496.25 N
6	6g	77395 N

Table 2: Load Calculations

C. *Element Types used*

The fuselage attachment bracket, I-section spar and rivets are meshed using shell 63 element type. The bolts are modeled using Beam 188 element type.

D. *Geometric Model*

The 3D model of the wing fuselage attachment bracket is created in NX-CAD software. Because of its small thickness, the mid surface of the bracket is extracted in the NX-CAD software and converted as parasolid model. The parasolid model is imported into Ansys to perform static analysis.

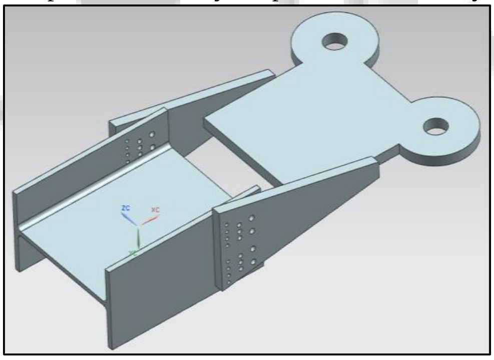

Fig. 7: 3D model in NX-CAD and the model used for analysis

E. *Finite Element Model*

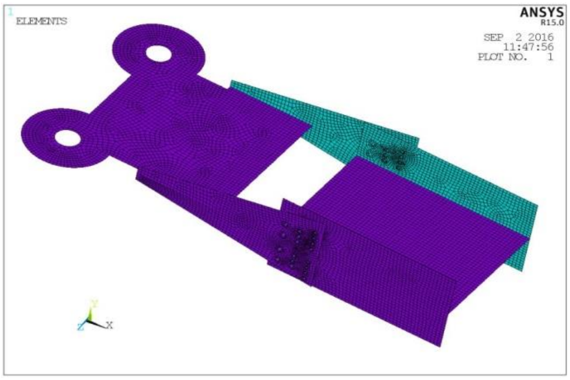

Fig. 8: Finite Element model with shell elements

A Finite Element model was developed with shell and beam elements. A total of 12975 elements were created for the analysis.

1) Boundary Conditions

The semicircle portion of the bracket holes are constrained in all DOF

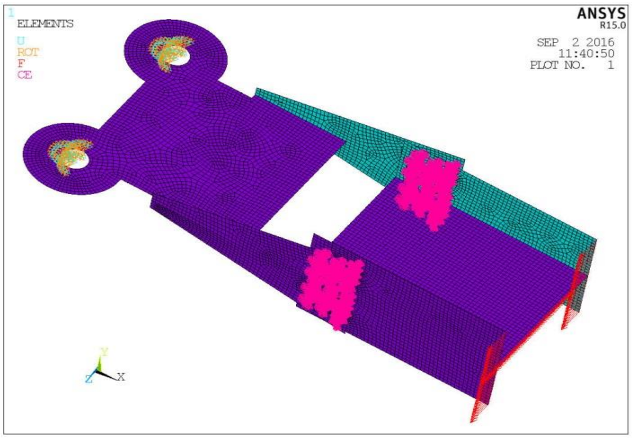

Fig. 9: Boundary conditions used for static analysis

2) Results of Static analysis

The analysis the results are documented. The plots for the load case-1 (1g load) are plotted.

Fig. 10: Total deflection for 1g load

3) Stresses

From the analysis the maximum and minimum principle stresses along with VonMises stresses are plotted. The principle stresses are further used to estimate the fatigue life of the component. The maximum principle stress obtained is 145.9 N/mm^2. The minimum principle stress obtained is 0.296e-5 N/mm^2.The VonMises stress value of 140 N/mm^2.

