International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 04 Issue: 08 | Aug -2017 www.irjet.net p-ISSN: 2395-0072

Design and Analysis of Lug Joint in an Airframe Structure Using Finite Element Method

C.V.Rama kishan[1], A.Mahaboob Basha[2]

[1]PG student,department of mechanical engineering,Chiranjeevi Reddy Institute of Engineering and Technology,Anantapur,AP-515001
[2]Assistant professor,department of mechanical engineering, Chiranjeevi Reddy Institute of Engineering and Technology,Anantapur,AP-515001

---***---

Abstract - *Lugs are joint type elements most widely used as structural supports for pin connections type assemblies. Lug and pin joints have been designed based on theoretical strength of materials models and experimental data developed in the 1950's. With the increasing technology in mathematics usage of numerical methods like finite element analysis (FEA) code components can be tested virtually. It is important to determine whether the results obtained from Finite element analysis with the standard theoretical acceptable values. This project deals with the design and analysis of a typical lug joint representative of an airframe structure applications. The design will provide safety against a) Lug failure, b) Pin failure. The types of loadings to be considered here is axial load. Aircraft design practices will be used for calculations. Simplified geometry was modeled in ANSYS Software. Margin of safety for each lug joint component was calculated using two methods, max peak stress and stress averaged over the contact area. Using peak stress was very conservative and predicted margins were much less than those calculated from the theoretical calculations.*

Key Words: FEA, lugs, pin, margin of safety, Vonmises stress, ANSYS.

1. INTRODUCTION

Lugs are connector type elements used as structural supports for pin connections. A lug, also known as a lifting lug is essentially a plate with a hole in it where the hole is sized to fit a clevis pin. Lugs are used in combination with clevis pins to transmit load between different mechanical components. In olden days prior to the 1950's, lugs were overdesigned as weight, cost and space were not design driving factors for joints. With the reducing of weight, cost, feasibility, availability and space requirements in the aerospace industry, a more accurate precise method of lug analysis was required. Analysis of a lug is deceptively complex since there are several simultaneous, interacting failure modes. These failure modes are associated with different areas of the lug

Typical position of lug joint in aero structure is shown in figure 1

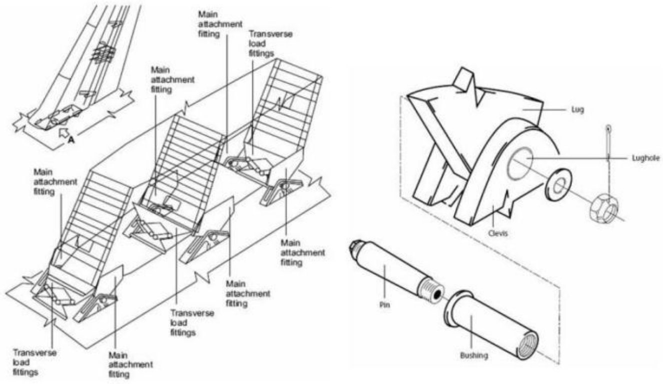


Figure:1 Vertical Tail to Fuselage Attachment Points and Associated Lug Geometry

1.1 Failure modes of a Lug

The failure modes for the lug are listed below. The numbers correspond with the labeled sections from the figure 2:

1. Tension failure across the net section
2. Shear failure along two planes
3. Bearing failure
4. Hoop tension failure / fracture on single plane
5. Out of plane buckling ("dishing") -- (not shown in the figure)

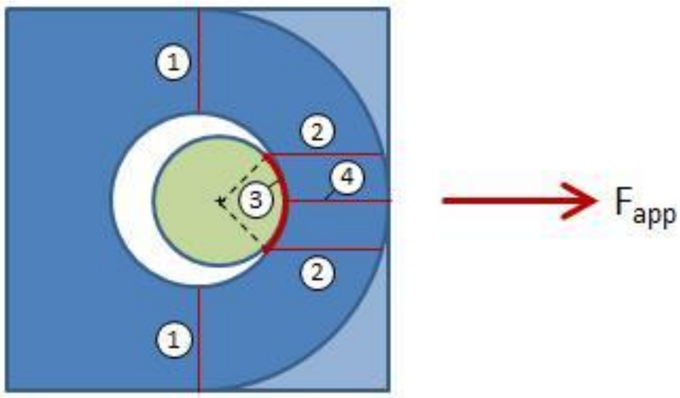


Figure:2 Failure modes

2. LUG MODEL

A simple double shear joint, show in Figure 3, was used for finite element model & analysis. Geometry is modeled using ANSYS design modeler.

