INTERNATIONAL JOURNAL OF ADVANCE RESEARCH, IDEAS AND INNOVATIONS IN TECHNOLOGY

ISSN: 2454-132X
Impact Factor: 6.078
(Volume 7, Issue 2 - V7I2-1402)
Available online at: https://www.ijariit.com

Impact of mesh size on UV joint and flange coupling in static structural analysis

Pranav Prashant Pandharpatte
pranav13pandharpatte@gmail.com
Vishwakarma Institute of Information Technology, Pune, Maharashtra

Harshawardhan Vijay Deore
deorehv@gmail.com
Vishwakarma Institute of Information Technology, Pune, Maharashtra

Dr. Atul Kulkarni
atul.kulkarni@viit.ac.in
Vishwakarma Institute of Information Technology, Pune, Maharashtra

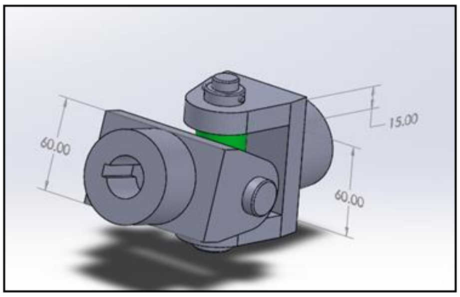
ABSTRACT

In finite element analysis mesh size plays a very important role in determining the accuracy of the result. This paper gives an idea about effect of mesh size on the result. On the basis of result obtained one can get idea regarding, choosing appropriate mesh size during analysis. Static analysis is performed in Ansys software. The models under consideration are UV joint and Flange coupling assembly (both separate) which are made in Solidworks.

Keywords: Finite element analysis, Mesh size, Mesh Density, Variation in meshing.

1. INTRODUCTION

In finite element analysis mesh size plays an important role in determining the accuracy of the result. According to FEA theory, smaller mesh size gives better result as compared to larger mesh size. It is also evident that smaller mesh size takes considerably more time in computing the result. The objective of this paper is to provide guidelines for choosing proper mesh size considering the time complexity in computing. Static structural analysis has been performed and results of stress and deformation are compared by plotting graphs.



Model from SolidWorks

2. PROBLEM STATEMENT COMPONENT 1: UV JOINT

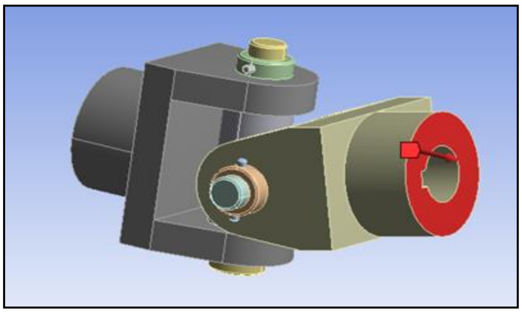
For analysis, first, Universal Joint assembly is taken. The component below in made in SolidWorks. One end of component is fixed and other end in under tensile force of 4500 N. The material used is structural steel.

3. STATIC STRUCTURAL ANALYSIS

The static structural analysis of both components is performed in Ansys. On the UV joint the force applied is considered as 4500N and on the flange coupling the moment applied is 5000N-m. The mesh sizes are taken as 3mm, 4mm, 5mm, 10mm, 15mm, 20mm, 25mm and 30mm. Both components are considered to be made of Structural Steel.

