e-ISSN: 2582-5208
International Research Journal of Modernization in Engineering Technology and Science
Volume:03/Issue:06/June-2021 Impact Factor- 5.354 www.irjmets.com

FINITE ELEMENT ANALYSIS OF WING-FUSELAGE LUG ATTACHMENT BRACKET OF A FIGHTER AIRCRAFT

Nishanth M*[1], Deepak S A*[2], Dr. Niranjan Hiremath*[3]

*[1]PG Scholar, M.Tech In Machine Design ,REVA University, Bangalore, Karnataka, India.

*[2]Assistant Professor, School Of Mechanical Engineering, REVA University, Bangalore, Karnataka, India.

*[3]PG Co-Ordinator & Associate Professor, School Of Mechanical Engineering, REVA University, Bangalore, Karnataka, India.

ABSTRACT

Aircraft's maneuverability is the prime factor which judges the performance in an airfield on the venture of war battle fields. The intense number of maneuvers will obviously result in enormous magnitude of loads along the topology of the aircrafts skeletal structures. If these loads are not taken care, it might eventually lead to unprecedented failure in mechanical components. Among these delicate parts lug brackets and I-spar beam are among the vital structural aspects which links between fuselage and wing of the aircraft. It is estimated that '6g' forces act upon the wingspans under tight maneuverers. Possessing the same ideology this research focuses on altering the thickness of aforementioned parts (lug bracket and I-spar) and simultaneously assigning few combinations of alloys like Ti-6Al-4V, Alloy Steel 4340, Al 2024 T351 and Aluminum 7075 T6. Preparation of these model are done in CAD software with meshing and analysis using FEA tools. In the venture of analysis, various behavior of the assembly model was retrieved by inspecting its static structural, modal analysis and its performance under rigorous fatigue tests. The post results are later compared initially with ideal models and comparative study is possessed to justify the obtained results.

Keywords: L-Bracket, I Spar, Maneuvers, Alloys, Fatigue Test, Wingspan, G Forces, Modal Analysis.

I. INTRODUCTION

One of the complex machines ever to exist is the fighter aircrafts that has the enormous agility to maneuver even at extreme G forces. And these are the aircraft which are extensively adapted by military application by all over the globe. An extensive research and development are being carried out across all the nation to master the skill in air field for their respective military troops. Even though the future scope of aircraft industry lends its hands towards the aid of Artificial Intelligence (AI) to fly independently without the need of human intervention, the need to develop an extremely light weight and rigid structures across the body of the aircraft is always under boom.

When an aircraft encounters a tight maneuver by the pilot, a humungous magnitude of load appears to be acted along the structure of the wingspan. This notorious load is developed due to the surge in the lift/drag enacted by the wing, vice-versa, resulting in stresses all along the wings of the aircraft. Thereby, it is the duty of the engineers to design the wing structure in such a manner that these loads considered and calculated even at worst case scenarios. It is estimated by the engineers that approximately '6g' forces is encountered by the wing at worst case maneuvers. In order to fail proof, the design of the wing, the wings are set on this benchmark to reference data.

With that as benchmark, this research work is carried out, to suggest the variations obtained when numerous alloys are adapted to different regions of delicate parts of the wings. Among these delicate parts the I-Spar beam and Lug-Bracket stands high in property wise. Here the I-Spar beam and Lug-Bracket have played vital role on this research work. With reference to it, two categories of models are generated i.e., Ideal models and Custom models, who's specifications are enlisted later on. Alloys of various characters are chosen wisely by referring the bibliographies.

II. RESEARCH GAP

Numerous research journals and various articles were scrutinized in the venture of this project. All the data were later arranged systematically as to figure out what are the possibility of loop holes present in this current field and opportunities. Presence of some of the vital opportunity in this field of research are:

e-ISSN: 2582-5208
International Research Journal of Modernization in Engineering Technology and Science
Volume:03/Issue:06/June-2021 Impact Factor- 5.354 www.irjmets.com

- There is possibility of altering and varying the thickness of the geometry of the model with reference to the controlled and reasonable manner.
- Use of other Titanium alloys with Aluminum alloys simultaneously to understand the behavior of these models under varied geometry.
- Performing modal analysis of the Ideal Model A & B (discussed later on) under Alloy steel ASIS 4043, Ti-6Al-4V, Alloy Steel 4340, Al 2024 T351 and Aluminium 7075 T6 as suggested by our primarily referred journal papers.
- Opportunity available to scrutinize the stress concentration on the sharp edges of the flanges in the Lug brackets and pin holes of the models.

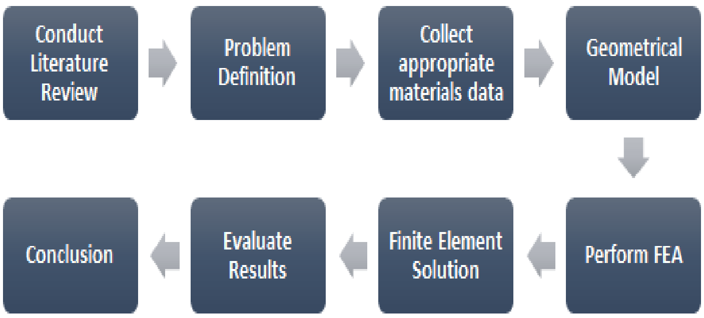
III. OBJECTIVES

The objective of this research is made possible from analysing and performing comparative study of lug bracket, I-spar and its flanges. The key objective of this research project is:

- To study the comparative behaviour of lug bracket and I-spar individual part under combinations of alloys like Ti-6Al-4V, Alloy Steel 4340, Al 2024 T351 and Aluminium 7075 T6.
- Investigate the behaviour of the aforementioned material under static, fatigue and modal analysis by altering the thickness of lug bracket and I-Spar beam.
- Concluding whether either of the used martials behave better than the ideal geometry which is under reference journal papers.
- If these concluded results procure better than the ideal ones, then justifying them how these materials behaved harmonically under various thickness of model.

