International Journal of Current Engineering and Technology
E-ISSN 2277 – 4106, P-ISSN 2347 - 5161
©2014 INPRESSCO®, All Rights Reserved
Available at http://inpressco.com/category/ijcet

Research Article

# Stress Analysis of Bolt used to connect Two Plates in Circular ArrayPpattern for Various Loads

Manoj Sayaji Sonawane[A*] and S.N.Shelke[A]

[A]Department of Mechanical Engineering, Sir Visvesvaraya Institute of Technology Nasik, Maharashtra, India

Accepted 10 July 2014, Available online 01 Aug 2014, **Vol.4, No.4 (Aug 2014)**

***Abstract***

*Most of mechanical connections are subject to dynamic loads. Rotating and reciprocating machinery generates significant cyclic loads (fluctuating). Same like that the nuts and the bolts play a most important role in the machinery fittings, machinery makings and the constructional purposes too. Threaded fasteners are probably the best choice to apply a desired clamp load to assemble a joint, at a low cost, with the option to disassemble whenever necessary. As bolted connections (joint) are of sever importance in most of the machines and automobiles etc. For each and every application, designing a unique solution has become time consuming and subjective in approach. There is a strong need to include the virtual validation of the design in the process of approval of the concept at the technical level. Lack of `Analysis' may result in improperly designed structure of the assembly that might get fail under normal working loads and/or other controlled conditions. It is necessary to design and validation of the bolted connection in less possible time and without any need for experimental testing, as it is not possible practically to do experimental testing for every new design of bolted connection. FEA tools and techniques are being used for stress analysis in bolts used to connect two steel plates in circular array pattern. Experimental test conducted in order to validate the results obtained in FEA.*

***Keywords:*** *FEA, Stress Analysis, bolt, Circular Array configuration, UTM, Shear test.*

## 1. Introduction

Manufacturing of large and complex structures is usually possible only when they are composed of assemblies of smaller parts joined together by variety of joining methods since most products are impossible to be produced as a single piece. Manufacturing components and then joining them into a single product is easy and less expensive than manufacturing the whole product at once.

In order to ensure the manufacturability, and reduce the overall manufacturing cost, certain suitable fastening and joining methods should be utilized. Mechanical fasteners can be described as mechanical element that mechanically joins two or more objects of an assembly with desired permanence, stability, and strength.

Mechanical fasteners offer various 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. This paper focuses on threaded fastener and analysis of them for circular array pattern under various loading condition.

### *1.1 Overview of Finite Element Analysis*

The finite element method (FEM) is a computational technique used to obtain approximate solutions of boundary value problems in engineering. Simply stated, a boundary value problem is a mathematical problem in which one or more dependent variables must satisfy a differential equation everywhere within a known domain of independent variables and satisfy specific conditions on the boundary of the domain

### *1.2 Program Overview*

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

### *1.3 General Procedure to Solve and Problem in Hypermesh*

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

*Corresponding author **Manoj Sayaji Sonawane** is a M.Tech (Design) Scholar and **S.N.Shelke** is working as HOD

*1.4 Building the Model*

In this project model is designed with the help of CATIA V 5.0 .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 ".igs" format to import it in Hypermesh.

*1.5 Apply Loads and Obtain the Solution*

Boundary conditions and Different loads acting on the model are applied .The loads applied for this problem in the Hypermesh program are: DOF constraints, Forces.

## 2. Materials and method

The bolts used to connect two plates are made of EN8 grade mild steel. Typical values of shear yield stress is 693.5 Mpa and yield stress is 950 Mpa, Poisson's ratio is 0.3, density 7.8 x $10^3$kg/$m^3$ and Young's modulus is 2.1 x $10^5$ Mpa .The plate is made of cold rolled –cold annealed steel. Typical value of yield stress is 340 Mpa, Poisson's ratio is 0.3, density is 7.8 x $10^3$kg/$m^3$ and Young's modulus is 2.05 x $10^5$ Mpa. FEA software is used to analyze, two different bolted arrangements to connect the two plates with M10 bolts keeping number of bolts Eight. Experimental testing is done to validate the results obtained from FEA software.

