IJARIIT INTERNATIONAL JOURNAL OF ADVANCE RESEARCH, IDEAS AND INNOVATIONS IN TECHNOLOGY

ISSN: 2454-132X
Impact Factor: 6.078
(Volume 7, Issue 2 - V7I2-1409)
Available online at: https://www.ijariit.com

Modeling, assembly, and static structural analysis of a cotter joint

Ruturaj D. Bhosale
ruturaj.21810042@viit.ac.in
Vishwakarma Institute of Information Technology, Pune, Maharashtra

ABSTRACT

The objective of this project is the calculation of dimensions, modeling, Assembly, and Static Structural analysis of cotter joint. Theoretical method is used here to calculate the dimensions. The modeling and Assembly are done in CATIA V5R20 and Static Structural analysis in ANSYS 2020R1. While doing the static structural analysis more focus is given on generating the most efficient mesh and the comparison of results with respect to four different types of mesh is done. Here the parameters evaluated and compared are Equivalent (von-Mises) Stress, Shear Stress and Total Deformation. The analysis of the cotter joint is done for the tensile loading with one end kept as fixed support.

Keywords: *Cotter joint, Theoretical design, CAD modeling, Meshing, Static structural analysis*

1. INTRODUCTION

A cotter joint is used to connect to co-axial rods, which are subjected to either axial tensile force or axial compressive force. It is also used to connect a rod on one side with some machine part like a crosshead or a base plate on the other side.

It is not used for connecting shafts that rotate and transmit torque. The principle of wedge action is used in a cotter pin. A cotter is a wedge-shaped piece made of steel plate. The joint is tightened and adjusted by means of wedge action of the cotter. The construction of a cotter joint, used to connect two rods A and B. Rod-A is provided with a socket end, while rod-B is provided with a spigot end. The socket as well as the spigot is provided with a narrow rectangular slot. A cotter is tightly fitted in this slot passing through the socket and spigot. The cotter has uniform thickness and the width dimension b is given a slight taper. The taper varies from 1 in 48 to 1 in 24 and it may be increased up to 1 in 8, if a provision of locking device is made.

Previously there is a variety of research done regarding the analysis of cotter joint some of the researches along with its outcomes are mentioned as follows. Research by [3] Aashokrao Shinde and Omkar Chandrakant Vibhute on modeling and analysis of cotter joint was done. They also studied the values of Equivalent (von-Mises) Stress with respect to the changing load values. [4] Sashikant T. More and Dr. R.S. Bindu studied the effects of mesh size on the accuracy of analysis and states that mesh size is a critical part of analysis. Research by [5] Padmakar Raut compares various performance parameters with respect to linear and quadratic forms of tetrahedral and hexahedral mesh.

Research by [6] Weibing Liu, Mamtimin Geni, and Lie Yu have obtained different FEA accuracy by different element size and type. It was observed that calculation accuracy under hexahedral element is higher than tetrahedral element.

2. THEROTICAL DESIGN OF COTTER JOINT

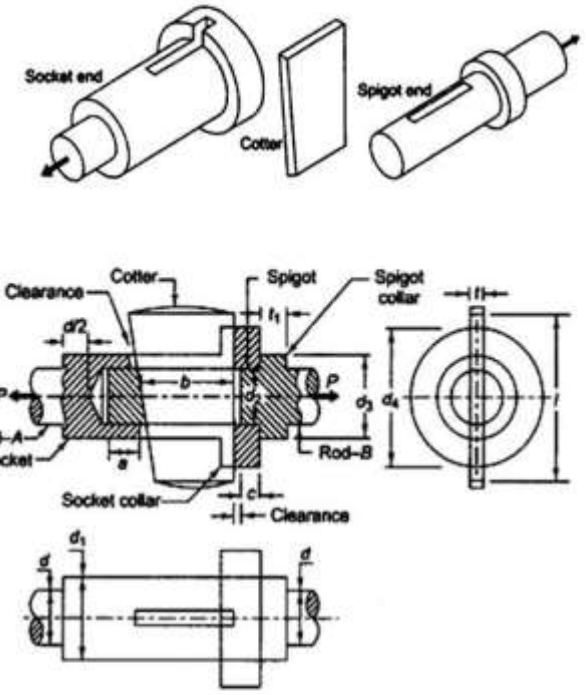

Fig. 1: Cotter joint

Let,
'P' = Load carried by the rods (N),
'd' = Diameter of the spigot (mm),
'd1' = Outside diameter of socket (mm),