Table 1: Material properties of Structural steel

Young's modulus (GPa)	210
Poisson's ratio	0.3
Yield Strength (MPa)	433
UTS (MPa)	460



Force application

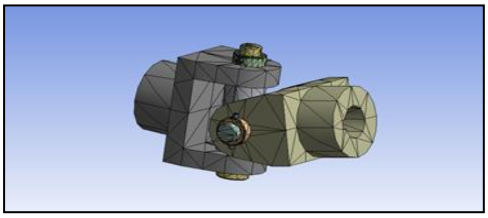


Fig 1(30 mm mesh size)

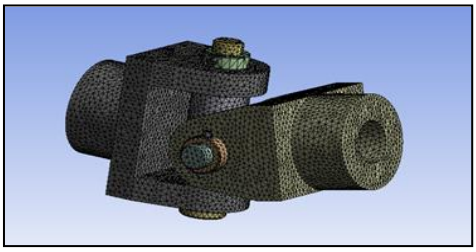


Fig 2 (3 mm mesh size)

4. RESULT

4.1 Equivalent Stress (Vonmises stress)

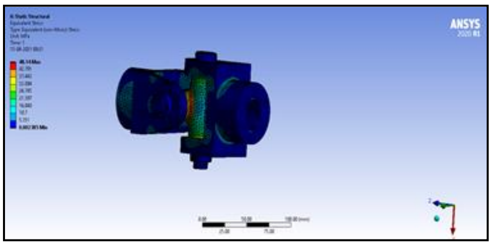


Fig 3 (3 mm mesh size)

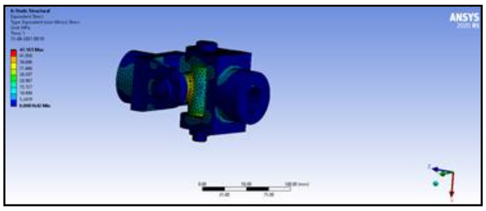


Fig 4 (4 mm mesh size)

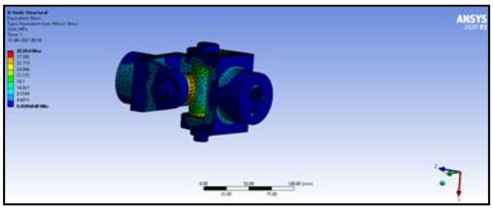


Fig 5 (5 mm mesh size)

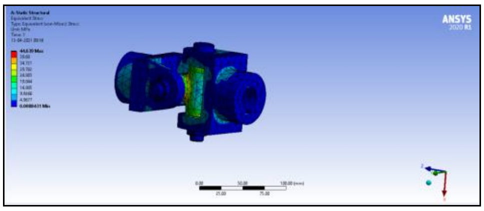


Fig 6 (10 mm mesh size)

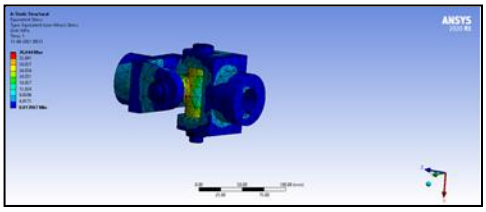


Fig 7(15 mm mesh size)

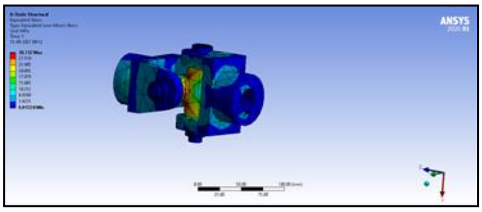


Fig 8(20 mm mesh size)

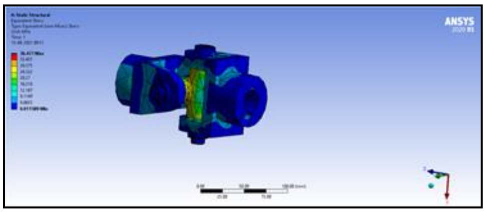


Fig 9(25 mm mesh size)

