International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 2 Issue 3, March - 2013

Stress Analysis Of Riveted Lap Joint

Suyogkumar W. Balbudhe
Student of M.Tech (CAD/CAM),
Rajiv Gandhi College Of Engineering,Research and Technology,Chandrapur(M.S.)

Prof. S. R. Zaveri,
Associate Professor in Mechanical Department,
Rajiv Gandhi College Of Engineering,Research and Technology,Chandrapur(M.S.)

Abstract

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 bonded 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 V16.0 and analysis of riveted lap joint can be done by using ANSYS (Workbench) with a version of 14.0.

1.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 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.

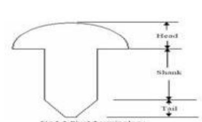
2. Introduction to Rivet

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.

2.1 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 2.1). According to Indian standard specifications rivet heads are of various types. Rivets heads for general purposes are specified



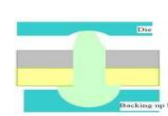


Fig. 2.1

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 fig 1.2, necessary force is applied at the tail end with a die until the tail deforms plastically to the required shape.

2.2 Design stresses

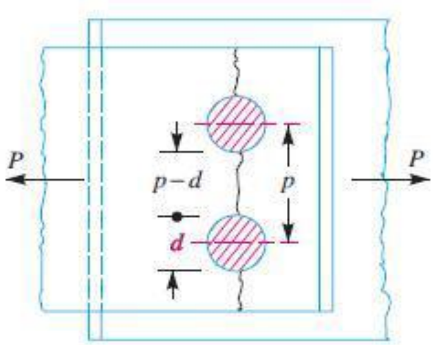
2.2.1 The rivet joints are analyzed on the basis of following assumptions:

I. Rivets are loaded in shear the load is distributed in proportion to the shear area of the rivets
II. There are no bending or direct stresses in rivets
III. Rivet holes in plate do not weaken the plate in compression
IV. Rivets after assembly completely fills the hole
V. Friction between the adjacent surfaces does not affect the strength of the joints however the actual stress produced decreases
VI. 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.

2.2.2 Theories of Failures of Rivet Joints

2.2.2.1.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=1.5d, where "d" is the



diameter of the

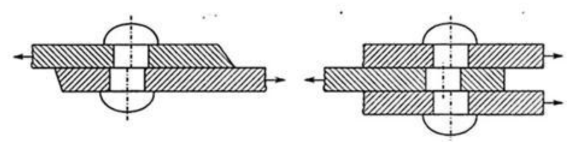
2.Tearing of the plate across a row of rivet:-

Tearing resistance required to tear off the plate per pitch length, $P_t = A_t.\sigma_t = (p-d)\ t.\sigma_t$

Where p = pitch of the rivets;
d = diameter of the rivet hole;
t = thickness of the plate;
σ_t = permissible tensile stress for the plate material.

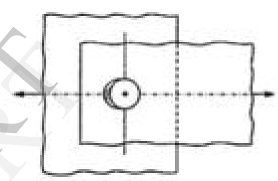
2.2.2.3. Shearing of rivet

Thus shear strength is,
$P_s = n\ \pi/4\ d^2\ T_{max}$ - for single shear and
$P_s = 2 \times n\ \pi/4\ d^2\ T_{max}$ - theoritically in double shear and
$P_s = 1.875 \times n\ \pi/4\ d^2\ T$ - for double shear,according to Indian boiler regulations
Where, T_{max} = Shear strength of rivet;
n = number of rivets.



2.2.2.4. Crushing of the plate (or) rivets :-

The crushing strength is,$P_c = n\ d\ t\ \sigma_c$
Where, σ_c = Crushing strength of rivet;
n = no of rivets under crushing;
d = diameter of rivet = $6.1\sqrt{t}$;
t = thickness of plate



2.2.2.5. Efficiency of riveted joint

$$\eta = \frac{\text{Strength of the joint in the weakest mode}}{\text{Strength of the un punched plate}}$$

3. 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. As 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.

3.1 Program overview

Analysation of any problem in ANSYS has to go through three main steps. They are,
Pre-processor, Solution & 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.,

3.2 General procedure to solve and problem in ansys

The ANSYS software has many finite element analysis capabilities ranging from simple, linear,static

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 2 Issue 3, March - 2013

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.

3.2.1 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 oftware by providing the proper constraning.This model developed with three stages first upon developing all the object as a part file single pieces with with seprate file and then assembal with constrained in assembally i.e. product enviorment. Then converted this model into .stp format to importing the ANSYS (workbench).

3.3 Apply loads and obtain the solution

a) 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

(b) 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".

(c) Specifying load step options The following path is used to specify load step options.

Main Menu=>Solution=>Insert option

(d) Initiating the solution The solution to the given problem is initiated by using the following path. Main Menu=>Solution=>Solve

(e) Initiating the results After the solution any type of result can be seen by following path.

Main Menu=>Solution=>Insert Results.

4. 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 aluminium alloy plate material having 215mm*2mm*15mm 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 4mm, apply a load of 2058 N on one side and the other end is fixed in the ANSYS(workbench). The normal riveted joint one is compared it with the changing the dome shape of rivet into cap type rivet it is found that the stress concentration on the rivet will be minimized in case of cap type rivet also the deflection will de reduced that means strength of rivet increased. We can observe the reduced stress comparison from the figures.