‘d2’ = Diameter of spigot or inside diameter of socket (mm),
‘d3’ = Outside diameter of spigot collar (mm),
‘t1’= Thickness of spigot collar (mm),
‘d4’ = Diameter of socket collar (mm),
‘c’ = Thickness of socket collar(mm),
‘b’ = Mean width of cotter(mm),
‘t’ = Thickness of cotter(mm),
‘l’ = Length of cotter (mm),
‘a’ = Distance from the end of the slot to the end of rod (mm),
σt = Permissible tensile stress for the rods material (MPa),
τ = Permissible shear stress for the cotter material (MPa),
σc = Permissible crushing stress for the cotter material (MPa).

Here we consider, a cotter joint is subjected to tensile loading with following material properties and design considerations:

Table 1: Material Properties and design considerations

Material	Carbon steel, 1040 annealed
Force applied (P)	150 kN
Factor of safety (FOS)	4
Ultimate tensile stress	500 MPa
Ultimate shear stress	250 MPa
Ultimate Crushing stress	1000 MPa
Taper of cotter	1 in 32

3. CALCULATION OF DIMENSIONS

The Permissible stresses for the Socket, Spigot and cotter are as follows.

σt = (Ultimate tensile stress / FOS) = (500/4) = 125 N/mm2,

σc = 2*(Ultimate tensile stress / FOS) = 2* (500/4) = 250 N/mm2,

τ = 0.5*(Ultimate tensile stress / FOS) = 0.5*(400/6) = 62.5 N/mm2,

P = 150 KN,
Taper of cotter is 1 in 32 is considered,
Dimensions of spigot ‘d’ = 40mm,
Thickness of cotter ‘t’ = 12.4 mm,
Diameter of spigot or inside diameter of socket
‘d2’= 50 mm,
Outer diameter of socket ‘d1’= 70mm,
Diameter of spigot collar ‘d3’ = 60mm,
Socket collar ‘d4’ = 100 mm,
‘a’ = ‘c’ = 30 mm,
Width of cotter ‘b’ = 98mm,
Thickness of spigot collar ‘t1’= 18 mm.

4. CAD MODELLING AND ASSEMBLY OF COTTER JOINT

3D CAD modeling is a versatile tool Available to convert imagination into reality. There are a variety of software’s available which provide a platform for modeling of a wide range of objects with certain approximation. But those approximations are within the tolerable considerations. Here we use CATIA V5R20 for modeling of Cotter joint. CATIA (Computer Aided Three-Dimensional Interactive Application) is a multi-platform software suite developed by the French company Dassault Systems. CATIA enables the creation of 3D parts, from 2D sketches, sheet metal, composites, and molded, Forged or tooling parts up to the definition of mechanical assemblies. The software provides advanced technologies for mechanical surfacing. It provides tools to complete product definition, including functional tolerances as well as kinematics definition. CATIA provides a wide range of applications for tooling design, for both generic tooling and mould & die. Following figures shows the sketches of cotter joint in CATIA V5R20.

