International Journal of Mechanical Engineering and Robotics Research

ISSN 2278 – 0149 www.ijmerr.com
Vol. 2, No. 3, July 2013

Research Paper

STRESS ANALYSIS OF RIVETED LAP JOINT

Suyogkumar W Balbudhe[1]* and S R Zaveri[1]

*Corresponding Author: **Suyogkumar W Balbudhe,** ✉ Suyog24@yahoo.co.in

This project deals with the stress analysis of riveted lap joints. The present work involves the appropriate configuration and characterization of these joints for maximum utilization. By using finite element method, stress and fracture analyses are carried out under both the residual stress field and external tensile loading. Using a two step simulation, riveting process and subsequent tensile loading of the lap joint are simulated to determine the residual and overall stress state. Residual stress state due to riveting is obtained by interference and clamping misfit method. By employing different interference and clamping misfit values, the effects of riveting process parameters on stress state are examined. Two cracks namely the semi elliptical surface crack at faying surfaces of plates and the quarter elliptical corner crack at rivet hole are the most widely observed crack types in riveted joints. Fracture analysis of cracked riveted joints is carried out by introducing these two crack types to the outer plate at a plane perpendicular to the loading. The finite element technique was used throughout the analysis of present work. The present work showed that riveted joints are superior in strengthening to the riveted joints. The riveted joint seems to strengthen and balance the stress and distributed uniformly. This improves the efficiency and life time of the riveted joints. Modeling is done by CATIA V 16.0 and analysis of riveted lap joint can be done by using ANSYS (Workbench) with a version of 14.0. FEM result can be analyzed with mathematical data.

Keywords: Riveted lap joint, Ansys workbench V14.0, Finite element technique riveted lap joint, Ansys workbench V14.0, Finite element technique

INTRODUCTION

Manufacturing large and complex structures is usually possible only when they are composed of assemblies of smaller parts joined together by variety of joining techniques since most products are impossible to be produced as a single piece. Manufacturing components and then joining them into a single product is easier and less expensive than manufacturing the whole product at once. In order to ensure the manufacturability, and reduce the overall manufacturing cost, certain fastening and joining methods should be utilized. Mechanical fasteners can be described as devices that

[1] Mechanical Department, Rajiv Gandhi College of Engineering, Research and Technology, Chandrapur (M.S.).

mechanically join two or more objects of an assembly with desired permanence, stability, and strength. Mechanical fasteners offer several options for joining and fastening mechanical components together. Mechanical fastening methods can be categorized into two main types: permanent (welding, bonding, riveting, etc.), and detachable joints (bolt, screw, pin, etc.). Selection of appropriate method among these alternatives should be based on permanence, cost and strength of the fastener.

INTRODUCTION TO RIVET

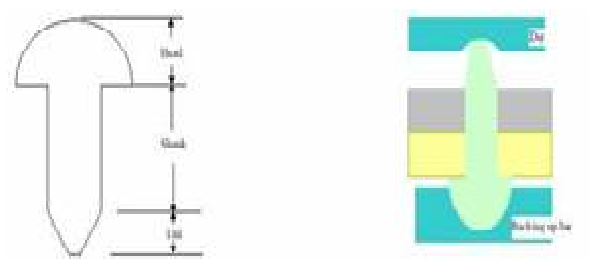
Rivets are permanent, non-threaded, one-piece fasteners that join parts together by fitting through a pre-drilled hole and deforming the head by mechanically upsetting from one end. Rivets are the most widely used mechanical fasteners especially in aircraft fuselage structures. Hundreds of thousands of rivets are utilized in the construction and assembly of a large aircraft. Solid rivet with universal head is one of the most widely used rivet type in aircraft fuselage manufacturing and repairing processes.

A riveted joint, in larger quantities is sometimes cheaper than the other options but it requires higher skill levels and more access to both sides of the joint A rivet is a cylindrical body called a shank with a head. A hot rivet is inserted into a hole passing through two clamped plates to be attached and the heads supported whilst a head is formed on the other end of the shank using a hammer or a special shaped tool. The plates are thus permanently attached.