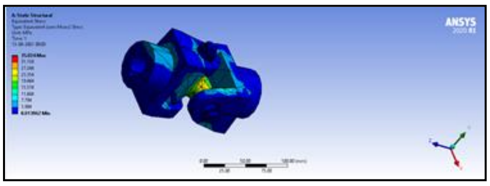
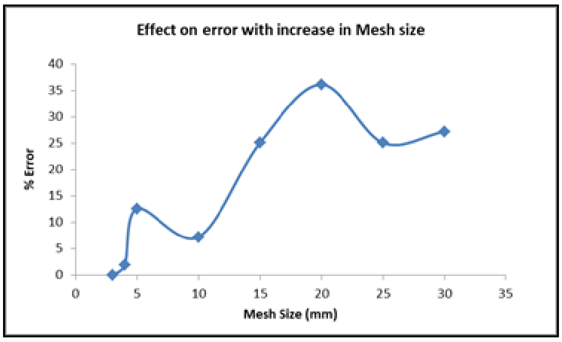


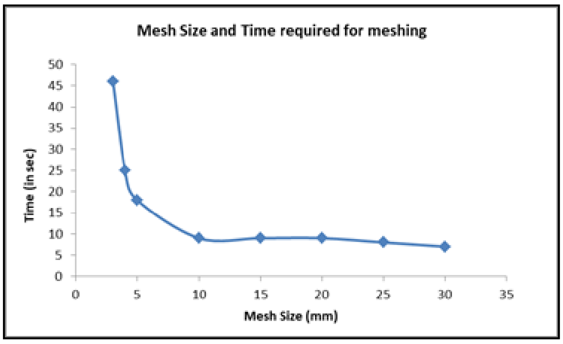
Fig 10(30 mm mesh size)

Table 2: Stress result and error

Mesh size(mm)	Equivalent Stress (Mpa)	Error %	Time required (sec)
30	35.024	27.24	7
25	36.477	25.12	8
20	30.723	36.17	9
15	36.044	25.13	9
10	44.639	7.27	9
5	42.064	12.62	18
4	47.165	2.02	25
3	48.14	0	46



Graph 1



Graph 2

4.2 Deformation

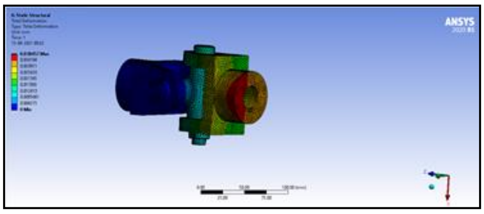


Fig 11 (3 mm mesh size)

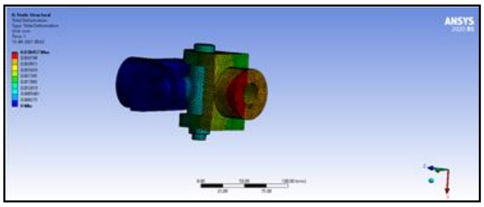


Fig 12(4 mm mesh size)

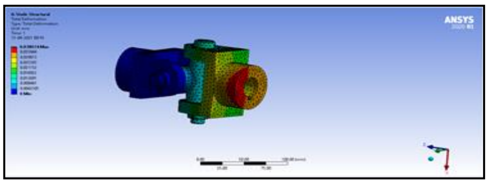


Fig 13 (5 mm mesh size)

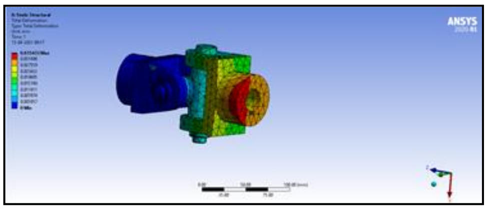


Fig 14 (10 mm mesh size)

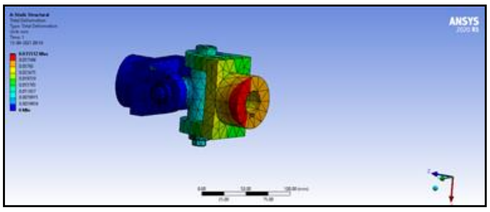


Fig 15 (15 mm mesh size)

