ISSN: 2277-9655
(I2OR), Publication Impact Factor: 3.785

IJESRT

INTERNATIONAL JOURNAL OF ENGINEERING SCIENCES & RESEARCH TECHNOLOGY

FINITE ELEMENT ANALYSIS OF TRI-CLAMP

Sahil*, Shrey Durvasa, Shubham, Manoj Kumar
Project Associative, Eduhive Engg Consult
Student, Department Of Mechanical Engineering, Chandigarh University, India
Student, Department Of Mechanical Engineering, Chandigarh University, India
Production executive Sperry plast limited

ABSTRACT

The objective of this paper is to analyze the Tri-Clamp. This work describes a model of the tri clamp is developed using Catia. Finite element analysis performed by the Ansys software. The Tri-Clamp will be used under ASME VIII-1 service. This product cannot be calculated to code rules due to the complexity of its geometry. Instead the rules of ASME VIII-2 are used with ASME VIII-1 allowable stresses to determine its acceptability. Pressure inside the shell is 1500 psi and the model is restrained by applying a fixed support at the clamp.

KEYWORDS: Tri Clamp, Fea, Catia, Ansys.

INTRODUCTION

Tri clamps or tri-clamps (sometimes called Tri cover) fittings are a class of sanitary flanges, clamps, gaskets and other fittings that can be of great use for the still builder. They are used in column to boiler connections, breaks in the column to allow for modular setups, detachable takeoff / lyne arms, sight glasses, fill and drain ports on the boiler, and many other uses.These unions enable extremely easy and fast set up and takedown of equipment. Triclamps are also compatable with the standard flange on a Sankey beer keg, and with the Easy flange, making them ideal for DIY purposes and low cost still builds.

Model Generation and Boundary Condition of Try Clamp

Model is created in Catia and imported in ansys workbench as stp file. The model used in the analysis represents 1/2 of the Tri-Clamp due to symmetry. A global mesh size of 0.09375" has been applied using 3D tetrahedral solid elements. The mesh size results in a reported error of less than 5%.In figure 1 shows Tri Clamp Assembly