## 3. FEA-investigation for bolts connected in circular array configuration

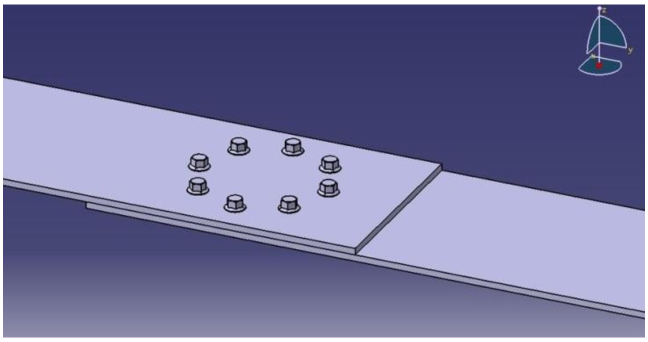


**Fig 1:** Solid Model (Circular Array)

Fig-1 shows, two plates of size 500X180X30mm are connected with M8 X 1.25 X 25 mm Bolts eight in numbers with circular array fashion along the PCD of 120mm and overlapping length of 241 mm. A static analysis of M8 X 1.25 mm flange bolts and nuts are used to connect the two plates is performed using solid modeling software CATIA. Geometry modeling of the two plates connected in circular array fashion including nuts and bolts as shown in Fig-1

Finite element software (Hypermesh-v-12) is used for stress analysis. The elements used in Plate assembly model are all 8-node solid hex elements C3D8R [4] with average element size of 3mm. Finite element model for stress analysis is consisted of 22726 elements and 26530 nodes shown in Fig. 2.To model the contact between the bolts shank and plate holes, bottom surface of upper plate and top surface of lower plate CONTACT elements are used. The contact elements of lower plate are acting as master elements and that of upper plate acting as slave elements.

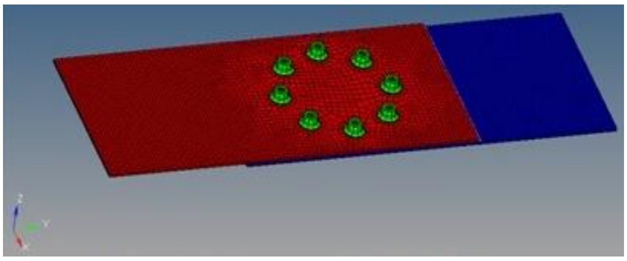


**Fig. 2:** Meshing

*3.1 Loading condition*

Plate assembly is subjected under a load varying from 0 - 5000 N at right end of plate -1 and keeping left end of plate-2 fixed as analysis of bolt has to be done under shear loading. And the stresses induced in bolts are observed in all three directions. The stresses in direction of loading are point of interest, as the tensile force is applied in Y-direction, the stresses in the direction of loading (stress components S22 ) are recorded for analysis under 1000 ,2000, 3000, 4000, and 5000 N loading conditions as shown in Fig-3 to Fig-6.