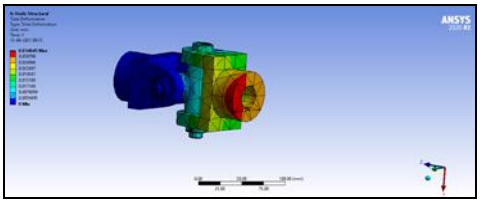


Fig 16 (20 mm mesh size)

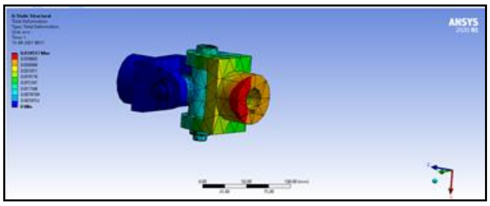


Fig 17(25 mm mesh size)

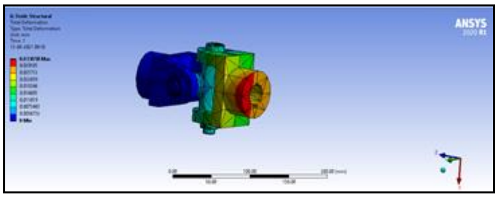
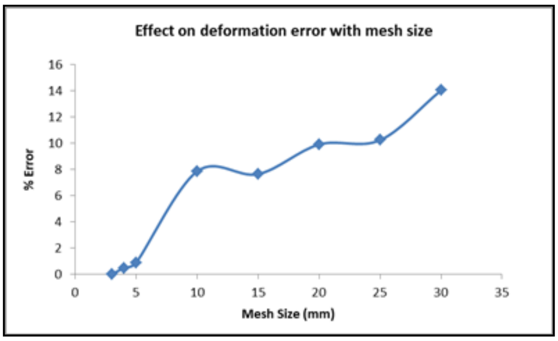


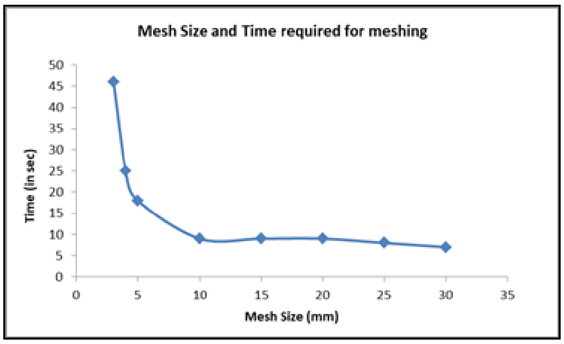
Fig 18 (30 mm mesh size)

Table 3: Deformation result and error

Mesh size(mm)	Deformation(mm)	Error %	Time required (sec)
30	0.033058	14.03	7
25	0.034517	10.24	8
20	0.034645	9.91	9
15	0.035512	7.65	9
10	0.035433	7.86	9
5	0.038074	0.9	18
4	0.038262	0.5	25
3	0.038457	0	46