Rivet

A Rivet is a short cylindrical rod having a head and a tapered tail. The main body of the rivet is called shank (see Figure 1). According to Indian standard specifications rivet heads are of various types.

Figure 1: Riveting Process



Rivets heads for general purposes are specified Figure 1 by Indian standards IS: 2155-1982 (below 12 mm diameter) and IS: 1929-1982 (from 12 mm to 48 mm diameter). Riveting is an operation whereby two plates are joined with the help of a rivet. Adequate mechanical force is applied to make the joint strong and leak proof. Smooth holes are drilled (or punched and reamed) in two plates to be joined and the rivet is inserted. Holding, then, the head by means of a backing up bar as shown in Figure 1, necessary force is applied at the tail end with a die until the tail deforms plastically to the required shape.

Design stresses

The rivet joints are analyzed on the basis of following assumptions:

- Rivets are loaded in shear the load is distributed in proportion to the shear area of the rivets.
- There are no bending or direct stresses in rivets.
- Rivet holes in plate do not weaken the plate in compression.
- Rivets after assembly completely fills the hole.

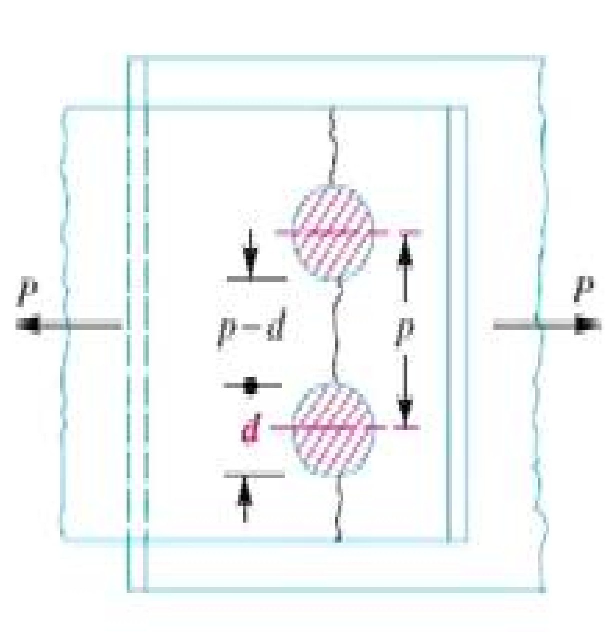
- Friction between the adjacent surfaces does not affect the strength of the joints however the actual stress produced decreases.
- When rivet is subjected to double shear, the shear force is equally distributed between the two areas of shear.

Thus according to the theory the failure of rivet may occur due to any one of the following modes.

Theories of Failures of Rivet Joints

Tearing of the Plate at an Edge: A joint may fail due to tearing of the plate at an edge as shown in below figure. This can be avoided by keeping the margin, $m = 1.5\ d$, where "*d*" is the diameter of the rivet.

Figure 2: Tearing of Plate



Tearing of the Plate Across a Row of Rivet: Tearing resistance required to tear off the plate per pitch length, $Pt = At.\sigma t = (p - d)\ t.\sigma t$

where p = Pitch of the rivets;

d = Diameter of the rivet hole;

t = Thickness of the plate;

σt = Permissible tensile stress for the plate material.

Shearing of Rivet: Thus shear strength is:

$Ps = n\pi/4\ d2\ T\max$—for single shear and

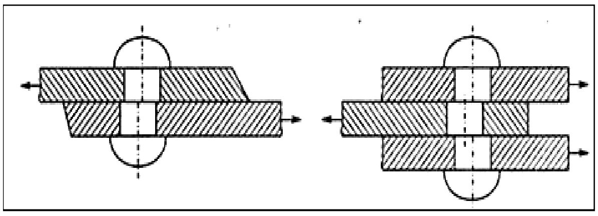
$Ps = 2 \times n\pi/4\ d2\ T\max$—theoretically in double shear and

$Ps = 1.875 \times n\pi/4\ d2\ T$—for double shear, according to Indian boiler regulations

where, $T\max$ = Shear strength of rivet;

n = Number of rivets.