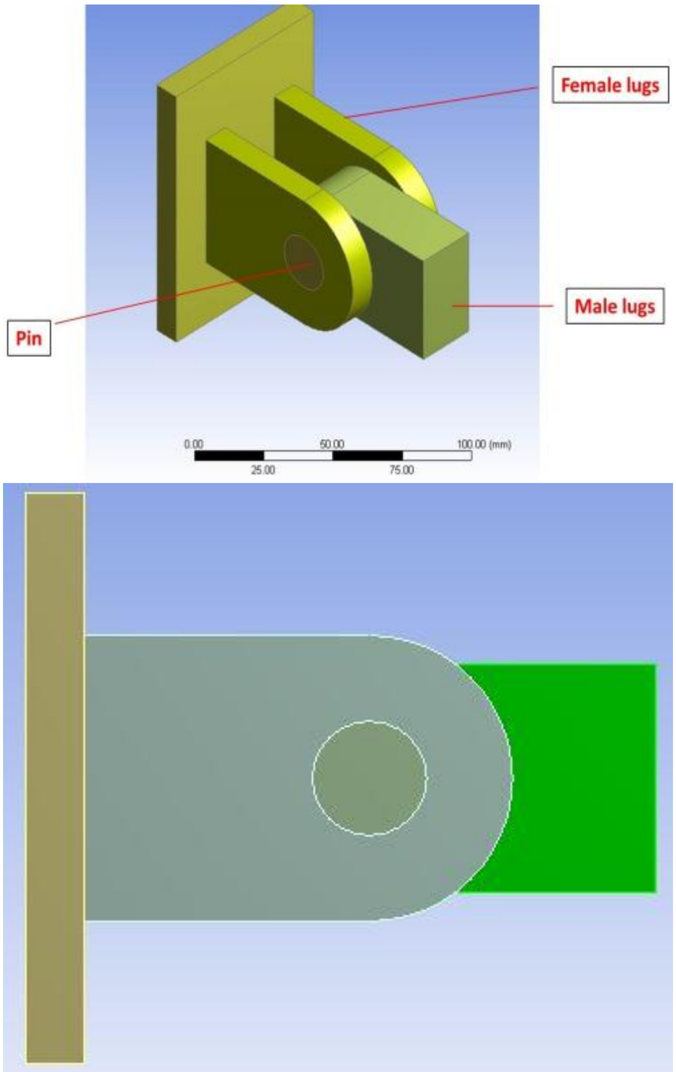


Figure:3 simplified Double shear Lug joint

Geometry details:

Thickness of female lug is: 10mm

Thickness of male lug is: 23mm

Width of lug is: 50mm, Pin diameter is: 20mm

3. MATERIALS FOR LUG & PIN

Most commonly used materials for lug joins are Aluminum alloys and nickel alloys. For pin component widely used materials are steel alloys. This project shows the comparison between Inconel and Aluminum alloy material for the applied loading conditions. The following table shows the mechanical properties of materials.

Component	Material	Density(kg\m3)	Young's modulus(Mpa)	Poission's ratio	Yield limit(Mpa) , Se	Ultimate Limit(Mpa), Su
Lug	Inconel718	8220	2.08E+05	0.284	725	1035
	2024T351AL	2780	7.31E+04	0.33	324	495
Pin	4130 Steel	7850	2.05E+05	0.3	435	670

Table -1: Material data

The chemical composition of Inconel and 20243T51Al is listed in the following table.

Inconel718		20243T51AL	
Element	Percentage	Element	Percentage
Carbon	0.08 max	Iron, Fe	97.03 – 98.22
Manganese	0.35 max	Chromium, Cr	0.80 – 1.10
Phosphorus	0.015 max	Manganese, Mn	0.40 – 0.60
Sulfur	0.015 max	Carbon, C	0.280 – 0.330
Silicon	0.35 max	Silicon, Si	0.15 – 0.30
Chromium	17-21	Molybdenum, Mo	0.15 – 0.25
Nickel	50-55	Sulfur, S	0.04
Molybdenum	2.80-3.30	Phosphorous, P	0.035
Columbium	4.75-5.50		
Titanium	0.65-1.15		
Aluminum	0.20-0.80		
Cobalt	1.00 max		
Boron	0.006 max		
Copper	0.30 max		
Tantalum	0.05 max		
Iron	Balance		