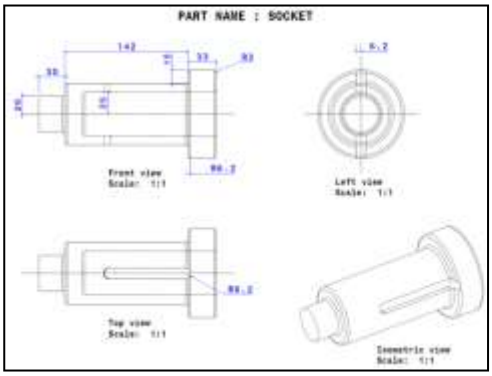

Fig. 2.1: Socket sketch

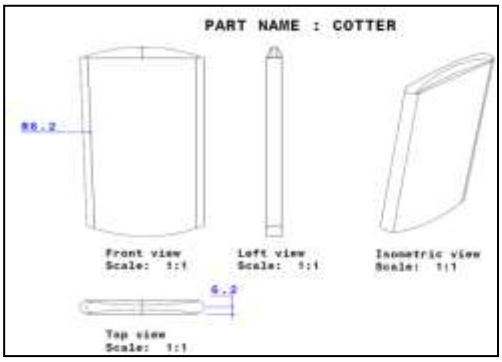

Fig. 2.2: Cotter sketch

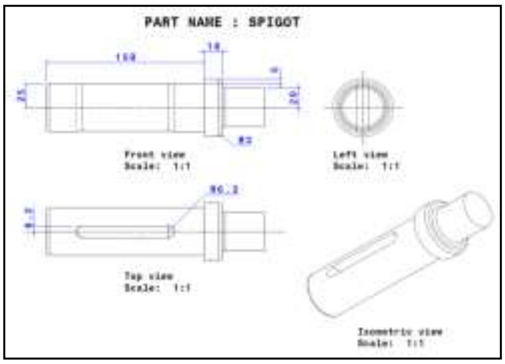

Fig. 2.3: Spigot sketch

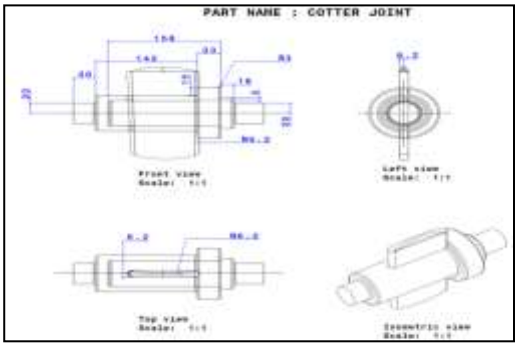

Fig. 2.4: cotter joint assembly sketch

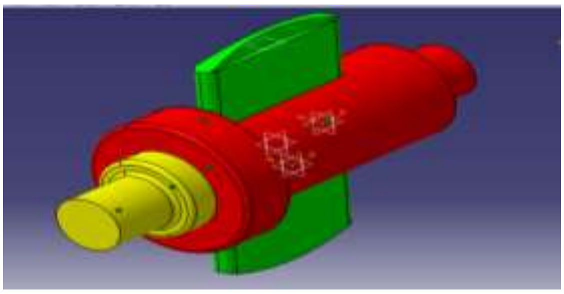

Fig. 2.5: CAD Assembly of cotter joint

5. FEA ANALYSIS OF COTTER JOINT

Finite element analysis (FEA) is a computer aided method for an analysis of a product under the influence of real-world forces, vibration, heat, fluid flow and other physical effects. Finite element analysis predicts whether a product will break, wear out or work the way it was designed. It is called analysis, but in the product development process, it is used to predict what's going to happen when the product is used. FEA fundamentally works by breaking down a real object into a large number of finite elements.

The solution of the mathematical model helps predict the behavior of each element. A computer then adds up all the individual behaviors to predict the behavior of the actual object. FEA is an approximate method of analysis subjected to errors. So, it is best suited until the errors and approximations are under the tolerable considerations. FEA helps us to analyze product affected by many physical effects, including: Mechanical stress, Mechanical vibration, Fatigue, Motion, Heat transfer, Fluid flow, Electrostatics, Plastic injection molding.

A static structural analysis is a vital tool to study the displacements, stresses, strains, and forces in structures or components caused by a variety of loads that do not induce significant inertia and damping effects. The main assumption of static structural analysis is steady condition of loading and reaction.

The FEA tool used in this case is ANSYS 2020R1. The ANSYS 2020R1 program is self-contained general purpose finite component program. It provides an entire resolution to design problems. It consists of powerful design capabilities like full consistent quantity solid modeling, design optimization and automotive vehicle meshing. It Also provides a variety of meshing options like tetrahedron mesh, hex dominant mesh, Sweep mesh, etc. It Also provides refinement mesh options for critical regions of the product.

In this project the cotter joint is analyzed against the tensile force of 150KN which is applied on one end keeping the other end fixed. The Static structural analysis is done by generating 4 different types of mesh. For analysis we need to remove the unwanted curvature for proper meshing purpose.

Mesh Type 1: Basic Fine mesh

This is the basic type of mesh which is generated just by selecting the fine option from the sizing options.