IV. METHODOLOGY

The below flowchart depicts the illustration on how this research work was carried out. These outlines the process undertaken by us, in order to achieve our desired parameters.



1. **Conduct literature reviews:** The following mentioned research journals were scrutinized to understand the interaction and behavior of research work-related topics. This step provided a validated insight of how the opportunities and trends go on towards on interest. A series of journals, articles and publication data were imbibed to know the importance and drawbacks of the current trends and opportunities.
2. **Problem Definition:** On referring the journals and subjective articles, the problem statements were set and were pointed out hierarchically. These missions were the ignite for this current research work in the verge of the study.
3. **Collect appropriate materials data:** The details with regards to this research work were collected schematically and briefed out at the introduction section of this research work. Details representing the mechanical properties, chemical properties and initial analogy of the aluminum alloys, titanium alloys were retrieved and represented herewith.
4. **Generate Geometrical:** Model The models prepare in this current research work were aided from CATIA_V5 for modelling and drafting purposes. These were the base for further analysis in FEA tools.
5. **Perform FEA ANSYS**: Workbench tool was incorporated for retrieving static structural, modal and fatigue test from prepared 3D CAD models. Numerous data needs to be obtained from post analysis via graphical and pictorial representation.

6. **Finite Element Solution:** Post analysis, the retrieved data were tabulated in tabular column for the ease of comparison from humungous available data. These were schematically arranged, which is represented in the forthcoming section.
7. **Evaluate obtained results**: Comparison of multiple behaviors were notified and made analyze with our respective guides and sources, for the in-depth changes in characteristics of the system. As these aid in evaluating the obtained results with our prepared custom models.
8. **Concluding the result:** Once al the parameter was retrieved, notified and evaluated, then a set off observation is observed and highlighted. These highlighted bulletins showcase the peculiarities of the analyzed models.

V. MODELING AND ANALYSIS

In this research work, of finite element analysis section, the preparation of geometrical modelling, meshing of model and conduction of analysis are elaborately explained herewith.

1. Geometrical modelling preparation:

After extensive research from analyzing the data, changes were made alongside the Lug bracket setup and I-spar support for fuselage. The 2D designs of the parts were initially modelled in AutoCAD.

Once the feasibility of the drawing was verified, the same data were taken in for 3D CAD model. The tool used for modelling Lug Bracket, I-spar, rivet joints is CATIA_V5. It was later procured for assembly section and saved in ".CATproduct." for future references. The assembled sections were later saved for in various format as ".stp" as to be compatible for further ANSYS software for meshing and analysis. The files prepared for drafting are saved in as .dwg and .pdf formats for future references. By this, we have modelled an ideal model and optimized custom model.

Total of 3 assembled sections were modelled in this project use. The individual thickness of the lugs bracket and I-spar were varied accordingly as per various dimensions as listed below:

- **Ideal Model A & Ideal Model B: Thickness 10mm**
- **Custom Model A: Thickness 8mm**
- **Custom Model B: Thickness 12mm**

The reference names were put forth as to distinguish across various result sections which are discussed later on here. The model preparation for Ideal Model A and Ideal Model B is referred from journal paper named "Simulation of Wing-Fuselage Attachment Bracket Lug for Fighter Aircraft". This journal data holds primary reference tool for our procurement in further preparation of CAD models.

The CAD modelling of wing fuselage lug attachment bracket for a typical fighter aircraft is as shown in the below figures. The attachment bracket consists of lug and a portion of spar connected to each other by several rivets.

This lug consists of two pin holes with integrated bottom and top flanges which is used to connect the I spar. The pin holes in lug helps in connecting bracket to the fuselage frame by pin joints. The geometrical specifications of lug bracket for the "6g" design, considered for analysis is taken from a referred journal source.