Table -2: Material composition data

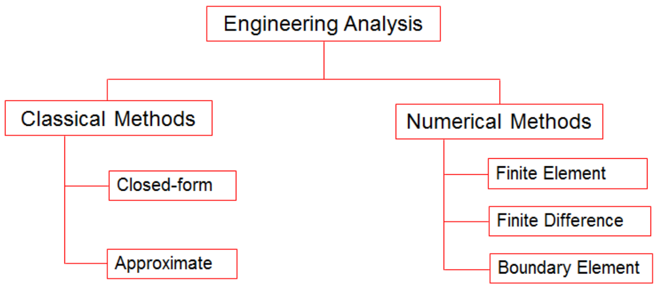
4. FINITE ELEMENT ANALYSIS OF A LUG JOINT

There are mainly 2 methods for solving engineering situation of a component to find its strength and safety.

4.1 Classical Methods:

- Closed-form solutions are available for simple problems such as bending of beams and torsion of prismatic bars
- Approximate methods using series solutions to governing differential equations are used to analyze more complex structures such as plates and shells

– The classical methods can only be used for structural problems with relatively simple geometry, loading, and boundary conditions



4.2 Numerical Methods:

Boundary Element Method: Solves the governing differential equation for the problem with integral equations over the boundary of the domain. Only the boundary surface is meshed with elements.

Finite Difference Method: Replaces governing differential equations and boundary conditions with corresponding algebraic finite difference equations

Finite Element Method (FEM)

- Capable of solving large, complex problems with general geometry, loading, and boundary conditions
- Increasingly becoming the primary analysis tool for designers and analysts
- The Finite Element Method is also known as the Matrix Method of Structural Analysis in the literature because it uses matrix algebra to solve the system of simultaneous equations
- The Finite Element Method (FEM) is a numerical approximation method. It is a method of investigating the behavior of complex structures by breaking them down into smaller, simpler pieces.
- These smaller pieces of structure are called (finite) elements. The elements are connected to each other at nodes.

Why is FEA needed?

FEA is needed for the following reasons --

- To reduce the amount of prototype testing
  – Computer simulation allows multiple "what-if" scenarios to be tested quickly and effectively.
- To simulate designs that are not suitable for prototype testing
  – Example: Surgical implants, such as an artificial knee etc...
- The bottom line:
  – Cost savings.
  – Time savings... reduce time to market!
  – Create more reliable, better-quality designs.

4.3 Introduction to ANSYS

ANSYS stands for analysis systems developed by ANSYS Inc.it is one of the most popular numerical tool for structural, thermal and fluid dynamics analysis. 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 process, enabling the best CAE results. ANSYS Workbench is comprised of five modules:

- Simulation for performing structural and thermal analyses using the ANSYS solver.
- CFX-Mesh for generating a CFX-Pre mesh for the CFX-5 solver.
- Design Modeler for creating and modifying CAD geometry to prepare the solid model for use in Simulation or CFX-Mesh.
- DesignXplorer and DesignXplorer VT for investigating the effect of variations input to the response of the system.
- FE Modeler for translating a Nastran mesh for use in ANSYS.

4.4 Meshed model: Ansys Workbench is used to mesh the model using linear solid 185 type element which has the capabilities of taking structural loads and responds for its application of load.

SOLID185 is used for 3-D modeling of solid structures. It is defined by eight nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions. The element has plasticity, hyper elasticity, stress stiffening, creep, large deflection, and large strain capabilities.

SOLID185 Homogeneous Structural Solid Element Description

SOLID185 Structural Solid is suitable for modeling general 3-D solid structures. It allows for prism, tetrahedral, and pyramid degenerations when used in irregular regions. Various element technologies such as B-bar, uniformly reduced integration, and enhanced strains are supported.

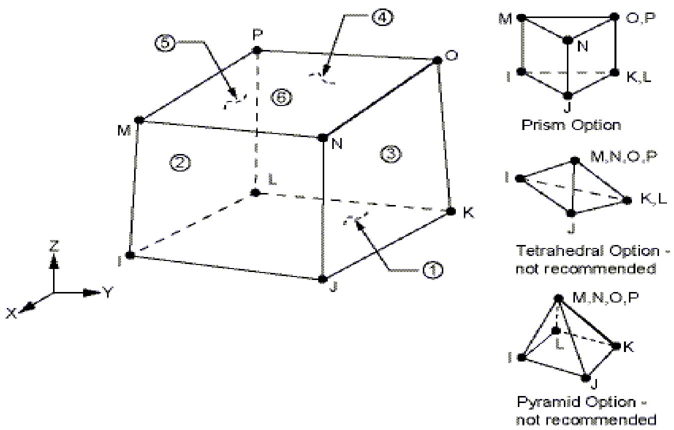


Figure 4: SOLID185 Homogeneous Structural Solid Geometry

Lug and pin are meshed in Ansys workbench and provided contact between pin and lug to calculate the contact status and contact pressure. Figure 5 shows meshed model of lug and pin assembly.