Figure: 1 Tri Clamp Assembly

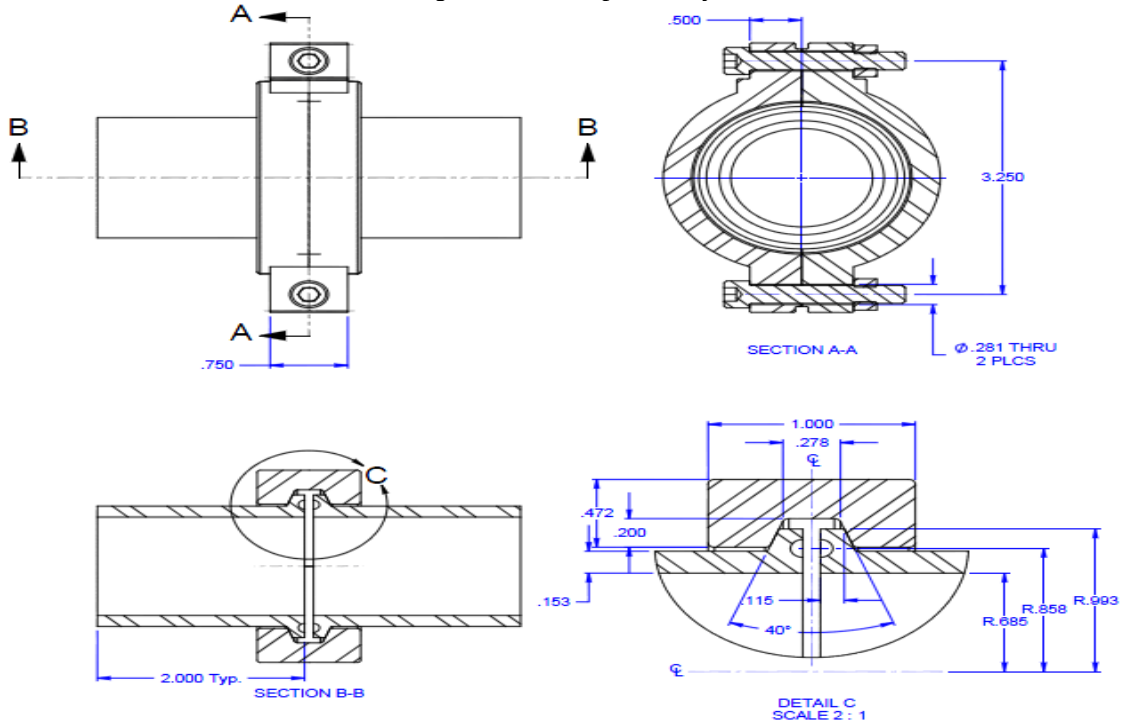


Fig.2- Different section view of the Tri-Clamp

MATERIAL USED

1) Material 1 is used clamp structures.
2) Material 2 I used for the pipe caps
3) Material 3 is used for the bolts used in the triclamps

Basic need of the materials used in the triclamps is the strength of the materials used so that it can absorb the stresses generated during the clamping process.

	Material 1	Material 2	Bolting 3
Material =	SA-182 F316L	SA-403 316L	SA-193 B7
Application =	Clamps/Ferrules	Pipe Caps	Bolts
Sm [psi] =	14,800	14,800	25,000
Sy [psi] =	16,400	16,400	88,500
E1 =	1.0	1.0	1.0
E2 =	1.0	1.0	1.0