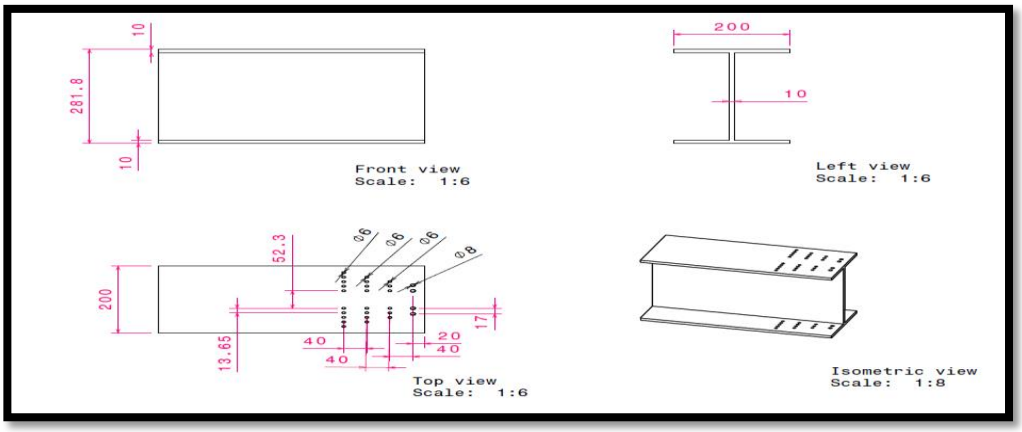


Fig.1.1 Design draft of 10 mm Thickness I -Spar beam

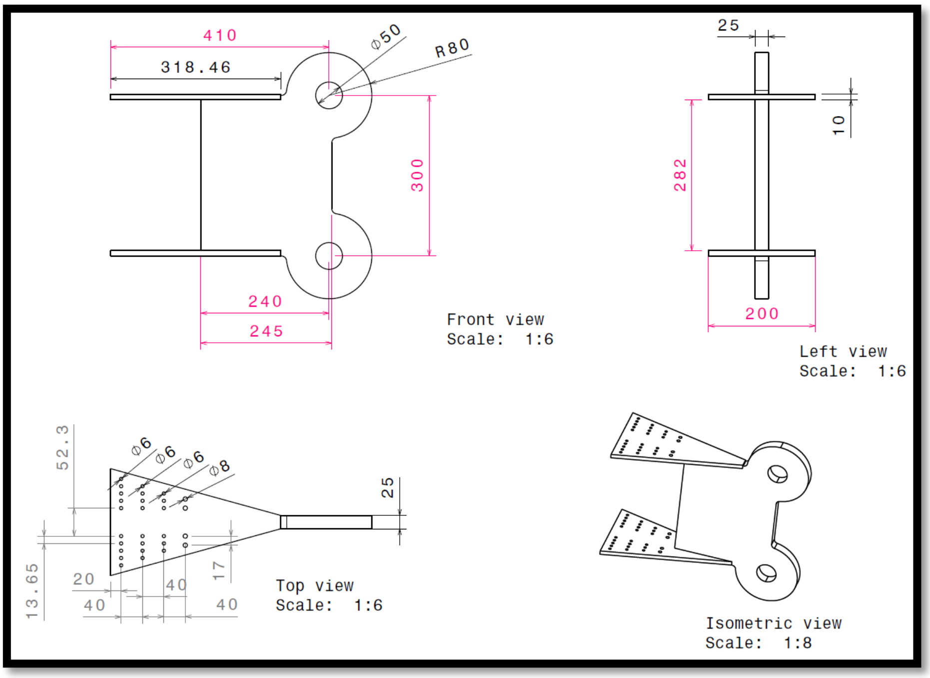


Fig. 1.2 Design draft for 10 mm thickness Lug bracket

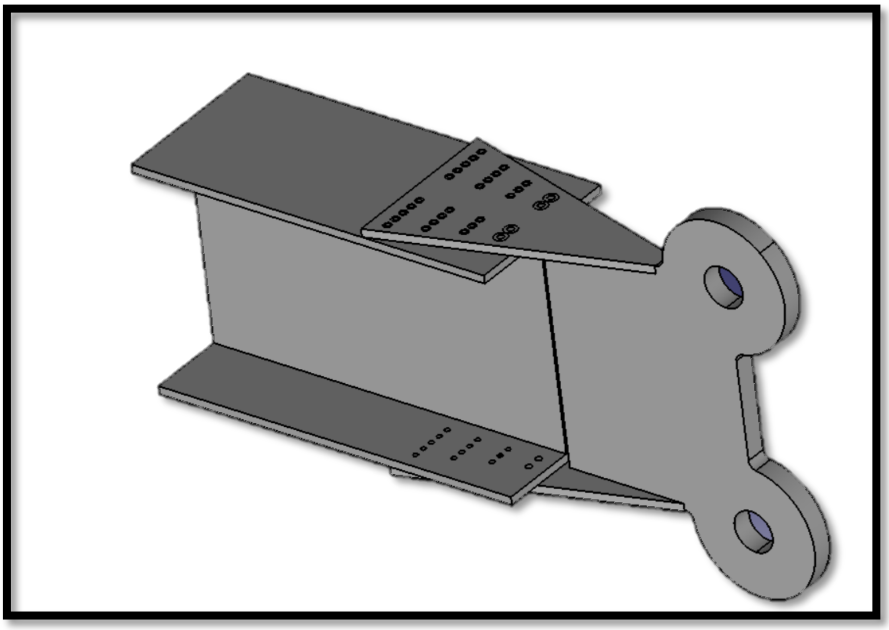


Fig. 1.3 Assembled section of Lug bracket and I-Spar beam

2. Finite Element Analysis

- **Pre -processing Steps -Define and apply material property:**

Now, the collected resource data is put forth to individual models based on from research study. The material property data are retrieved from various referred journal and research papers, which are listed in the references. For instance, the assigning of material is done with the help of ANSYS Workbench 17.1 software. The details of the same is listed below:

Table:1.1 Various materials assigned for this model

Model	Lug Bracket	I-Spar	Rivets
Ideal Model A	Alloy Steel 4340	Aluminium 7075 T6	Aluminium 7075 T6
Ideal Model B	Ti-6Al-4V	Al 2024 T351	Al 2024 T351

Custom Model A	Ti-6Al-4V	Alloy Steel 4340	Alloy Steel 4340
Custom Model B	Ti-6Al-4V	Aluminium 7075 T6	Aluminium 7075 T6

Properties like density, Young's Modulus, Poisson's ratio, yield strength having isotropic properties are fed as input data. The material property for rivets and I-spar are considered to be the same, the reason is justified as because the more malleable property is preferred rather than master property taken for lug brackets.

A total of 4 individual materials are used in the project whose material assignment vary.