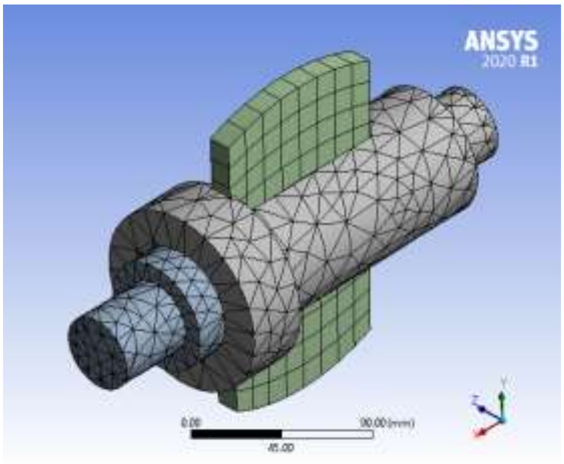

Fig 3.1: Basic fine mesh

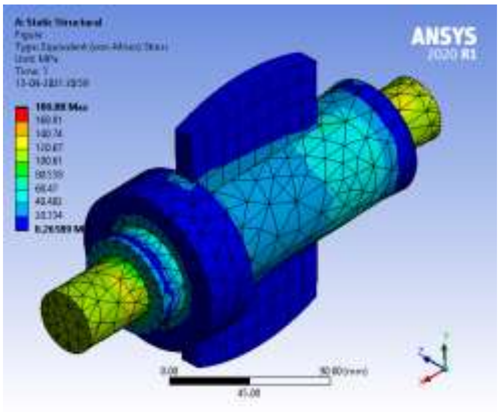

Fig 3.2: Equivalent (von-Mises) Stress (mesh type 1)

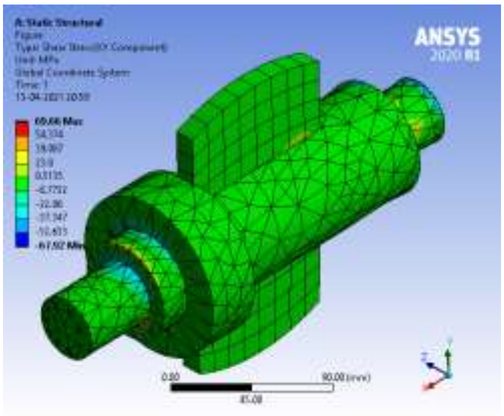

Fig 3.3: Shear Stress (mesh type 1)

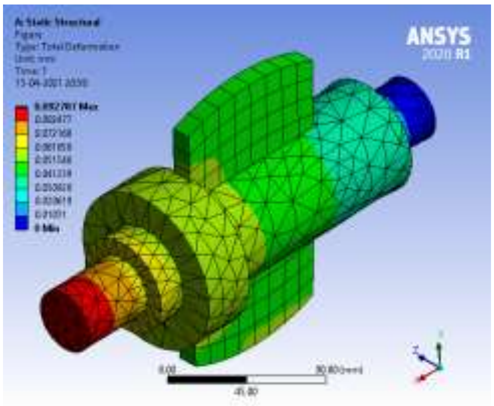

Fig 3.4: Total deformation (mesh type 1)

Mesh Type 2: Tetrahedron mesh

This type mesh is generated by selecting the method type as tetrahedron and element size 5mm. This mesh is also characterized by considering no adaptive sizing and considering the proximities and curvatures.

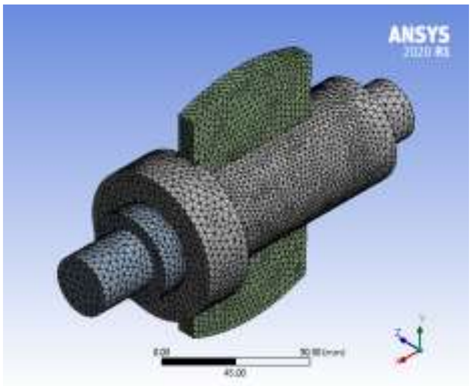

Fig 4.1: Tetrahedron mesh

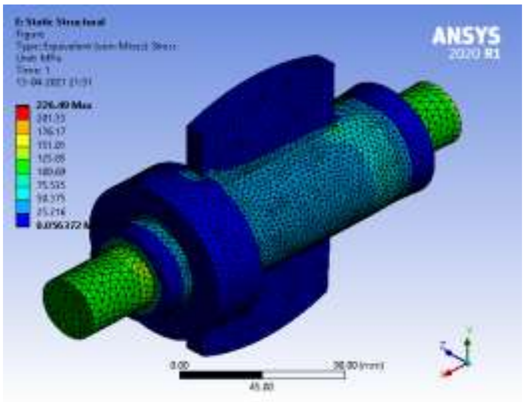

Fig 4.2: Equivalent (von-Mises) Stress (mesh type 2)