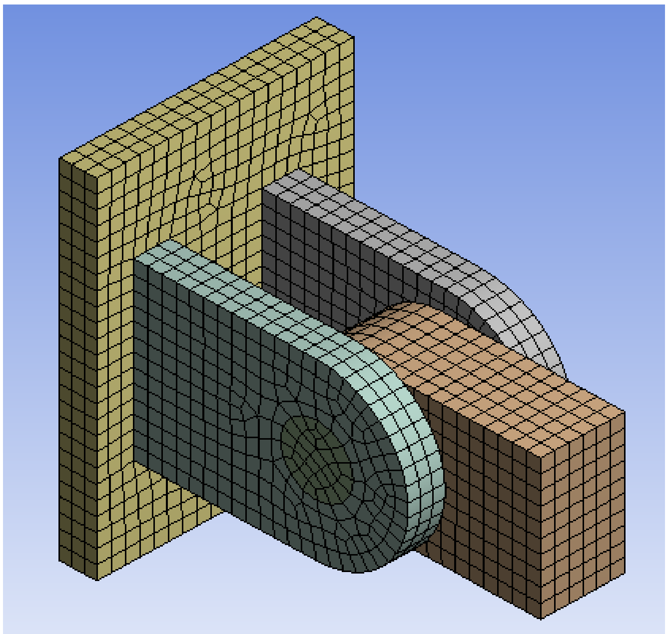


Figure 5: Meshed model

4.5 Boundary & Loading conditions

Typical boundary conditions have been considered for calculating realistic values of lug joint applications. Assumed Female joint is constrained over attached plate body and loading is applied on the male lug surface using force command in Ansys. The following figure shows the location of boundary conditions and loading.50KN is applied on end surface of the male lug to calculate the effect of contact pressure and proof stress. This 50KN load is taken from the data book of Mickel Nue for the current configuration of the geometry. Assuming boundary conditions on the side surface of the plate which is attached to female lugs. The following figure shows location of constraints and loading.

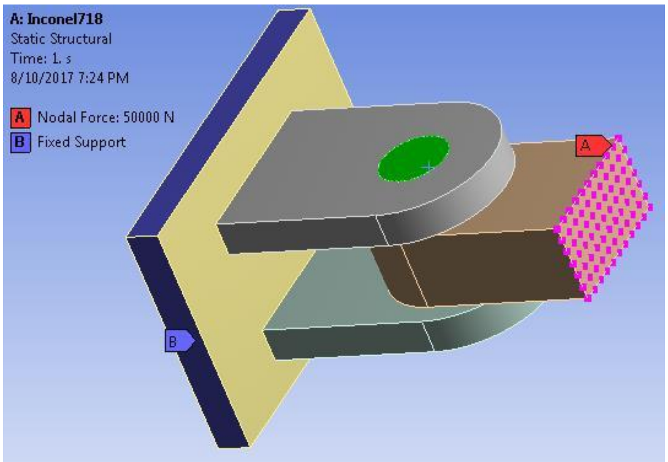


Figure 6: Loads & boundary conditions

Along applied loading and boundary conditions contacts are defined between lug and pin as shown in figure 6 in 3 positions.

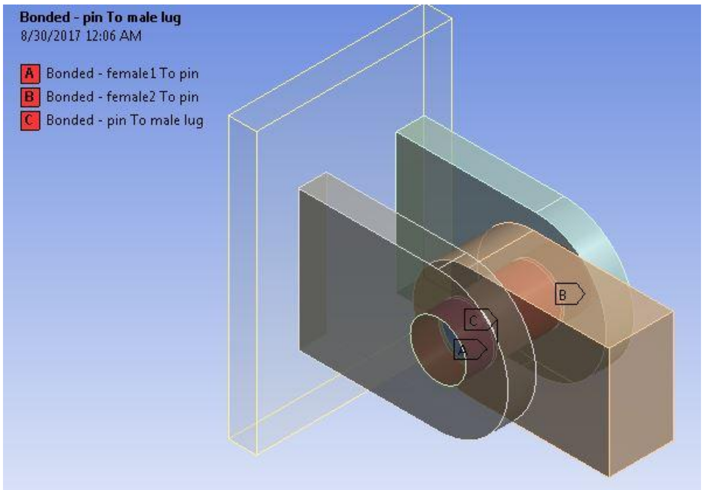


Figure 7: Contact faces

4.5 Solution

Ansys uses matrix inverse method for solving the unknown values like displacements, stress and strains. The behaviour of the structure is obtained by analysing the collective behaviour of the elements.Basic approach is as follows