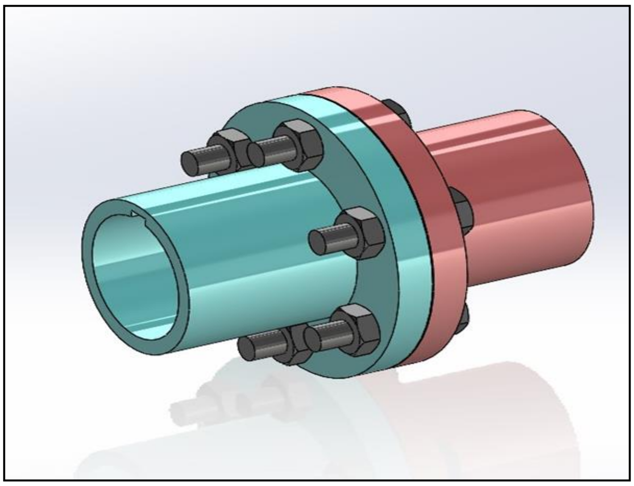
Graph 3

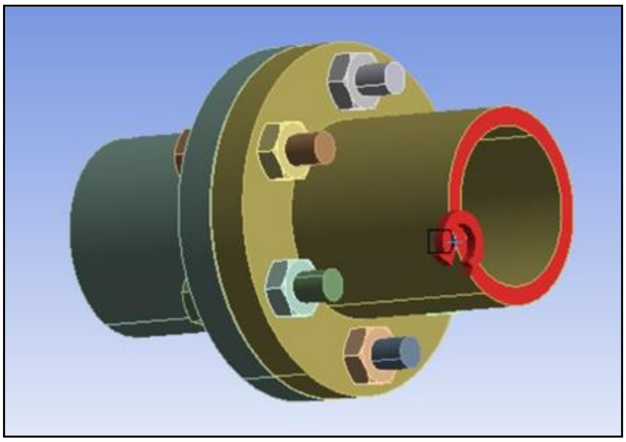


Graph 4

5. PROBLEM STATEMENT COMPONENT 2: FLANGE COUPLING

For analysis Flange Coupling assembly is taken.
The component below in made in Solid-works.
One end of component is fixed and other end is under moment of 5000 N-m. The material used is structural steel.


Model from solidworks


Moment application

6. RESULT

6.1 Equivalent Stress (Vonmises stress)

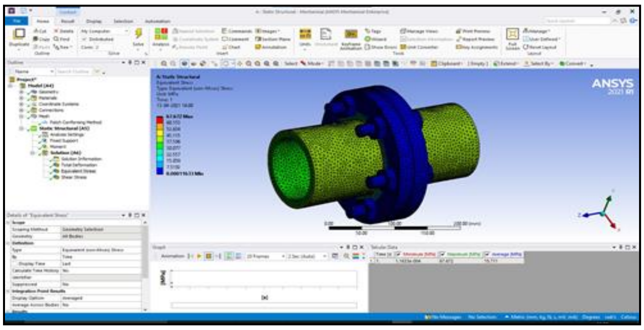


Fig 19 (3 mm mesh size)

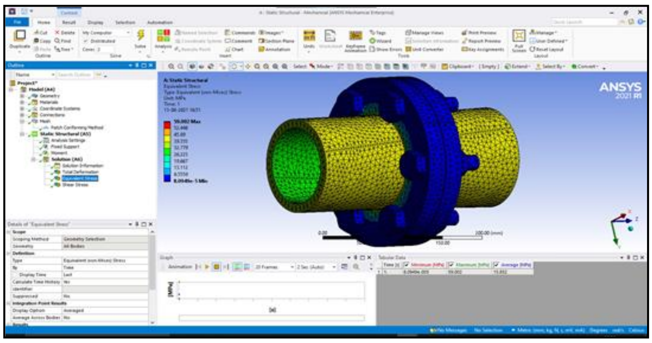


Fig 20 (4 mm mesh size)

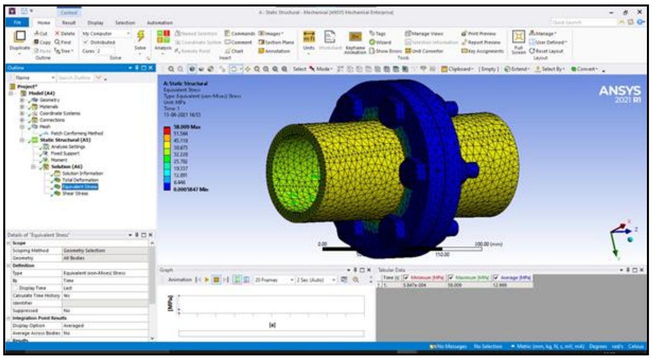


Fig 21 (5 mm mesh size)