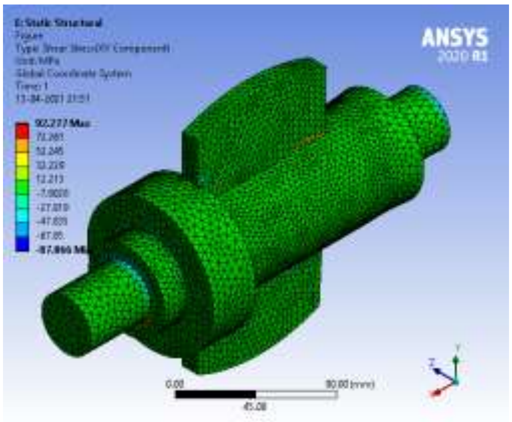

Fig 4.3: Shear Stress (mesh type 2)

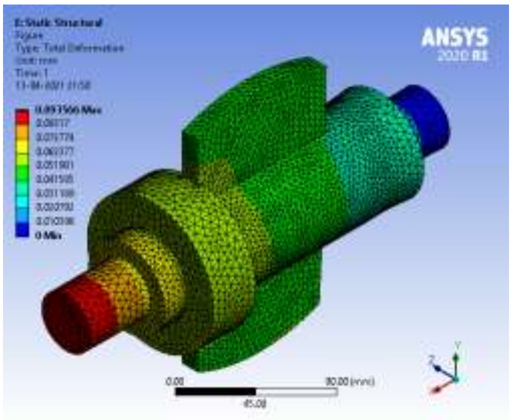

Fig 4.4: Total deformation (mesh type 2)

Mesh Type 3: Hex Dominant mesh

This type mesh is generated by selecting the method type as Hex dominant and element size 5mm. This mesh is also characterized by considering no adaptive sizing and considering the proximities and curvatures.

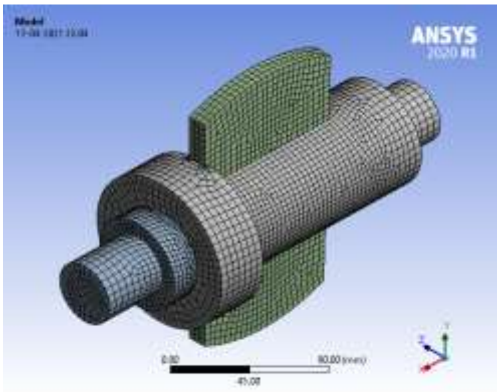

Fig 5.1: Hex dominant mesh

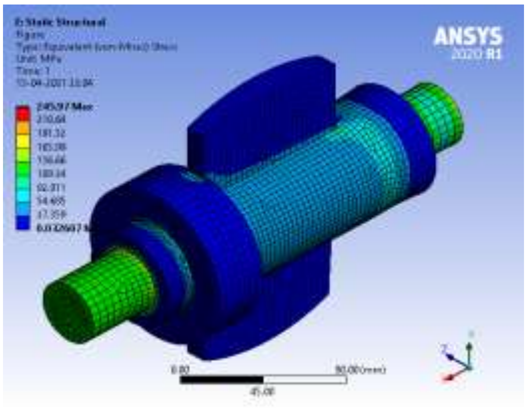

Fig 5.2: Equivalent (von-Mises) Stress (mesh type 3)

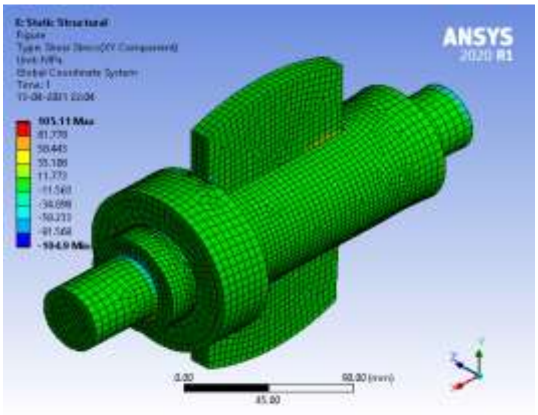

Fig 5.3: Shear Stress (mesh type 3)