- A given problem is discretized by dividing the original domain into simply shaped elements. Elements are connected to each other by nodes.
- Each node is capable of moving in six independent directions: three translations and three rotations. These are called the degrees of freedom (DOF) at a node.
- The relationship between an element and its surrounding nodes can be described by the following equation: $[\mathbf{k}]_e \{\mathbf{u}\}_e = \{\mathbf{f}\}_e$
- The elemental stiffness matrix $[k]_e$ is derived from geometry, material properties, and element properties.
- The elemental load vector {f} e describes the forces acting on the element.
- The displacement vector $\{u\}_e$ is the unknown in this equation. It describes how the nodes move as a result of the applied forces.

Elemental Equation $[\mathbf{k}]_e \{\mathbf{u}\}_e = \{\mathbf{f}\}_e$

- Next, the elemental stiffness matrices are assembled into a global stiffness matrix. The loads are also assembled into a global load vector. This results in the following matrix equation for the overall structure:

Global Equation $[\mathbf{K}]\{\mathbf{u}\} = \{\mathbf{F}\}$

- Next, apply the boundary condition to the model (constrain the model). Mathematically, this is achieved by removing rows and columns corresponding to the constrained degrees of freedom from the global matrix equation.

- Finally, the global matrix equation is solved to determine the unknown nodal displacements.

Element strains and stresses are then computed from the nodal displacements.

Contact nonlinear analysis has been performed to get the contact status and pressure between pin and lug geometry.

5. RESULTS & DISCUSSION

From the analysis required results has been considered and requested from ansys for plots. In this paper results are compared between 2 different materials. Deformation results, contact stress, stress results are compared for two materials.

The following table shows the stress, reserve factor, deformation and contact pressure values. Allowable for each material is calculated using the formula minimum of (Se/1.15, Su/1.15).

Inconel718					
Component	Max Vonmises stress(Mpa)	Allowable Stress (Mpa)	Reserve Factor (Rf)	Contact pressure (Mpa)	Deformation (mm)
Female lug	266.38	630	2.37	107.87	0.058
Male lug	241.47	630	2.61		
Pin(steel)	122.78	378	3.08		

Table -3: Results for inconel718

20243T51AL					
Component	Max Vonmises stress	Allowable	Reserve Factor (Rf)	Contact Pressure	Deformation (mm)
Female lug	192.78	281	1.46	85.98	0.51
Male lug	238.91	281	1.18		
Pin(steel)	148.03	378	2.55		

Table -4: Results for 20243T51AL

The following figures show the contour plot of all results on pin and lug geometry for inconel718.