Figure 3: Shearing of Plate



Crushing of the Plate (or) Rivets: The crushing strength is:

$Pc = n\ d\ t\ \sigma c$

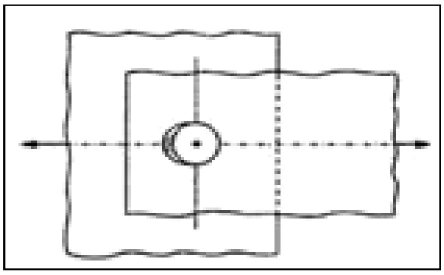
where, σc = Crushing strength of rivet;

n = No. of rivets under crushing;

d = Diameter of rivet = $6.1\ \sqrt{t}$;

t = Thickness of plate

Figure 4: Crushing of Plate



Efficiency of Riveted Joint

$$\eta = \frac{Strength\ of\ the\ Joint\ in\ the\ Weakest\ Mode}{Strength\ of\ the\ Unpunched\ Plate}$$

OVERVIEW OF ANSYS

The ANSYS (workbench) V 16.0 computer program is used for multipurpose finite element program, which may be used for solving several different field. Engineering analyses. An ANSYS (workbench) V 16.0 has been developed, other special capabilities, such as sub structuring, sub modeling, random vibration, free convection fluid analysis, acoustics, magnetic, piezoelectric, couple field analysis and design optimization have been added to the program.

Program Overview

Analyzed of any problem in ANSYS has to go through three main steps. They are, Pre-processor, Solution and Postprocessor. The input of the ANSYS is prepared using preprocessor. The general preprocessor contains powerful solid modeling and mesh generation capabilities, and is also used to define all the other analysis data such as geometric properties like real constants, material properties, constraints, loads, stiffness damping, etc.

General Procedure to Solve and Problem in ANSYS

The ANSYS software has many finite element analysis capabilities ranging from simple, linear, static analysis to a complex, nonlinear, transient dynamic analysis Any problem in ANSYS has to go through the three main steps Build the model, Apply loads and obtain solution Review the results.

Building the Model

In this project model is designed with the help of CATIA V 16.0 and also given the material property in this software by providing the proper constraning.This model developed with three stages first upon developing all the object as a part file single pieces with separate file and then assemble with constrained in assembly, i.e., product environment. Then converted this model into "stp" format to importing the ANSYS (workbench).

Apply Loads and Obtain the Solution

Applying Loads

Boundary conditions and Different loads acting on the model are applied either in the preprocessor as well as contact surface between the object also given. The loads in the ANSYS program are: DOF constraints, Forces, Surface loads, Body loads

Defining the Type of Analysis and Analysis Options

Static Analysis: Used to determine displacements, stresses, etc., under static loading conditions. Both linear and nonlinear static analysis. Nonlinearities can include plasticity, stress stiffening, large deflection, large strain, hyper elasticity, contact surfaces, and creep. Our present analysis is "static Analysis".

Specifying Load Step Options

The following path is used to specify load step options.

Main Menu => Solution => Insert option

Initiating the Solution

The solution to the given problem is initiated by using the following path.

Main Menu => Solution => Solve

Initiating the Results

After the solution any type of result can be seen by following path.

Main Menu => Solution => Insert Results.