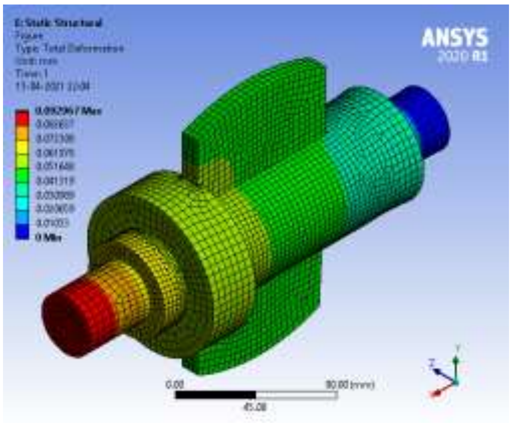

Fig 5.4: Total deformation (mesh type 3)

Mesh Type 4: Refinement mesh

This type mesh is generated by doing the edge refinement, as the maximum stress values were appearing at the edges. This attempt was done to refine the results obtained.

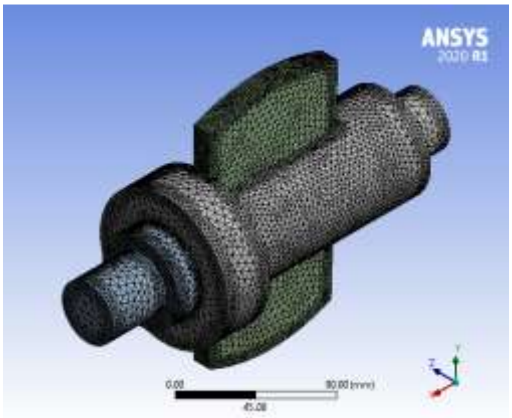

Fig 6.1: Refinement mesh

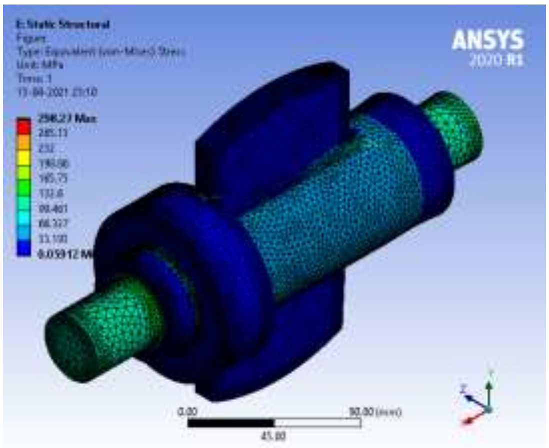

Fig 6.2: Equivalent (von-Mises) Stress (mesh type 4)

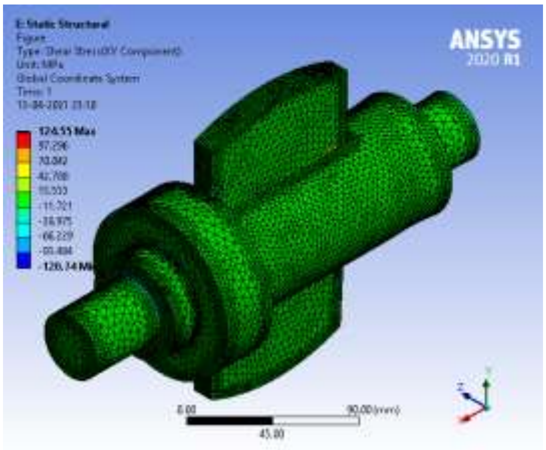

Fig 6.3: Shear Stress (mesh type 4)

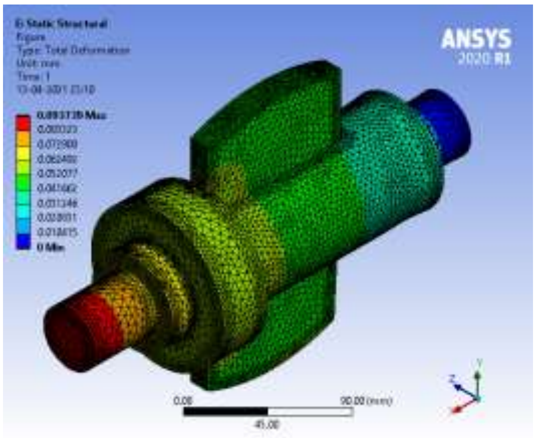

Fig 6.4: Total deformation (mesh type 4)

6. RESULT

The comparative analysis of Equivalent (von-Mises) Stress, shear stress and total deformation with respect to each of the four types of mesh generated is given below