E [psi] =	25,900,000	25,900,000	27,400,000
v =	0.30	0.30	0.30

Table 1. List of material used for Tri-Clamp

MESHING THE GEOMETRIC MODEL

The meshing of geometry was performed in ANSYS and as discussed earlier in the methodology. The meshing is carried out by using 3D tetrahedral elements having global mesh size of 1 inch, corresponding mesh details are as discussed below.

MESH DETAILS	
Physical reference	Mechanical
Relevance centre	Coarse
Element size	Default
Smoothing	Low
Transition	Fast
Span angle centre	Coarse
Nodes	61671
Elements	37181

Table 2. Details of meshing

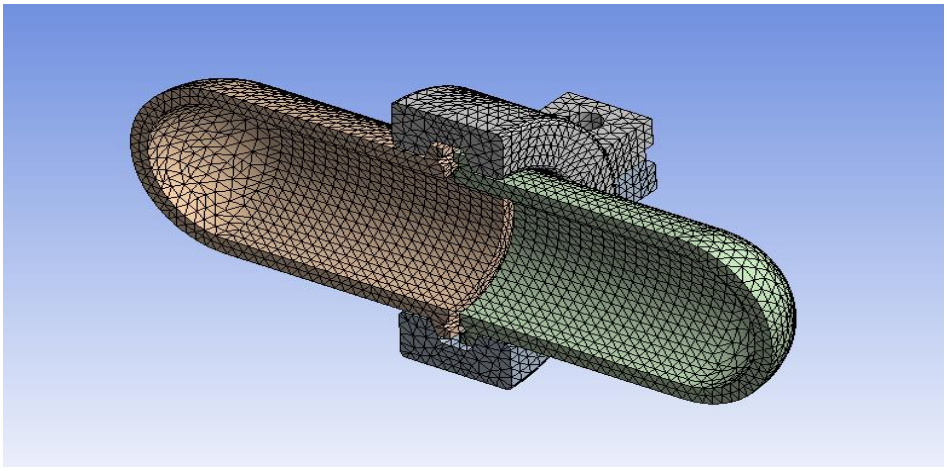
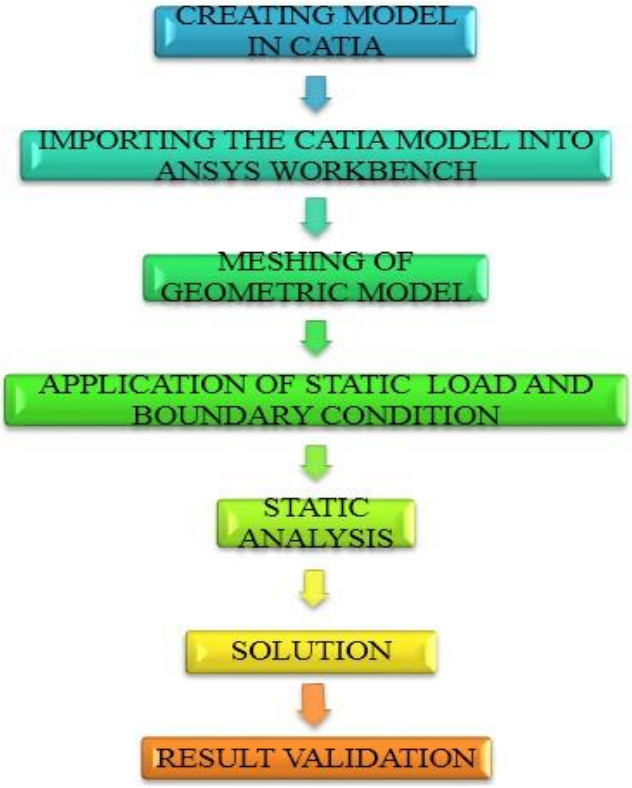