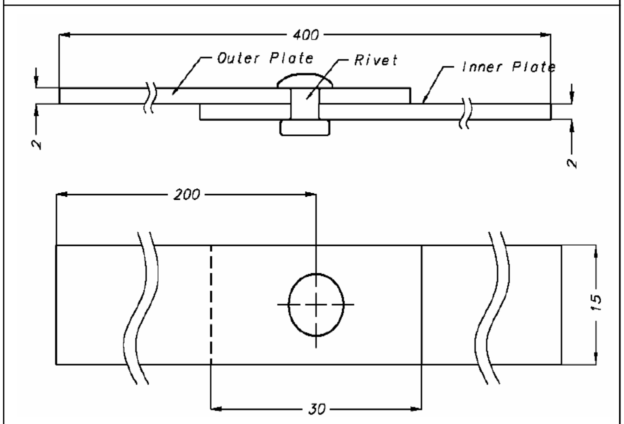
DESCRIPTION OF THE PRESENT WORK

The present work deals with the stress Analysis of riveted lap joint. This is quite commonly used technique for finding the strength of different applications like pressure vessels, aerospace, marine and mostly for leak proof joints like oil tanks, boilers, etc. In this a lap joint of aluminum alloy plate material having 215 mm * 2 mm * 15 mm and a friction factor of 0.2 is over lapped with the other plate having same dimensions and material and are joined by means of a rivet having diameter 4 mm, apply a load of 2058 N on one side and the other end is fixed in the ANSYS (workbench). The stresses on normal riveted joint is compared it with the analytical solution.

Determination of Specifications

The specifications of the present job has taken from the design considerations from the plate thickness as 2 mm and that are specified in the following diagram. Figure 5 shows the configuration, dimensions, constraints and loading conditions. The following assumptions and boundary conditions were considered throughout the idealization process:

Figure 5: Actual Dimension



Finite Element Model with Meshing, Boundary Conditions and Material Property

Figures 6 and 7 shows the CAD model prepared in CATIA also the property of material given in same software. This model meshed in ANSYS for analysis with element size 0.5 mm brick and triangular meshing with fine type.

Figure 6: CATIA Model

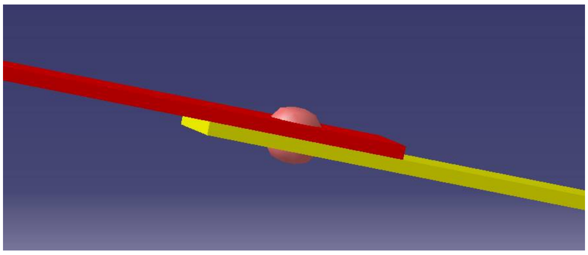


Figure 7: Meshing of CAD Model

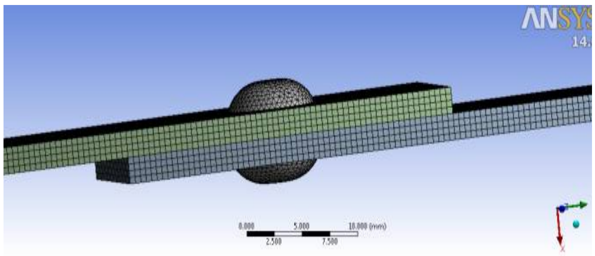


Figure 8: Boundary Condition

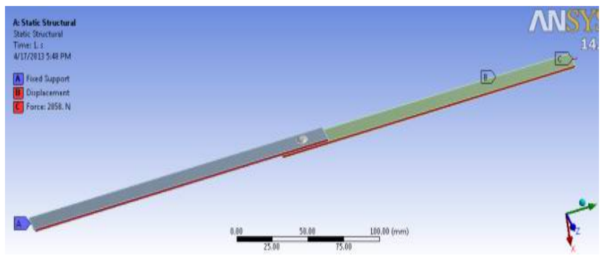


Table 1: Material Properties

Material		Young's Modulus	Poisson's Ratio
Plate	2024-T3	74 GPa	0.33
Rivet	2117-T4	71.7 GPa	0.33