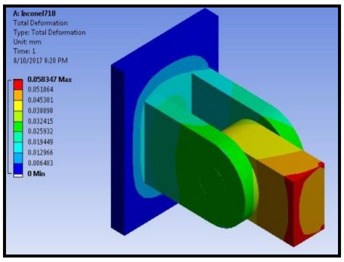


Fig8: Deformation

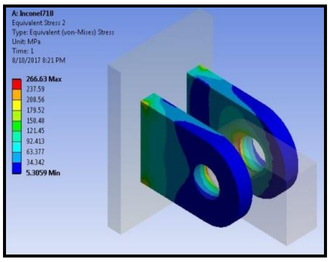


Fig9: Stress-female lug

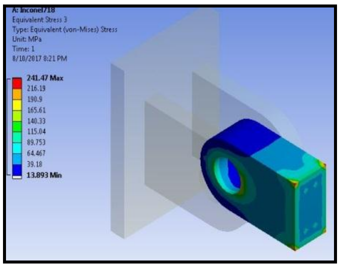


Fig10: Stress-male lug

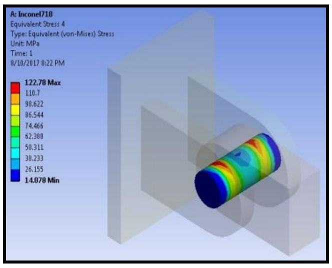


Fig11: Stress-pin

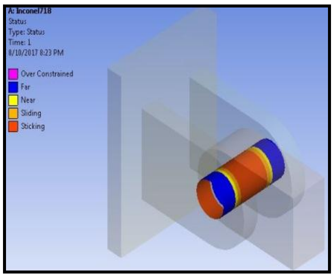


Fig12: contact status

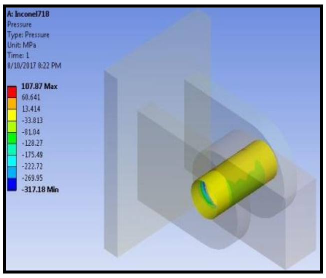


Fig13: Contact pressure

The following figures show the contour plot of all results on pin and lug geometry for 20243T51AL.

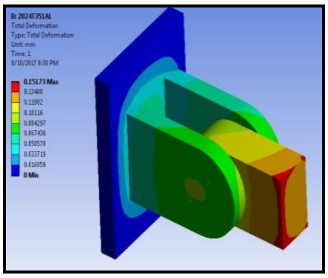


Fig14: Deformation

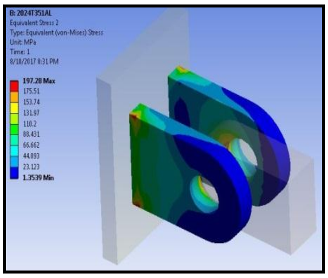


Fig15: Stress-female lug

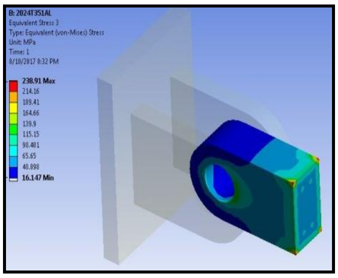


Fig16: Stress-male lug

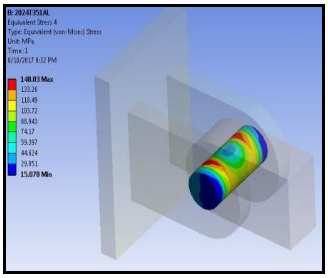


Fig17: Stress-pin

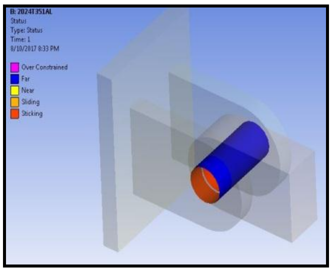


Fig18: contact status

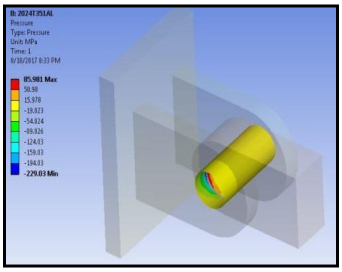


Fig19: Contact pressure

5. CONCLUSIONS

Contact Linear static analysis has been carried out to check the contact pressure, proof stress and deformation of lug and pin interface. The analysis techniques and knowledge presented here is most relevant for the design of lug joints in aero structures. The mechanical design of lug joint has remained relatively unchanged for even longer. Perhaps there is an opportunity for a total mechanical redesign of lug joint, derived from intentional design and optimization of all design parameters. The results shows a stress development of around 266 Mpa near the left side of the pin hole interface, this is due to contracting contact surface due to axial loading. Contact pressure of 107MPa & 85.98 has been observed on pin and lug interface.

ACKNOWLEDGEMENT

I express my gratitude to **MR. S. MAHABOOB BASHA M.Tech,** Assistant Professor and Head of Mechanical Engineering Department for helping me with his esteemed guidance and valuable solutions to the problems encountered. I thank reverend Principal **Dr. P.ISSAC PRASAD Ph.d** for helping me in many regards throughout the work.I also thank all the mechanical faculties who are helped me directly and indirectly in completion of this project.
I also thank the entire well-wishers whose co-operation and valuable suggestions have helped me for completion of the project successfully.

REFERENCES

[1] Design data hand book by K.mahadevan and k.Balaveera Reddy, 3rd edition 1987.

[2] B.K. Sriranga, Dr.C.σ. Chandrappa, R. Kumar and Dr.P.K. Dash, “Stress Analysis of Wing-Fuselage LugAttachment Bracket of a Transport Aircraft”, International Conference on Challenges and Opportunities in Mechanical

[3] Military Handbook 5H – Metallic Materials and Elements for Aerospace Vehicle Structures, Columbus: Battelle Memorial Institute, 1998