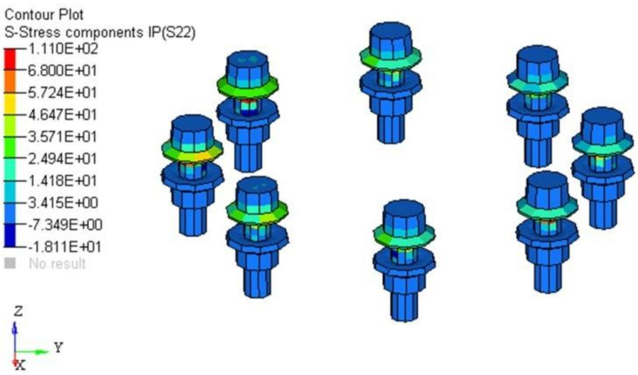



**Fig. 3:** Contour plot of stress component-S22 (1000 N load)

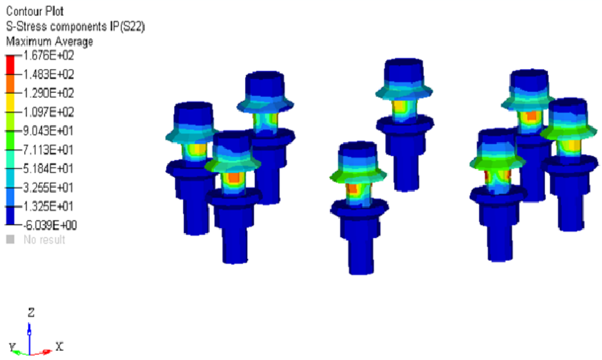



**Fig. 4:** Contour plot of stress component-S22 (2000 N load)

Stress result shows that under the load of 1000N maximum stress induced (Stress Concentration) is at the shank below the head region having value of 111 Mpa.

This is significantly less than bolt materials yield strength ($S_y$= 950 Mpa).So the plate assembly with M8 bolts in circular array fashion, eight in numbers is safe under the tensile load of 1000N.

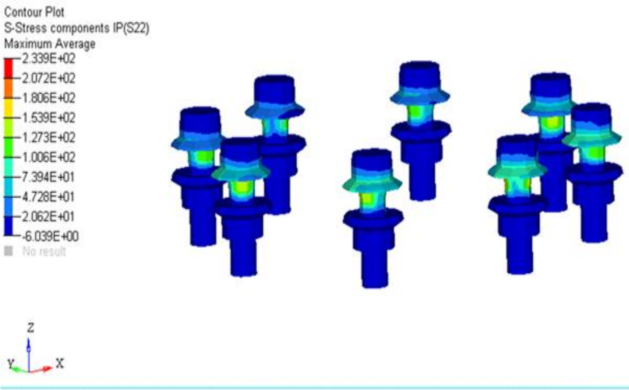



**Fig. 5**: Contour plot of stress component-S22 (3000 N load)

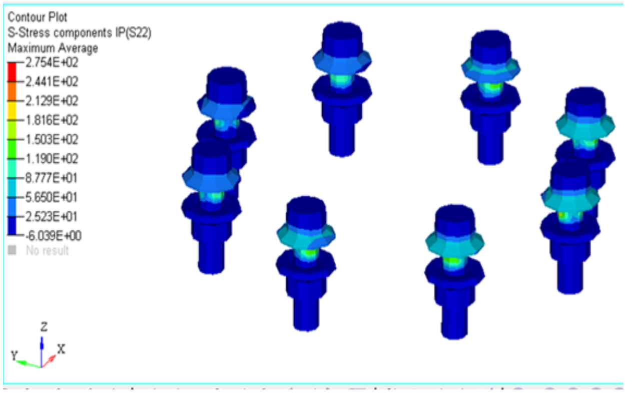
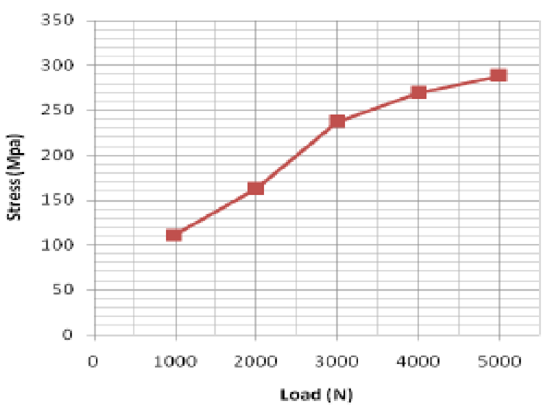



**Fig. 6:** Contour plot of stress component-S22 (5000 N load)

## 4. Result and discussion




**Graph 1:** Load Vs Stress Behavior (Analytical)

From the Fig-3 to Fig-6 we can see that, the stress concentration region in bolt is same i.e. Just below the head of the bolt at shank. Following table-1 shows the respective values of maximum shear stress produced in bolt during 1000 to 5000 N load in finite element analysis.