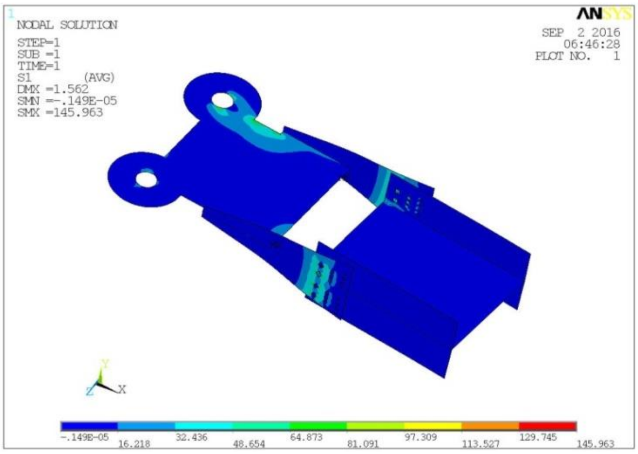

Fig. 11: 1st principle stress for 1g load

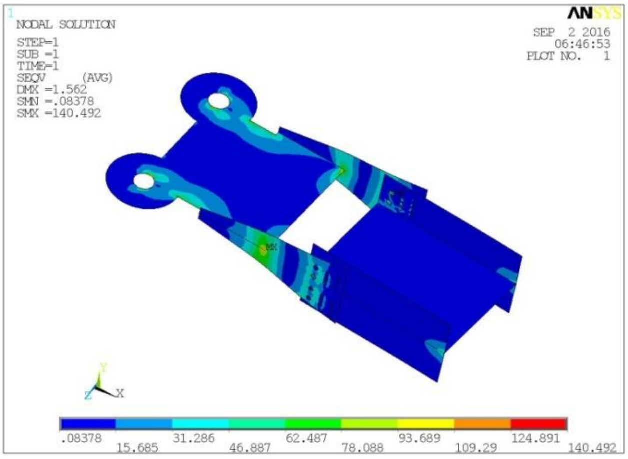

Fig. 12:VonMises stress for 1g load

From the analysis the displacements, Maximum, minimum principle stresses and VonMises stresses are calculated and tabulated as shown.

Load Case	Total Deflection (mm)	S1 (Mpa)	S2 (Mpa)	S3 (Mpa)	VonMises Stress (Mpa)
1g	0.539	146.9	78.59	0.0000029	142.2
2g	1.079	293.8	157.18	0.0000059	284.5
3g	1.61	440.79	235.77	0.0000088	426.8
4g	2.15	587.7	314.36	0.0000118	569.1
5g	2.69	734.65	392.9	0.0000148	711.38
6g	3.2	881.57	471.5	0.0000177	853.65

Table 3: Results

The yield strength of the material as mentioned above for steel alloy is 1600 Mpa. From the analysis results in the above table it is observed that the maximum VonMises stress is 853.65 Mpa for load case 6g.The VonMises stress is less than the yield stress of the material.

VI. FATIGUE ANALYSIS

Fatigue, or metal fatigue, is the failure of a component as a result of cyclic stress. The failure occurs in three phases: crack initiation, crack propagation, and catastrophic overload failure.

A Goodman diagram, sometimes called a Haigh diagram or a Haigh-Soderberg diagram, is a graph of (linear) mean stress vs. (linear) alternating stress, showing when the material fails at some given number of cycles.

The Goodman relation can be represented mathematically as:

$$\sigma_a = \sigma_{fat} \times \left(1 - \frac{\sigma_m}{\sigma_{ts}}\right)$$

A. *Steps involved in Goodman diagram*

The material properties like ultimate strength and endurance limit values are also given as input as

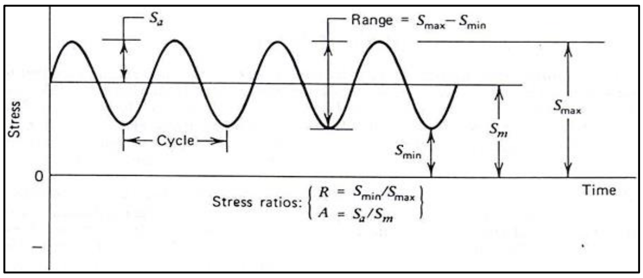

Fig. 13: Schematic Illustrating Cyclic Loading Parameters

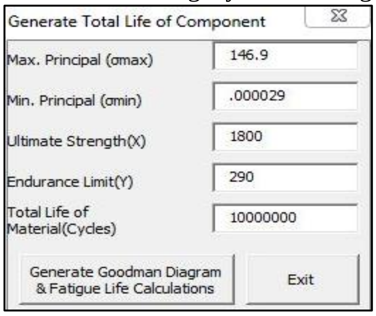

Fig. 14: Inputs given for the Goodman tool

VII. MODAL AND HARMONIC ANALYSIS

Modal analysis is used to determine the natural frequencies and mode shapes of a structure. The natural frequencies and mode shapes are important parameters in the design of a structure for dynamic loading conditions.