Fig.3.Tetrahedral meshing of the geometry



APPLICATION OF LOAD AND BOUNDARY CONDITIONS

- As per the problem statement the pressure force of 1500 psi is applied on the inner shell.
- Fixed support is applied on the one end of shaft which is connected to the motor.

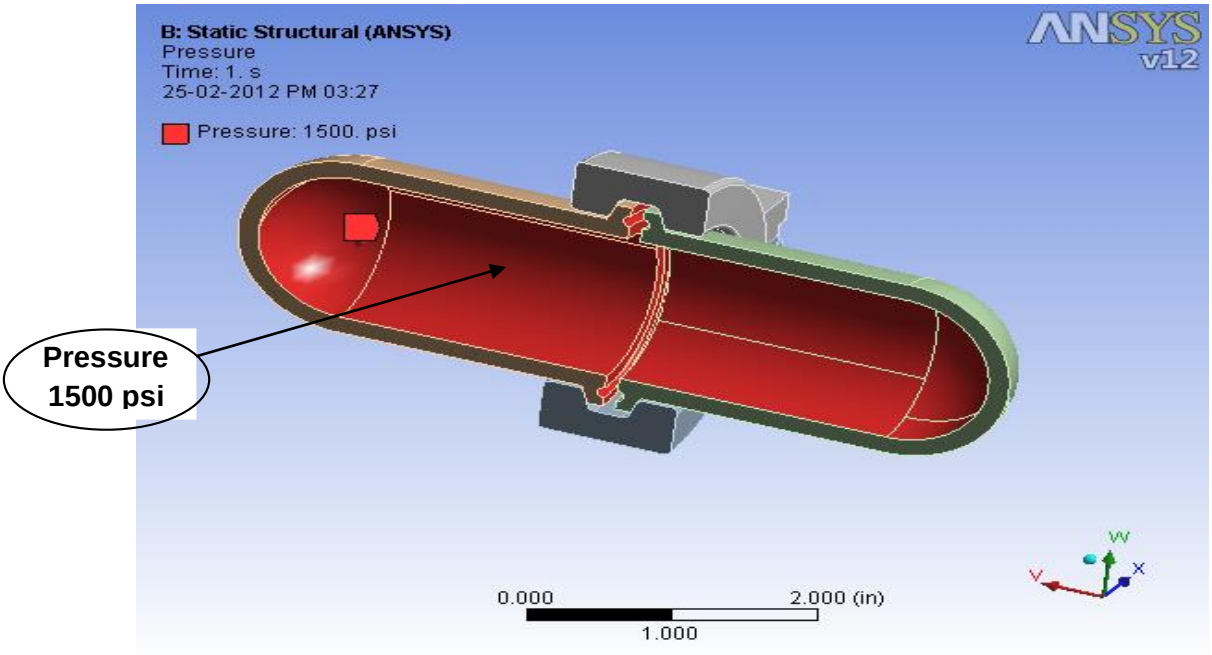


Fig.4.Pressure force applied on the inner shell

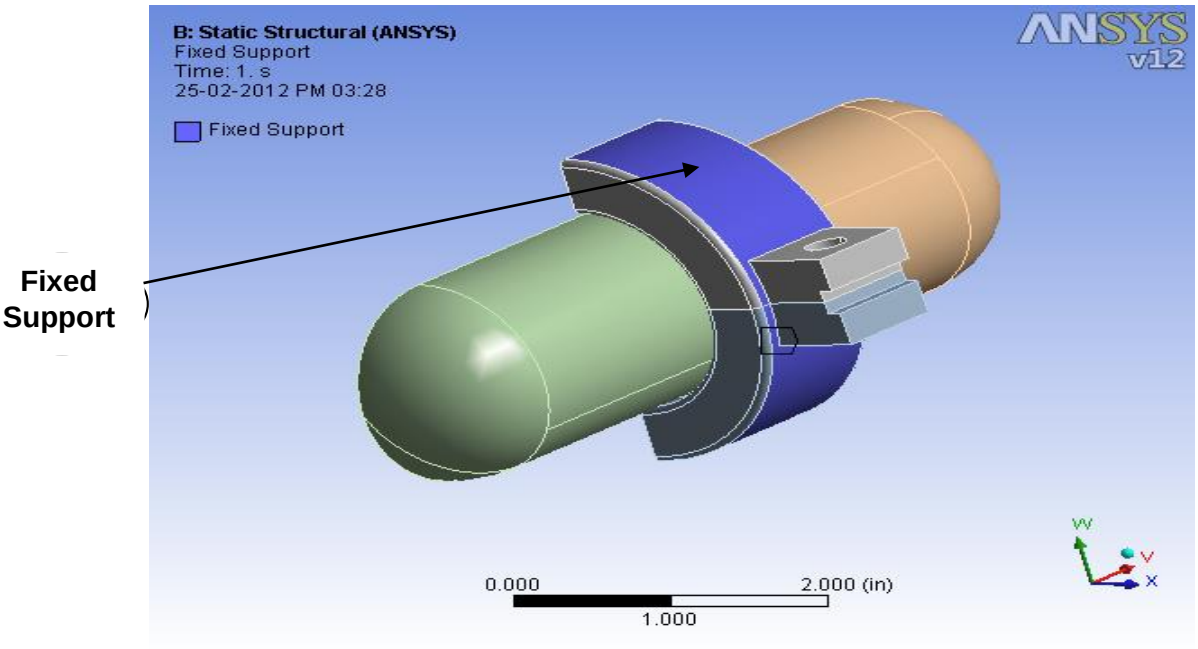


Fig.5 Fixed support onouter clamp

RESULT OF STATIC STRUCTURAL ANALYSIS

The analysis was done by applying the pressure force on inner shell and fixed support at the clamps.After analysis the maximum Von-Mises stress on the model a peak stress of 21911 psi is located on the clamp face in contact with the ferrule. The stress and deformation plot are as follows:

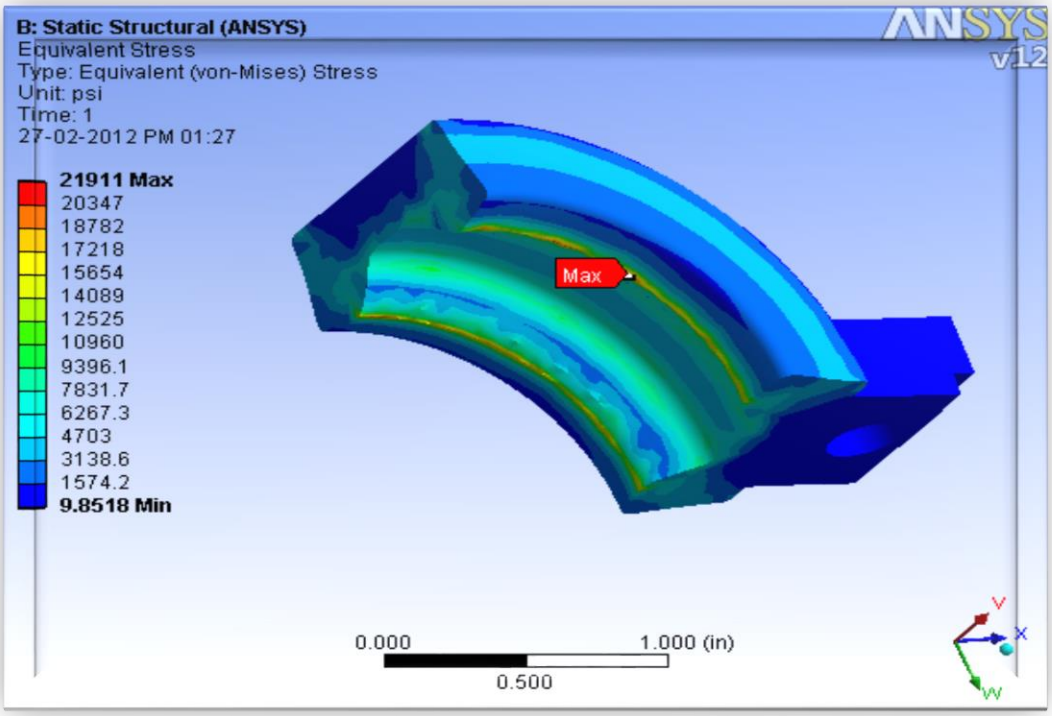


Fig6. Stress distribution plot

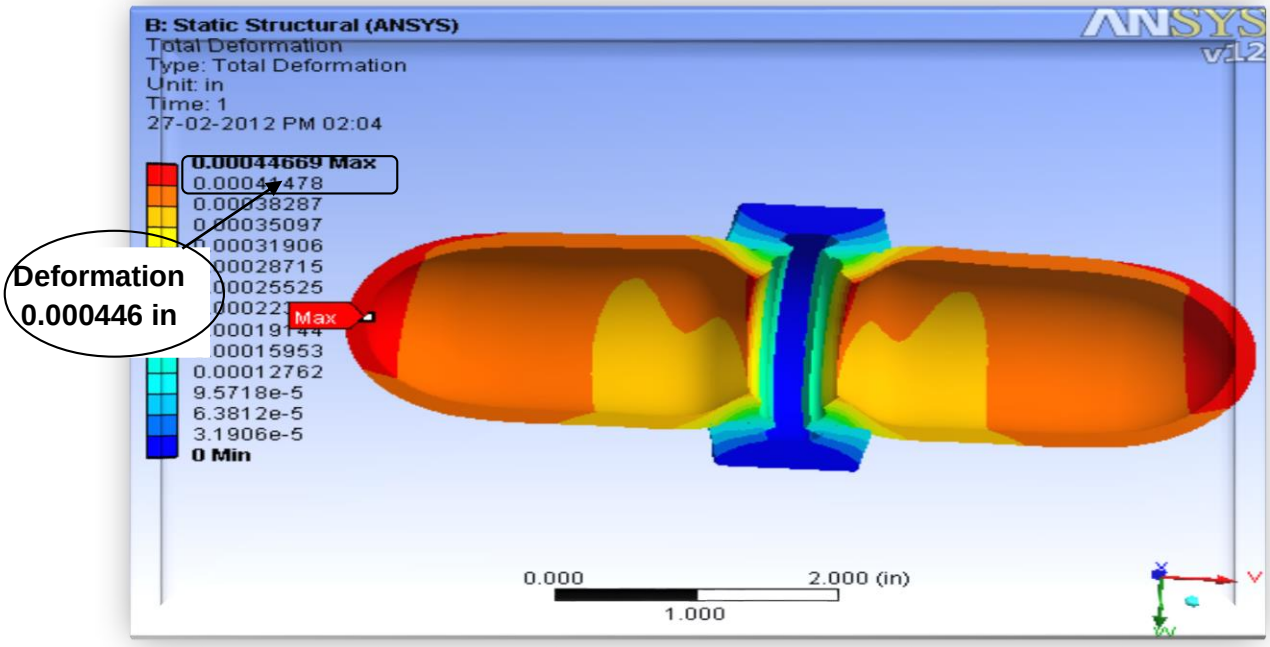


Fig.7. Total deformation plot

CONCLUSION

During the process the static load of 1500 psi was applied on the inner surfaces. And the fixed support on the outer body and the analysis was carried out on the outer tri clamp. Results so obtained showed the maximum Von-Misses stress of 21911 psi is located on the clamp face in contact with the ferrule. And the deformation observed was only

of .000446 in. From the above static structural it is clear that all checkpoints are satisfied and following conclusion can be drawn. The Tri-Clamp meets the design rule and the allowable stress is acceptable.

REFERENCES

[1] Joseph C. Slater , Wright Finite Element method: An easily extensible research-oriented finite element code, April 5,2004.
[2] O.C. Zienkiewicz , R.L. Taylor , The Finite Element Method, Fifth Edition, ISBN: 07506 50559
[3] S.S. Rao, The Finite Element Method in Engineering, fourth Edition, ISBN: 0750678283
[4] George S., Kollar, Laszloa P. Mechanics of Composite Structures.Cambridge University Press, New York, 2003
[5] Design Data book, "The McGraw-Hill Publication".
[6] J E Shigley, Mechanical Engineering Design, McGraw Hill Book Company, 1986.
[7] ASME VIII-1 Code