**Table 1** Stress results observed in FEA

| Load (N) | Stress observed in FEA (Mpa) |
|---|---|
| 1000 | 111.1 |
| 2000 | 162.3 |
| 3000 | 236.7 |
| 4000 | 269.1 |
| 5000 | 288.3 |

## 5. Experimental Investigation

### *5.1 Preparation of test specimen*

Two plates of CRCA material are taken .The eight numbers of holes drilled in circular array with PCD of 120 mm and overlapping length of 241 mm. These plates are then bolted together with M8 X 1.25 mm flanged bolts and nuts of carbon steel material by applying the sufficient torque required for tightening. The length of the plate includes the gauge length of 320mm and rest of the 180mm as gripper length to hold the test specimen in UTM for shear testing as shown in Fig- 8 and Fig-9.

**Fig. 7**: CRCA Plates for Experimental Testing (Circular Array)

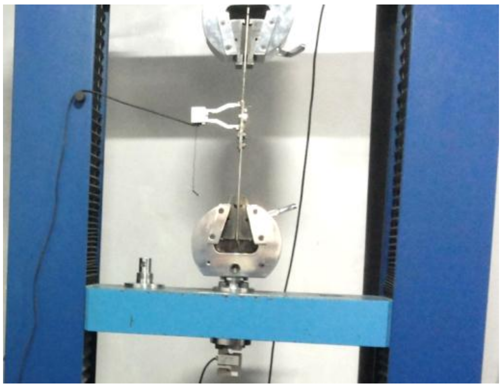
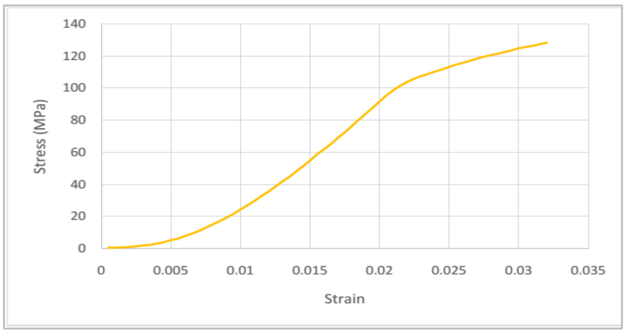


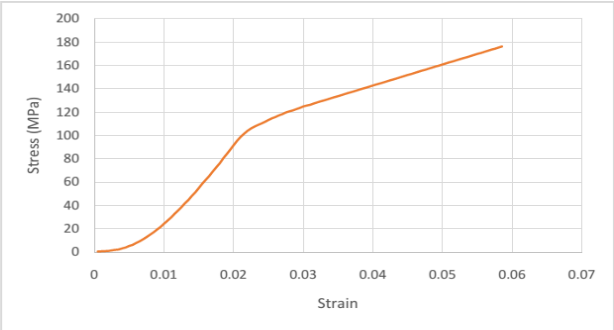
**Fig. 8:** Test Specimen under Shear Test