Table:1.2 Material property used in this research study

Al 2024 T351	
Property	**Value**
Density (kgm^-3)	2800
Youngs Modulus (GPa)	72
Bulk Modulus (GPa)	70.9
Shear Modulus (GPa)	42
Poisson's Ratio	0.33
Interpolation	1
Tensile Ultimate Strength (MPa)	503

Alloy Steel AISI 4340	
Property	**Value**
Density (kgm^-3)	7850
Youngs Modulus (GPa)	210
Bulk Modulus (GPa)	140
Shear Modulus (GPa)	80
Poisson's Ratio	0.3
Interpolation	1
Tensile Ultimate Strength (MPa)	740

Aluminum 7075 T6	
Property	**Value**
Density (kgm^-3)	2810
Youngs Modulus (GPa)	71.7
Bulk Modulus (GPa)	70.3
Shear Modulus (GPa)	2.7
Poisson's Ratio	0.33
Interpolation	1
Tensile Ultimate Strength (MPa)	572

Ti 6Al 4V	
Property	**Value**
Density (kgm^-3)	4430
Youngs Modulus (GPa)	113
Bulk Modulus (GPa)	118.5
Shear Modulus (GPa)	42
Poisson's Ratio	0.34
Interpolation	1
Tensile Ultimate Strength (MPa)	950

3. Meshing CAD model

The tools used for meshing the modelled assembly is ANSYS Workbench 17.1. Refined meshes were taken into consideration with nearing irregulated geometry regions. The refinement of the meshed were controlled to by automatic program controlled by the software. In case of the irregular structure of the shapes in the geometry of the model, the relevance center were considered to be coarse with smooth and transition rate. As per the geometry dimensions the optimum edge length is taken as 1.16 mm. As seen in below figures, the meshes are refined finitely when approaching curvatures. There are several types of element shapes which are further divided into various classes depending on their use. Major use of tetrahedral elements is undertaken even along the curvature proximities.

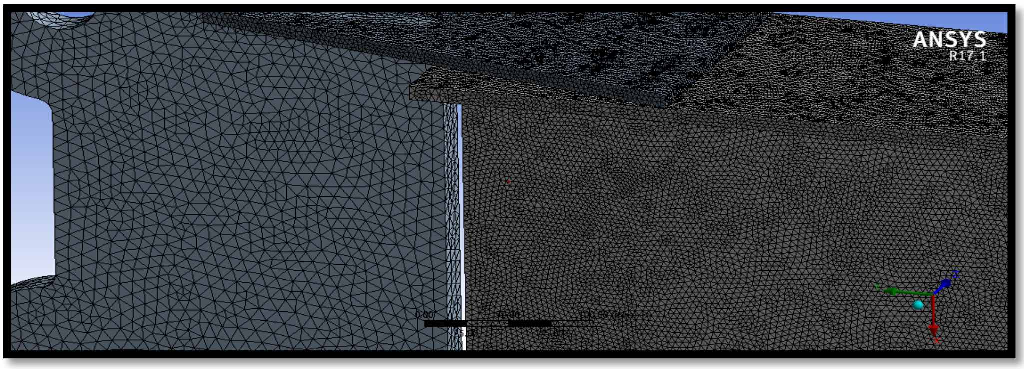


Fig.1.4 Zoomed view of refined meshes for analysis in ANSYS

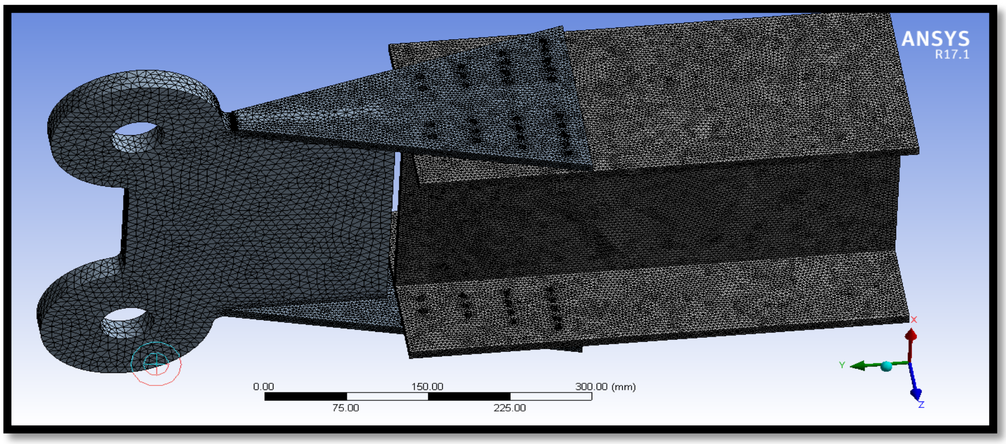


Fig.1.5 Overall view of refined meshes used in assemblies for analysis

The rivets were considered to be frozen part within the model. The reason is, the rivet geometry holds extremely negligible interaction when compared to the other parts of the model. The main agenda of this project is to procure the study and interaction of loads/forces of the lug brackets and I-spar model only. Hence the rivets behavior is neglected within the shed of our research.

4. Incorporating Boundary Conditions

The tool used for FEA is ANSYS Workbench 17.1 which is used in post meshing done at previous meshing steps. Since ANSYS was versatile for our use, the purpose of our project was utilized well versed to our requirements. In the venture of the project modal analysis is also performed to find the behavior of the model under natural frequencies in various modal frequency. The behavior is later compared for numerous activities of the model under external loads and frequencies.

Static structural analysis along with fatigue test were given internal connection in the verge of analysis. Results obtained from static structural results were taken for fatigue test performance. Large deformation was pre-determined and hence this function was switched on prior to analysis.

The magnitude of the load is + 90585 N which is acted towards the negative or positive z axis. This is decision is taken by referring the previous data from journal and research papers. The weight of wing is always acted towards the gravity, they cause the forces to act downwards.