A. *Natural Frequency*

Natural frequency is the frequency at which a system naturally vibrates once it has been set into motion.

The natural frequencies depend on stiffness of the geometry and mass of the material.

$$f = \frac{1}{2\pi}\sqrt{\frac{k}{m}}$$

First 10 natural frequencies and mode numbers

Mode No	Frequency (Hz)
1	28.1
2	108.7
3	204.4
4	329.9
5	469.2
6	495.3
7	725.6
8	801.3
9	848.6
10	905.4

Table 4: Results

1) *1st Mode shape @ 28.06 Hz*

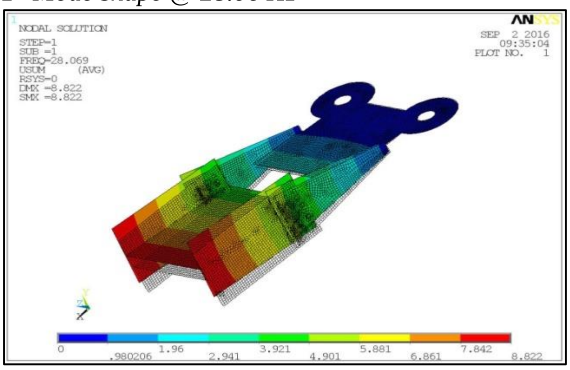

Fig. 15: 1st Mode shape @ 28.06 Hz of bracket

B. Harmonic Analysis

Harmonic analysis was carried out on the wing fuselage attachment bracket to determine the deflections and stress of a structure at the frequencies obtained from the modal analysis.

1) Results of Harmonic Analysis

The harmonic analysis the graph between frequency and amplitude over the frequency range of 0 -500 Hz is plotted.

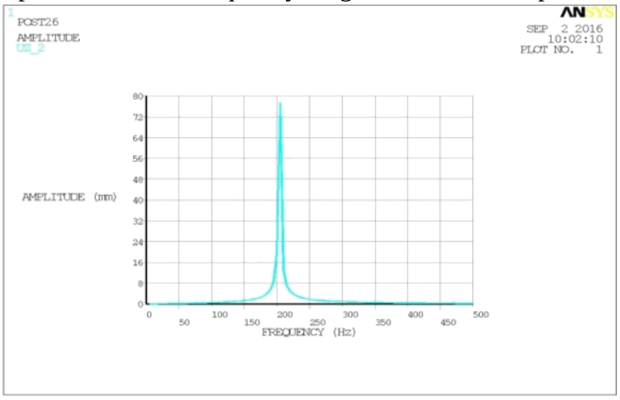

Fig. 16: Graph between frequency and amplitude

The maximum amplitude is obtained at the natural frequency of 204 Hz. The stresses plotted at the frequency of 28 Hz.