FEM and Analytical Calculation

FEM Analysis

Figure 9: Stress Distribution on Riveted Lap Joint

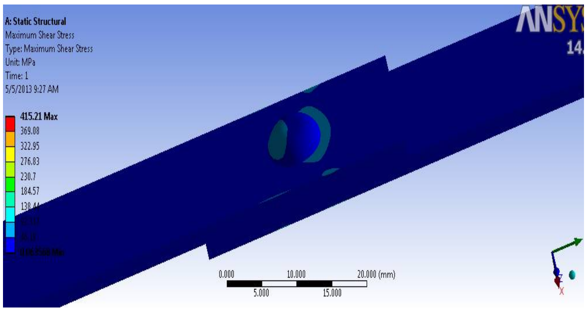


Figure 10: Stress Distribution on Rivet

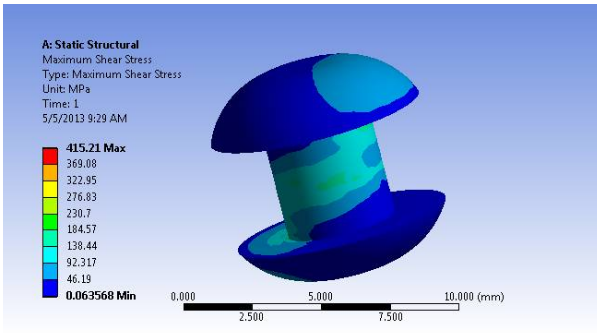
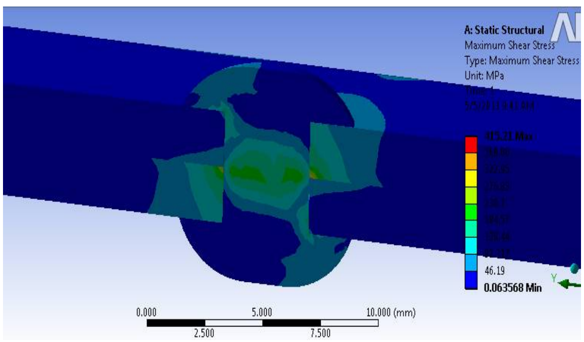


Figure 11: Cut Section Model



Analytical Calculation

- Force applied on the plate (Actually pressure applied on surface then converted into force by calculation)

  P(Pressure) = 68.6 Mpa

  $$\therefore F = P \times A = 68.6 \times 30 = 2058\ N$$

- Crushing stress on the rivet (σ_c)

  $$\sigma_c = \frac{F}{dt} = \frac{2058}{4 \times 2} = 257.25\ Mpa$$

  where, F: Force,

  d: Diameter of rivet and

  t: Thickness of plate

  p: Pitch

- Tearing of the rivet across a row of rivet (σ_t)

  $$\sigma_t = \frac{F}{(p-d)t} = \frac{2058}{(15-4)\times 2} = 93.54\ Mpa$$

- Shearing of rivet (τ_s)

  $$\tau_s = \frac{F}{\frac{\pi d^2}{4}} = \frac{2058}{\frac{\pi}{4}4^2} = 163.77\ Mpa$$

- Maximum Shear Stress

  $$\tau_{max} = \frac{1}{2}\sqrt{\sigma_t^2 + \tau_s^2}$$

  $$= \frac{1}{2}\sqrt{(93.54)^2 + (163.77)^2} = 170.31$$

CONCLUSION

- The results obtained from ANSYS (Workbench) software for the Single riveted lap joints are compared with analytical data it is shown that maximum shear stress with the help of FEM in the range of 138.44 Mpa To 184.57 Mpa and Manual Data of maximum shear stress is 170.31 Mpa both are the nearer it conclude that the result of both are match with each other.

- Finite Element Method is found to be most effective tool for designing mechanical components like riveted lap joints.
- ANSYS can be used for analysis of complex and simple models of different type without any effect on practical and economical issues.

SCOPE FOR THE FUTURE WORK

There is a huge scope for the future work, on the resent topic of stress analysis Riveted lap joint.

- By the consideration the design, and practical with these we can get some more accurate results.
- By considering a particular application and load leads to a scientific research.
- By changing the thickness of plate and thickness of dome.

REFERENCES

1. Harris C E, Piascik R S and Newman C (1999), "A Practical Engineering Approach to Predicting Fatigue Crack Growth in Riveted Lap Joints", *International Conference on Aeronautical Fatigue (ICAF)*, WA, Seattle.
2. Kurmi R S and Gupta J K (2009), *A Text Book of Machine Design*.
3. Silva L F M, Gonçalves J P M, Oliveira F M F and de Castro P M S T (2000), "Multiple Site Damage in Riveted Lap-Joints: Experimental Simulation and Finite Element Prediction", *Int. J. Fatigue*, Vol. 22, pp. 319-338.
4. Tirupathi R Chandrupatla and Ashok D Belegundu (2011), *Introduction to Finite Elements in Engineering*, 4^{th} Edition.