As the model is comply symmetric in shape, the force of application direction can be chosen in either of the side i.e., +/- values, which would still result as same solution only. The components of force if taken along x-axis is taken as (x,y,z) : (90585, 0, 0) N. The application of force is applied to the end cross-section of the I-spar beam.

The bending load acting at the root of the bracket is calculated for '6g' load, with FOS 1.5 which creates a load of 90585 N. It is introduced at one end of the spar beam in downward direction as maximum lift is generated in the wings during take-off. This load will essentially create the required bending moment at the root of the bracket where wing and fuselage will be attached. Since the geometry of the model is symmetric in nature application of positive or negative magnitude will still yield in same results.

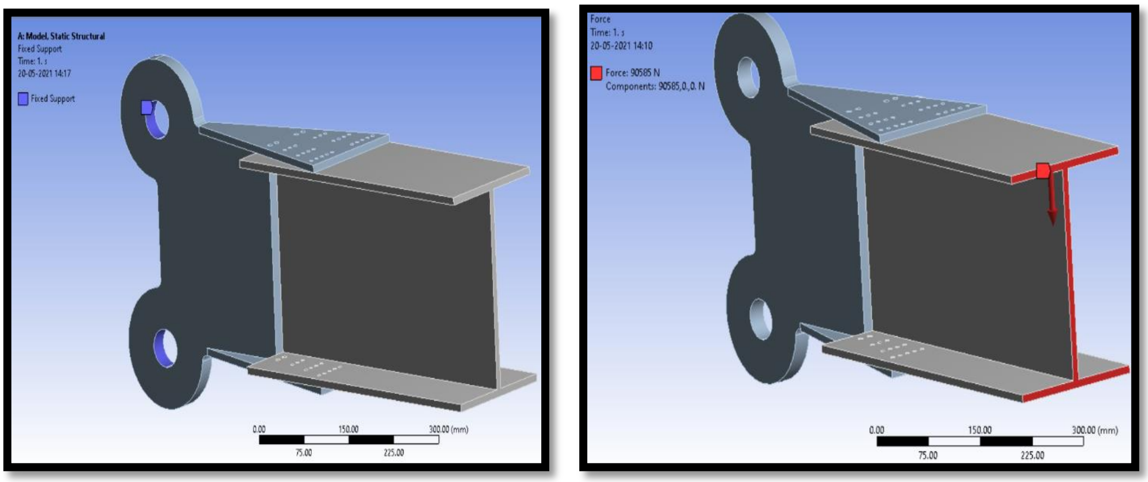


Fig.1.6 Representation Load vector in Assembly section in I-Spar beam (Left Image) & fixed supports in the pin holes of Lug-bracket (Right Image)

The application of supports is provided in the semi-circled inner perimeter surface of the lug brackets. Rest of the contacts regions are entitled to be automatic contact constrained.

The solution expected from FEA tools were Total deformation, Equivalent Elastic Strain, Equivalent strain, Equivalent (Von-Mises) stress, Maximum Principal Stress, Minimum Principal Stress, Maximum Principal Elastic Strain, Minimum Principal Elastic Strain, Directional Deformation, Fatigue Test (Life, Damage, Safety Factor) and Modal Analysis (Total deformation).

Table.1.3 Representing load directions along axis

Model	Load application direction from axis		
	X	Y	Z
Ideal Model A	✓	✓	✓

IRJMETS

e-ISSN: 2582-5208
International Research Journal of Modernization in Engineering Technology and Science
Volume:03/Issue:06/June-2021 Impact Factor- 5.354 www.irjmets.com

Ideal Model B	✓	✓	✓
Custom Model A	✓	-	-
Custom Model B	✓	-	-

5. Inputting data for Fatigue Analysis Test

The fatigue property details are listed below, with possessing zero mean stress values as consideration in this project. The fatigue property was taken from alternating stress sections listed in the research paper and journal papers.

Table. 1.4 Fatigue Tests Material property used for this research study for Al 2024 T351 & Alloy Steel AISI4340

Al 2024 T351	
Cycles	Alternating Stresses (Pa)
50000	1.40E+08
100000	1.27E+08
150000	1.20E+08
200000	1.17E+08
250000	1.10E+08

Alloy Steel AISI 4340	
Cycles	Alternating Stresses (Pa)
10000	3.50E+08
100000	2.50E+08
1000000	1.70E+08
10000000	1.30E+08
100000000	3.50E+08

Table. 1.5 Fatigue Tests Material property used for this research study for Aluminium 7075 T6 & Ti6Al 4V

Aluminum 7075 T6	
Cycles	Alternating Stresses (Pa)
10000	3.50E+08
100000	2.50E+08
1000000	1.70E+08
10000000	1.30E+08

Ti 6Al 4V	
Cycles	Alternating Stresses (Pa)
10000	9.70E+08
100000	8.20E+08
1000000	8.00E+08
10000000	7.80E+08
100000000	9.70E+08

VI. RESULTS AND DISCUSSION

Post analysis the results for all the 4 models were retrieved, where all the parameters must be considered. Certain results of the post analysis are listed below representing few categories like total deformation. The values of other parameters are enlisted in the form of tabular columns. Since large amount of data is acquired from this analysis, it has been ensured that those values are systematically arranged in tabular column for better approach on analyzing the results. Only few reference images are represented below, as the obtained images are numerous.