Table 2.1: Equivalent (von-Mises) Stress values

Mesh Type	Equivalent (von-Mises) Stress
Mesh 1	180.88 MPa
Mesh 2	226.49 MPa
Mesh 3	245.97 MPa
Mesh 4	298.27 MPa

Table 2.2: Shear Stress values

Mesh Type	Shear Stress
Mesh 1	69.66 MPa
Mesh 2	92.277 MPa
Mesh 3	105.11 MPa
Mesh 4	124.55 MPa

Table 2.3: Total Deformation values

Mesh Type	Total Deformation
Mesh 1	9.2787e-002 mm
Mesh 2	9.3566e-002 mm
Mesh 3	9.2967e-002 mm
Mesh 4	9.3739e-002 mm

The comparative analysis of Equivalent (von-Mises) Stress, shear stress and total deformation with respect to number of mesh elements for each of the four types of mesh generated is given below.

Table 2.4: No. Of elements values

Mesh Type	No. of mesh elements
Mesh 1	4874
Mesh 2	142356
Mesh 3	35478
Mesh 4	426768

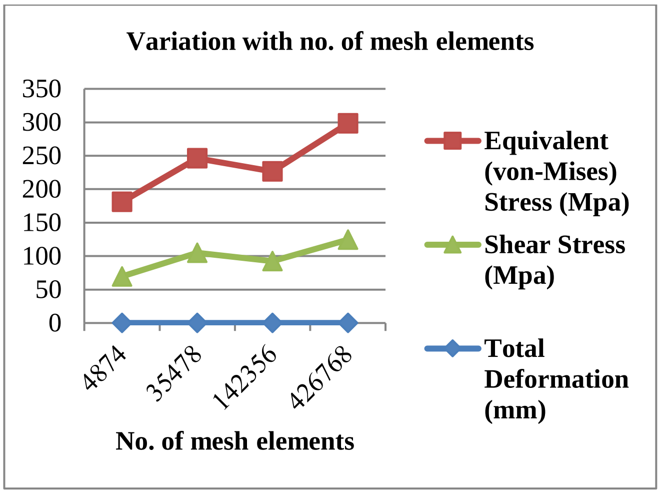

Chart 1: Variation of parameters with no. of mesh elements

7. CONCLUSION

For the proper Static Structural Analysis of an element or an assembly, proper meshing is required. So, from the results obtained we can conclude that if we follow the meshing order from mesh 1 to mesh 4 then the values of the parameters get refined. From the result tables it is seen that the maximum values of Equivalent (von-Mises) Stress, shear stress and total deformation are obtained for the mesh 4 (Refinement mesh) so in this case it is the most suited meshing type. Generally, with the increase in number of mesh elements the results are refined, but on the other hand with the increase in number of mesh elements the time required for the solution also increases. From the line chart-1 we can see that for this case, mesh 3 (hex dominant) has better results with a smaller number of mesh elements as compared to mesh 2 (tetrahedron mesh). So, we can conclude that for static structural analysis hex dominant mesh is preferable over tetrahedron mesh. This conclusion agrees with the result obtained by [6] Weibing Liu, Mamtimin Geni, and Lie Yu. We can also conclude that this order of meshing can be implemented for the analysis of other mechanical elements or an assembly too.

8. REFERENCES

[1] “Design of machine element”, V.B Bhandari – 3rd edition.
[2] "A text book of machine design."R.S khurmy & J.K Gupta.

[3] "Modeling and Analysis of Cotter Joint 3" by Ashokrao Shinde and Omkar Chandrakant Vibhute. International Journal of Engineering Research & Technology. Published by: www.ijert.org Vol. 6 Issue 07, July – 2017.

[4] "Effect of Mesh Size on Finite Element Analysis of Plate Structure" by Shashikant T. More and Dr. R. S. Bindu. International Journal of Engineering Science and Innovative Technology (IJESIT) Volume 4, Issue 3, May 2015).

[5] "Impact Of Mesh Quality Parameters On Elements Such As Beam, Shell And 3D Solid In Structural Analysis" by Padmakar Raut. (IJERA) ISSN: 2248-9622 www.ijera.com Vol. 2, Issue 6, November- December 2012.

[6] "Mesh-size Effect Study of Ductile Fracture by Non-local Approach" by Yaning Li and Tomasz Wierzbicki, Proceedings of the SEM Annual Conference, June 1-4, 2009 Albuquerque New Mexico USA.