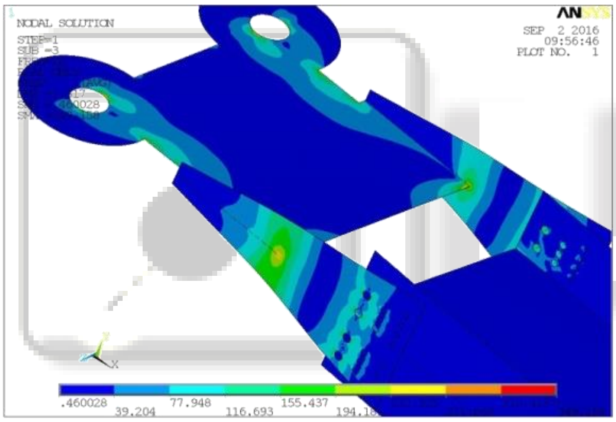

Fig. 17: Maximum stress plot at 28 Hz

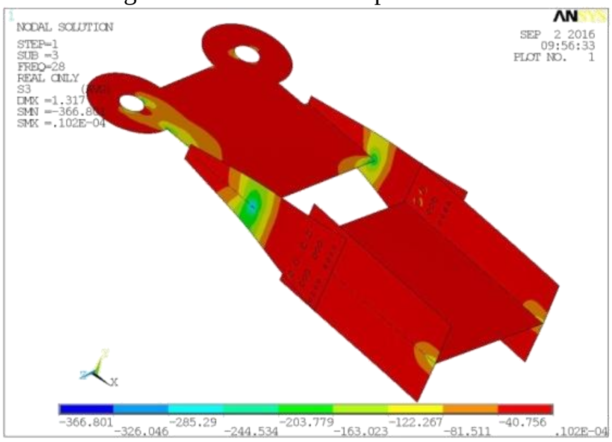

Fig. 18: Minimum stress plot at 28 Hz

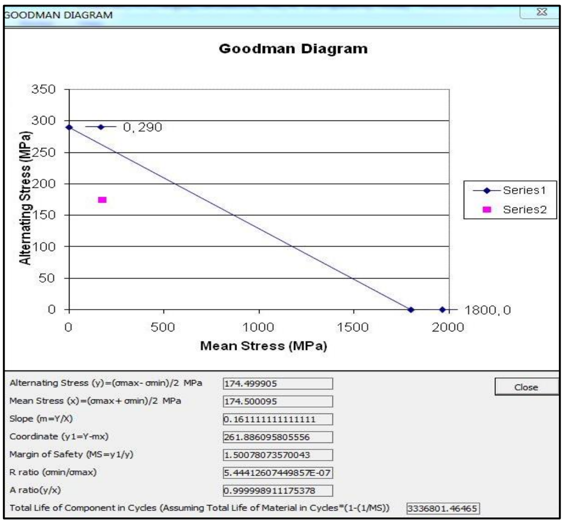

Fig. 19: Goodman diagram with details for 3g dynamic fatigue loads at 28 Hz

The maximum principle stress obtained from the harmonic analysis at 28 Hz is given as input in the Goodman diagram tool. The material properties like ultimate strength and endurance limit values.

VIII. CONCLUSIONS

In this project a detailed Finite element analysis of the fuselage attachment under the worst loading condition was carried out. During the part of project a dynamic and fatigue analysis of bracket was carried out using finite element analysis package and Goodman diagram. The stresses obtained from the static and dynamic analysis were used to do life estimation of the component using Goodman diagram. From the static analysis it was concluded that the wing fuselage attachment bracket is safe for the static analysis for all 6 load cases.

But, however the fuselage bracket has infinite life cycle only for the load case of up to '3g'. From the harmonic analysis it was concluded that the wing fuselage attachment bracket is safe for the stresses developed due to dynamic loads. Life estimation of the wing fuselage attachment bracket was also done for the stresses developed due to dynamic loading.

REFERENCES

[1] Harish E.R.M, Mahesha. K, Sartaj Patel. ""Stress Analysis for Wing Attachment Bracket of a six seater Transport Airframe Structure".(IJERT) Vol. 2, Issue 7, July 2013 ISSN: 2319-8753.
[2] Shashi kumar.C, Nagesh.N, Ganesh."Design and Analysis of Wing fuselage attachment bracket for fighter aircraft". (IJERT) volume 4, Issue 1, January-February, 2016 ISSN: 2091-2730.
[3] SrirangaB.k, Kumar .R."Stress Analysis and Fatigue Life Prediction of Wing- Fuselage Lug Joint Attachment Bracket of a Transport Aircraft". eISSN: 2319-1163 | pISSN: 2321-7308.
[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 - 2012), 11-13 July, 2012.

[5] Amardeepsingh, Jashandeepsingh and Paviterpuneetsingh have published a paper on "Lug Attachment of Wing-Fuselage".International Journal of Aerospace and Mechanical Engineering Volume 1 – No.1, September 2014ISSN (O): 2393-8609.
[6] K. Kathereean, H.S Pearson, and G.J. Gilbert, ―Fatigue crack growth of a corner crack in an attachment lugs‖. Department 72-77, zone 415, Lock Head, Georgia Company, Marietta, Georgia, 30063, USA.
[7] S.Freidrich, J. Schijve, ―Fatigue crack growth of corner cracks in lug specimens‖. Delft university of technology, Department of aerospace engineering, Netherlands, Jan 1983.
[8] O. Gencoz, U.G. Goranson and R.R. Merrill," Application of finite element analysis techniques for predicting crack propagation in lugs", Boeing Commercial Airplane Company, Seattle, Washington, 98124, USA.
[9] R. Rigby and M. H. Aliabadi,"Stress intensity factors for cracks at attachment lugs". British Aerospace, Filton, Bristol BS99 7AR,U.K.,Wessex Institute of Technology, Ashurst Lodge, Ashurst, Southampton S040 7AA, U.K.