1. Static structural analysis results

Table. 1.6 Data retrieved from FEA post analysis for static structural analysis under custom model

Behaviour	Model			
	Ideal Model A	**Ideal Model B**	**Custom Model A**	**Custom Model B**
Total Deformation (mm)	4.7	5.27	23.7	29.3
Equivalent Elastic Strain	0.0045	0.0055	0.013	0.014
Equivalent (Von-Mises) stress	871	627.8	954.4	1912
Maximum Principal Stress (MPa)	491	469.7	1890	2036
Minimum Principal Stress (MPa)	220	68	214	337.6
Maximum Principal Elastic Strain	0.0035	0.005	0.01	0.025
Minimum Principal Elastic Strain	1.80E-06	5.80E-06	4.81E-05	1.60E-05

Table.1.7 Data retrieved for individual axis load application for Ideal Model A & B static structural analysis

Behaviour	Ideal Model A			Ideal Model B		
	x	**y**	**z**	**x**	**y**	**Z**
Total Deformation (mm)	4.7	0.127	93.6	5.27	0.16	144.8
Equivalent Elastic Strain	0.0045	0.00005	0.039	0.0055	0.0009	0.069
Equivalent (Von- Mises) stress	871	109.07	7930	627.8	110	7921.2
Maximum Principal Stress (MPa)	491	30.4	8035	469.7	29	8262.8
Minimum Principal Stress (MPa)	220	7.3	924	68	8.7	1509
Maximum Principal Elastic Strain	0.0035	1.00E-04	0.03	0.005	0.0003	0.07

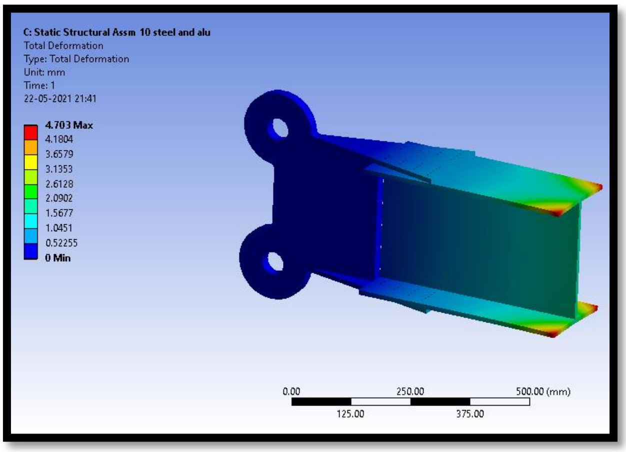


Fig.1.7 Ideal Model A (z direction load)

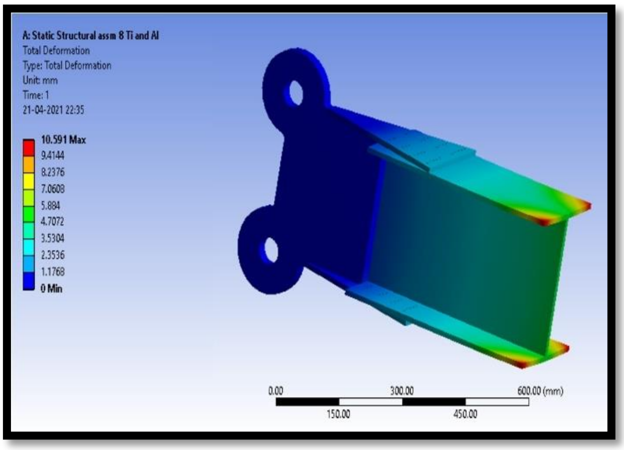


Fig.1.8 Custom Model A (z direction load)

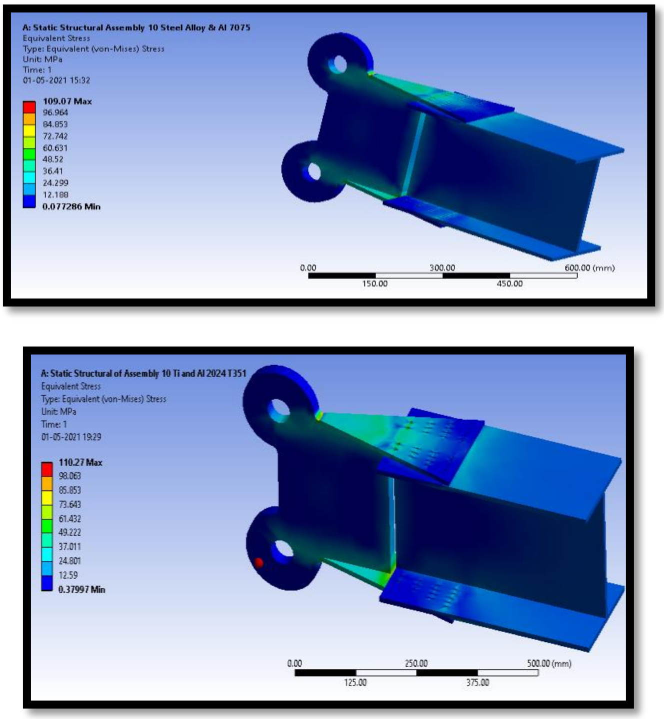


Fig.1.9 y directional of Ideal Model A (left image) & Ideal Model B (right image) representing the Von-Mises stresses

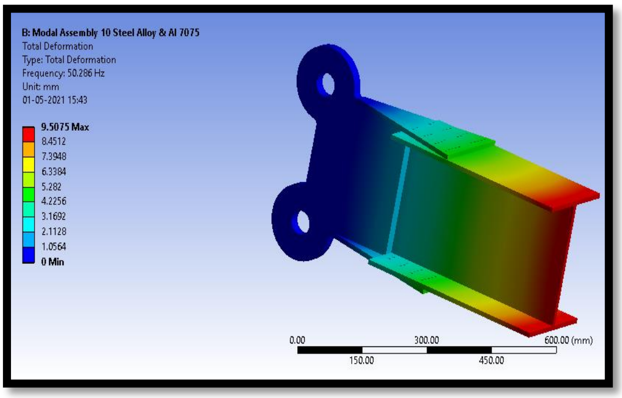
Table. 1.8 Fatigue Analysis Test data for the Ideal and custom models