4.1 Determination of Specifications

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

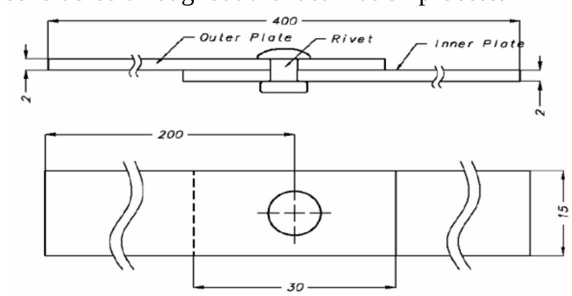


Fig.4.1.1 (Actual Dimension)

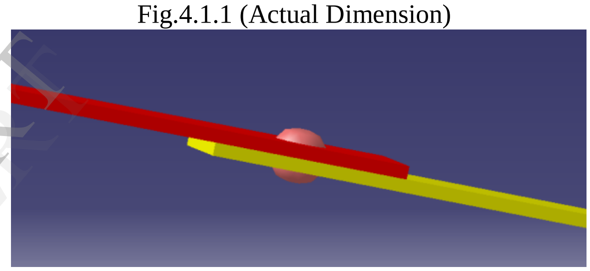


Fig.4.1.2 (CATIA Cap shape Model)

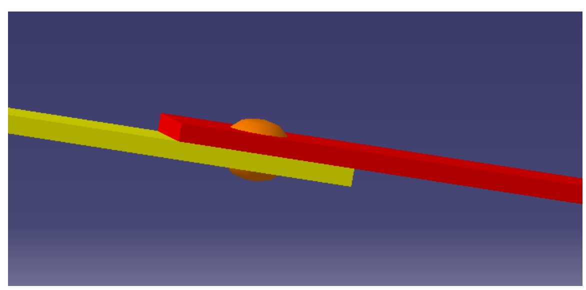


Fig.4.1.3 (CATIA Dome Shape Model)

4.1.1 Finite Element Boundary Conditions and results

Two different orientations in riveted lap joints were considered in the present work are listed as follows:

1. A lap riveted model with dome shaped,
2. A lap riveted model with cap shaped

4.2 Material properties

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

Table 4.2.1.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 2 Issue 3, March - 2013

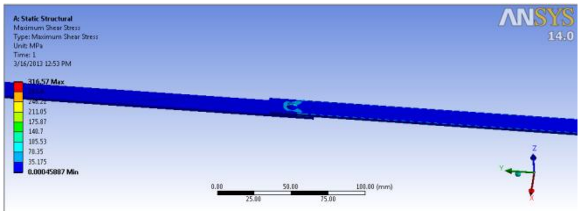


Fig1(a) Stress distribution of dome shape riveted lap joint

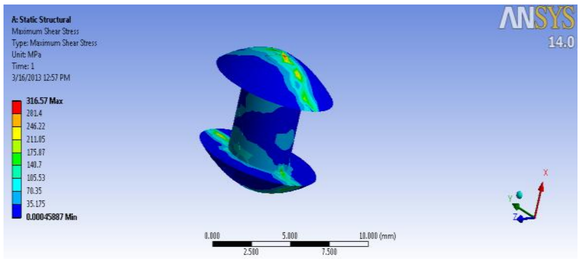


Fig1(a) Stress distribution of Dome shape rivet

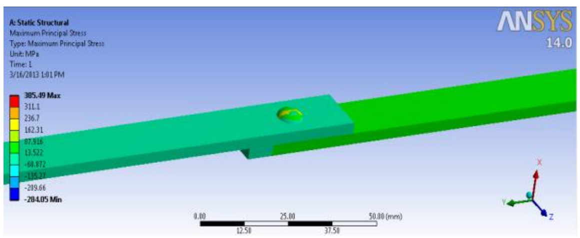


Fig. 2(a) Stress distribution of cap shape riveted lap joint

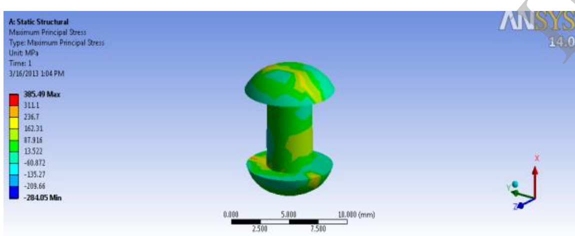


Fig.2(b) Stress distribution of Cap shape rivet

5. CONCLUSION

- The results obtained from ANSYS(Workbench) software for the Single riveted lap joints are compared with each other at different conditions of using different shape of rivets at described locations leads to decreasing the stresses, uniform distribution of load gives more efficient and life to the joints.
- 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.

7. 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] Kale Suresh, K.L.N.Murty & T.Jayananda Kumar presented the paper on “Analysis of adhesively bonded riveted joints” he considered a model for sample Analysis.
[2] Harris C. E., Piascik R. S., Newman C.,1999, “A practical engineering approach to predicting fatigue crack rowth in riveted lap joints”,International Conference on Aeronautical Fatigue (ICAF), W.A.: Seattle.
[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, 22, pp. 319–338.
[4] A Text Book Of Machine Design by R.S.kurmi, J.K.Gupta
[5] Introduction to Finite Elements in Engineering Tirupathi R.Chandrupatla / Ashok D.Belegundu