### *5.2 Testing on UTM*

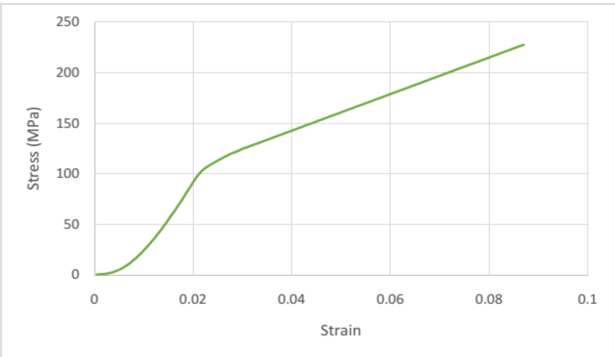
Fig-9 shows the test specimen is loaded on UTM. Strain gauge is used to record stress during shear testing. Non diffusible type strain gauge with data logger is used and it is located at maximum stress region as per the result of finite element analysis i.e. in proximity of bolt region. The test is conducted at a rate of 10 mm /min. Testing is done within the yield stress of bolt material.



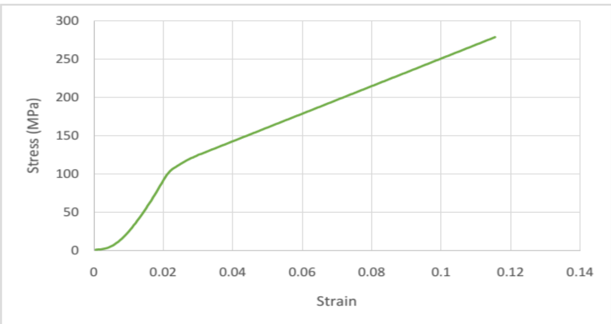

**Graph 2:** Stress- Strain behavior of bolt under 1000 N



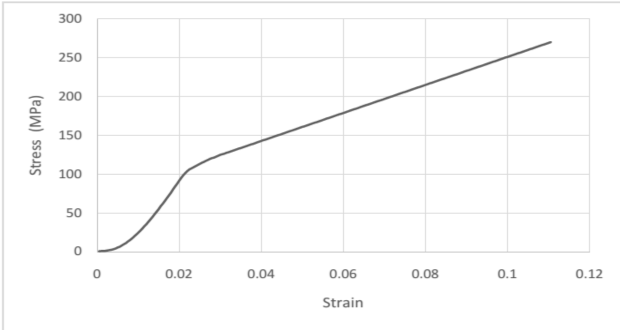

**Graph 3:** Stress- Strain behavior of bolt under 2000 N




**Graph 4:** Stress- Strain behavior of bolt under 3000 N



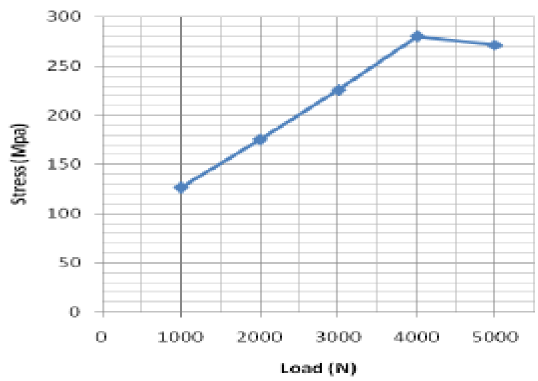

**Graph 5:** Stress- Strain behavior of bolt under 4000 N




**Graph 6:** Stress- Strain behavior of bolt under 5000 N

**Table 2**: Experimental result

| Load (N) | Readings determined by FEA method for circular array configuration (Mpa) | Readings recorded during physical Experimentation (Mpa) | Percent Variation in results (Analysis Vs Experiment) |
|---|---|---|---|
| 1000 | 111.1 | 126.403 | 12.11% |
| 2000 | 162.3 | 175.02 | 7.26 |
| 3000 | 236.7 | 255.84 | 4.58 |
| 4000 | 269.1 | 280.3 | 3.96 |
| 5000 | 288.3 | 271.34 | 6.21 |




**Graph 7:** Load Vs Stress Behavior (Experimental)

## Conclusions

From FEA and Experimental result it is concluded that the FEA technique can be suitably employed for the design analysis of bolted connection. As bolted connections are of sever importance in most of the machines and automobiles etc. It is necessary to design the bolted connection in less possible time and without any need of experimental testing, as it's not possible practically to do experimental testing for every new design of bolted connection. However as value of stress induced in bolt for 5000 N of loading is 271.34 Mpa which is less than the yield stress of bolt material (693.5 Mpa) hence this bolted arrangement is more safe for connecting two plates with M8 X 1.25 X25 mm. From this analysis we also came to know that for connection of two plates having same geometry and material, we can reduce the number of bolts used to connect the plates from 8 Nos to 6 Nos.