Fatigue Test Data	Model			
	Ideal Model A	**Ideal Model B**	**Custom Model A**	**Custom Model B**
Life	1.00E+06	1.00E+07	1.00E+07	1.00E+07
Damage	2.90E+05	1.00E+32	1.00E+32	1.00E+32
Safety Factor	15	15	15	15

Table. 1.9 Fatigue test data retrieved for Ideal Model A & B along x,y and z directions

Fatigue Test Behaviour	Ideal Model A			Ideal Model B		
	x	**y**	**z**	**x**	**y**	**z**
Life	1.00E+06	1.00E+06	1.00E+06	1.00E+07	1.00E+07	1.00E+07
Damage	2.90E+05	3.88E+03	1.00E+32	1.00E+32	3.33E+03	1.00E+32
Safety Factor	15	15	15	15	15	15

2. Modal Analysis results



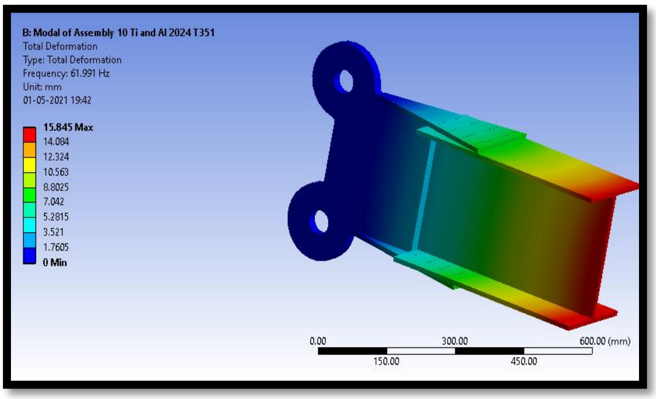


Fig. 1.10 Modal analysis of total deformation of Ideal Model B

Table. 1.10 Data retrieved for individual axis load application for Ideal Model A & B under modal analysis

Behaviour		Ideal model A			Ideal Model B		
		Direction of load along the coordinates					
		x	**y**	**z**	**x**	**y**	**z**
Modal Total Deformation (mm)	**max**	19.5	9.5	9.5	36.1	15.9	15.6
	min	0	0	0	0	0	0

Mode 1 (Hz)	50	46	50.2	61.8	54	63.9
Mode 2 (Hz)	101	98	101	114.5	111.2	117.7
Mode 3 (Hz)	278.7	275	276	291.3	290.7	294
Mode 4 (Hz)	394	390	386	448	446	443
Mode 5 (Hz)	429.9	423	426	453	454	456
Mode 6 (Hz)	471	463	464	482	482.6	483

Table. 1.11 Data retrieved from FEA post analysis for modal analysis

Modal Behaviours	**Ideal Model A**	**Ideal Model B**	**Custom Model A**	**Custom Model B**
Modal Total Deformation	19.5	36.1	17.664	17.6
Mode 1 (Hz)	50	61.8	67.2	68
Mode 2 (Hz)	101	114.5	115	116
Mode 3 (Hz)	278.7	291.3	250	249
Mode 4 (Hz)	394	448	394	386
Mode 5 (Hz)	429.9	453	405.8	407
Mode 6 (Hz)	471	482	465	466

3. Probe values from post analysis for individual parts

Table. 1.12 Probe values found on Lug bracket and I-spar for Ideal Model A

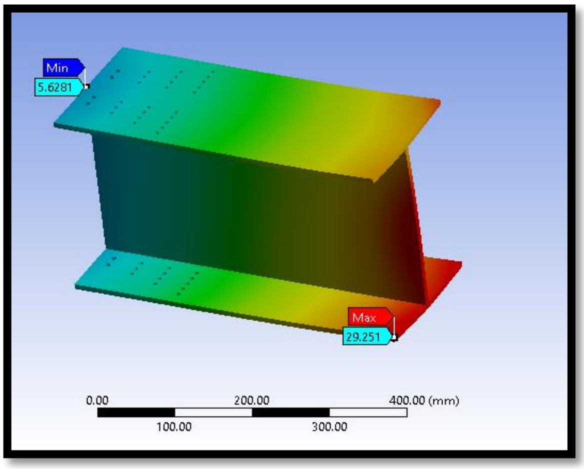
IDEAL MODEL A				
Magnitude	**Parts**	**Force Direction Axis**	**Total Deformation (mm)**	**Maximum Principal Stress (MPa)**
Max	I-Spar Beam	X	4.6	415
Min			0.3	-31.9
Max	Lug Bracket		0.85	490.2
Min			1.60E-04	-77
Max	I-Spar Beam	Y	0.12	13
Min			4.00E-02	-5.5
Max	Lug Bracket		6.00E-02	-29
Min			0	-10
Max	I-Spar Beam	Z	105.3	453
Min			25	-62
Max	Lug Bracket		52	6356
Min			1.00E-03	-911

Table. 1.13 Probe values found on Lug bracket and I-spar for Ideal Model B

Magnitude	IDEAL MODEL B			
	Part	Force Direction Axis	Total Deformation (mm)	Maximum Principal Stress (MPa)
Max	I-Spar Beam	X	5.2	415
Min			0.49	-40.3
Max	Lug Bracket		1.16	469
Min			3.00E-03	-89
Max	I-Spar Beam	Y	0.16	15.4
Min			0	9.80E-03
Max	Lug Bracket		9.50E-05	29
Min			5.00E-05	-16
Max	I-Spar Beam	Z	145	333
Min			32	29
Max	Lug Bracket		72	8262
Min			0	-1596

Table. 1.14 Probe values found on Lug bracket and I-spar for Custom Model A

CUSTOM MODEL A				
Magnitude				
	Parts	Force Direction Axis	Total Deformation (mm)	Maximum Principal Stress (MPa)
Max	I-Spar Beam	X	23.6	747.7
Min			6.5	-19.6
Max	Lug Bracket		13.6	1887.7
Min			0.1	-317.5



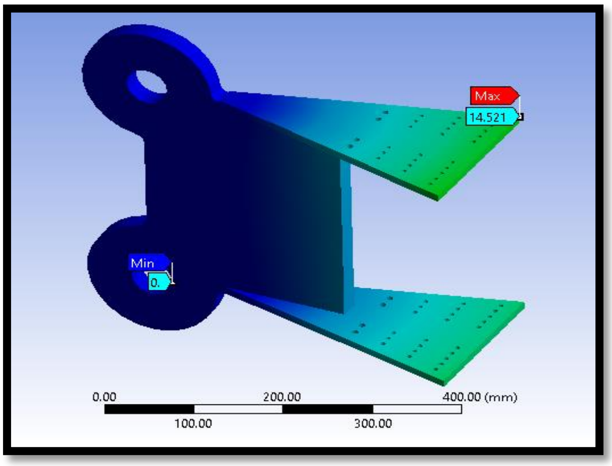


Fig. 1.11 (Left image) Lug Bracket of Custom Model B showing the probe values for total deformation and (Right Image) I-spar of Custom Model B showing the probe values for total deformation

Table1.15 Probe values found on Lug bracket and I-spar for Custom Model B

CUSTOM MODEL B				
Magnitude	**Part**	**Force Direction Axis**	**Total Deformation (mm)**	**Maximum Principal Stress (MPa)**
Max	I-Spar Beam	X	29	758
Min			5.6	95
Max	Lug Bracket		14.5	2020.6
Min			0	-179.5

VII. CONCLUSION

Upon conducting analysis on various customized samples and ideal models, we have concluded by stating following conclusions:

- The Custom Model B has performed well i.e., the 12 mm thickness model possessing Ti-6AL-4V and Aluminum 7075 T6 in Lug brackets and I-Spar respectively have yielded higher deformable rate i.e., 29 mm with 758 MPa Max. Principal stress with significantly comparing ideal model A & B where for lesser deformation they yield higher stress values.
- Ideal Model A & B holds the least Equivalent (Von- Moises) stress when compared amongst the other 2 custom model. As these data are vital in characterizing the behavior of models.
- Both the customized models i.e., Custom Model A & B hold significantly lesser deformation under its natural frequency modes compared to the Ideal Model A & B. Our Custom Model A & B deforms merely 50% of the Ideal Model A & B.
- The customized Model possessed much higher flexible rate compared to the Ideal Models, by maintaining the values well within the yield limits of the respective assigned materials

FUTURE SCOPE

- The optimization in topology of the lug attachment bracket can be done, by reducing the material presence around non-stressed zone region.
- A mathematical model of the same can be modulated to cross verify the analytical results from FEA results.
- Heat flow analysis of the Lug bracket and I-spar can be performed as the aircraft is always operated under varying temperatures (sub-zero temperatures) than the ideal performance tests.
- Use of advanced carbon-carbon composite material may further reduce the overall weight and would result in higher stiffness.
- Dynamic analysis of the I-spar and bracket can be carried out by providing periodic changes in.

ACKNOWLEDGEMENTS

The author gratefully acknowledges and would like to thank to Siva S, PG Scholar, M.Tech (Machine Design), REVA University for aiding in analysis part and important advices that greatly improved the manuscript.

VIII. REFERENCES

[1] Sumanth M, et. al., Comparative analysis of aircraft wing fuselage lug attachment bracket, International Journal for Technological Research in Engineering Volume 5, Issue 11, July-2018

[2] Shambu Kumar, Simulation of Wing-Fuselage Attachment Bracket Lug for Fighter Aircraft, International Journal for Ignited Minds (IJIMINDS) , Volume: 04 Issue: 11, Nov-2017

[3] Tarun Kumar B, Stress Analysis for Wing Attachment Brackets, International Journal of Engineering Research & Technology (IJERT), Vol. 5 Issue 05, May-2016

[4] Aleksandar Grbovića, Fatigue crack paths in light aircraft wing spars", International Journal of Fatigue, 123 (2019) 96–104, ELSVIER"

[5] Khalid Eldwaiba, et. al., Design of Wing Spar Cross Section for Optimum Fatigue Life, Procedia Structural Integrity, ELSEVIER

[6] Amith Kumar S, et. al., Static Structural Linear Analysis of Fuselage Lug Joint Bracket for a Transport Aircraft with Mid Wing Configuration, International Research Journal of Engineering and Technology (IRJET), Volume: 07 Issue: 11 | Nov 2020.

[7] Dr. S. El, Angovan, et. al., Design and Analysis Of Aircraft Wing Spar With Different Materials Using Ansys, Dr. International Journal of Mechanical and Production Engineering Research and Development (IJMPERD), Vol. 9, Issue 3, Jun 2019, 613-618

[8] Ajith V S, et. al., Study of Optimal Design of Spar Beam for the Wing of an Aircraft, IJEDR, Volume 5, Issue 3

[9] Harish E.R.M, et. al., Stress Analysis for Wing Attachment Bracket of a six-seater Transport Airframe Structure" International Journal of Innovative Research in Science, Engineering and Technology. Vol. 2, Issue 7, July 2013.