## References

Chen Zhong-He, Wang Wei-Qiang, Zhang Le-Wen (2008), *FEM analyses of stress and deformation of a flexible inner pressure bolt*, J China Univ Mining & Technol 18 (2008) 0584–0587

Young-Ho Lee , Hyung-Kyu Kim (2011), *Failure analysis of a pressure vessel bolt in a nuclear fuel frettingwear simulator*, Engineering Failure Analysis 18 (2011) 1735–1741.

Brooks CR, Choudhury A. (2002) *Failure analysis of engineering materials*. 1st ed. USA: McGraw-Hill.

Roozbeh Kiamanesh, M.ASCE;AliAbolmaali, M.ASCE and Mohammad Razavi, M.ASCE (2013), *Effect of Circular Bolt Pattern on Behavior of Extended End-Plate Connection*, journal of structural engineering.

Hosford WF. (2005)*Mechanical behavior of materials.* 2nd ed. USA: Cambridge University Press.

Dowling NE. *Mechanical behavior of materials*. 2nd ed. USA: Prentice Hall; 1999.

P.J. Gray, C.T. McCarthy (2010), *A global bolted joint model for finite element analysis of load distributions in multi-bolt composite joints, Composites*: Part B 41 (2010) 317–325

Chen Cao n, JanNemcik,NajAziz,TingRen (2013), *Analytical study of steel bolt profile and its influence on bolt load transfer,* International Journal of Rock Mechanics & Mining Sciences 60 (2013) 188–195

Mohsen Geramia, Hamid Saberi , Vahid Saberi , Amir Saedi Daryanb (2011), *Cyclic behavior of bolted connections with different arrangement of bolts*, Journal of Constructional Steel Research 67 (2011) 690–70

R. Lacalle, S. Cicero, D. Ferren o, J.A. A lvarez (2010). *Failure analysis of a bolt in a scaffolding system* 15 (2010) 237–24

Zhiwei Yu, Xiaolei Xu (2008), *Failure analysis of connecting bolts and location pins assembled on the plate of main-shaft used in a locomotive turbochanger* 15 (2008) 471–47

LI Ying-ming, MA Nian-jie, YANG Ke, SHI Jian-jun (2009), *Research on FRP bolt-end failure mechanism, Mining Science and Technology* 19 (2009) 0522–052

Zhang Kaifu, Cheng Hui, Li Yuan (2011), *Riveting Process Modeling and Simulating for Deformation Analysis of Aircraft's Thin-walled Sheet-metal Parts*, Chinese Journal of Aeronautics 24 (2011) 369-37

Mohamed Tahar Nasraouia , Jamel Chakharib , Boubaker Khalfi , Mustapha Nasria (2013), *Numerical and Experimental Study of Shear Loaded Bolted Joint*, International Journal of Current Engineering and Technology, INPRESSCO

Marcelo, A.L., Uehara, A.Y, Utiyama, R.M, Ferreira, I 2011), *Fatigue Properties of High Strength Bolts*, Procedia Engineering 10 (2011) 1297–1302

M. Reihanian a, K. Sherafatnia b, M. Sajjadnejad c (2011), *Fatigue failure analysis of holding U-bolts of a cooling fan blade*, *Engineering Failure Analysis* 18 (2011) 2019–2027

Hosford WF. (2005)*Mechanical behavior of materials*. 2nd ed. USA: Cambridge University Press.