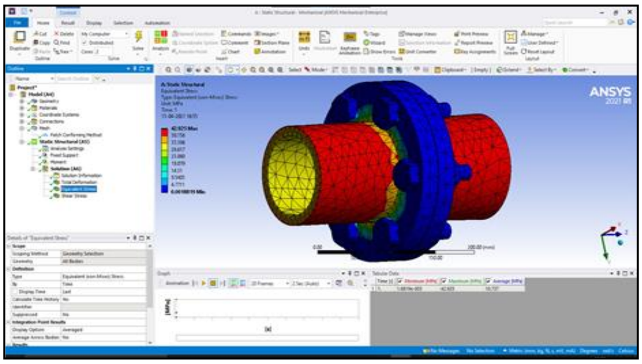


Fig 22 (10 mm mesh size)

Table 4: Stress result and error

Mesh Size	Equivalent Stress (MPa)	% Error in Stress	Time required (sec)
3	67.672	0	85
4	59.002	14.69441714	33
5	58.009	16.65776	25
10	42.925	57.65171811	14
15	41.32	63.77541142	11
20	41.263	64.00164797	11
25	42.168	60.481882	6
30	41.184	64.31623932	5

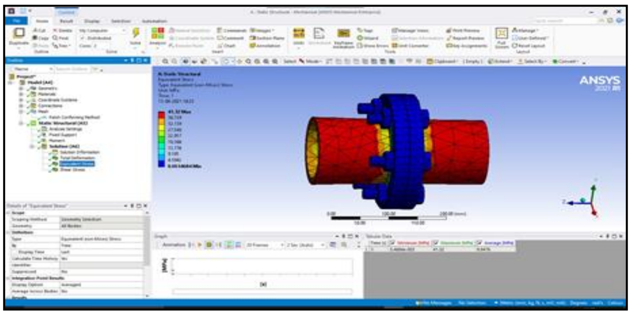


Fig 23 (15 mm mesh size)

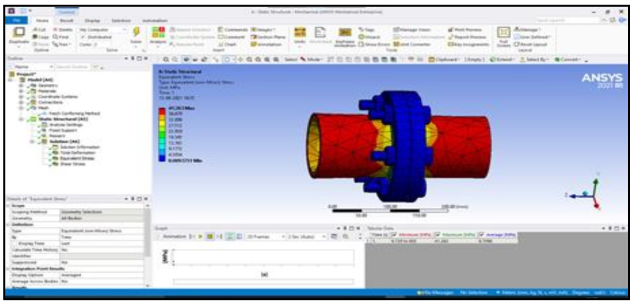


Fig 24 (20 mm mesh size)

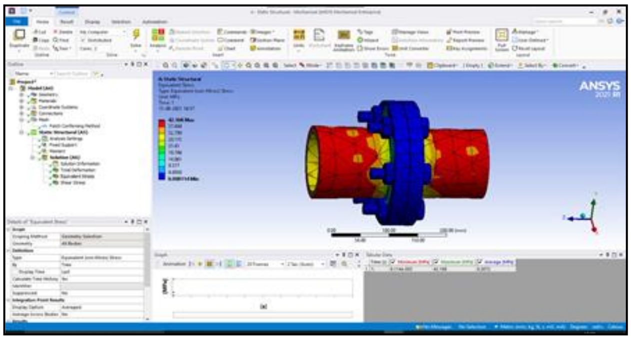


Fig 25 (25 mm mesh size)

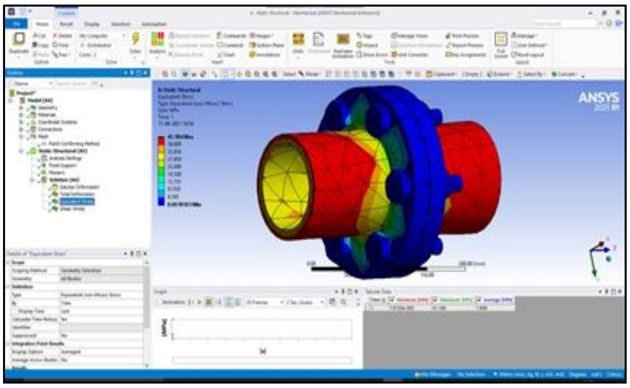
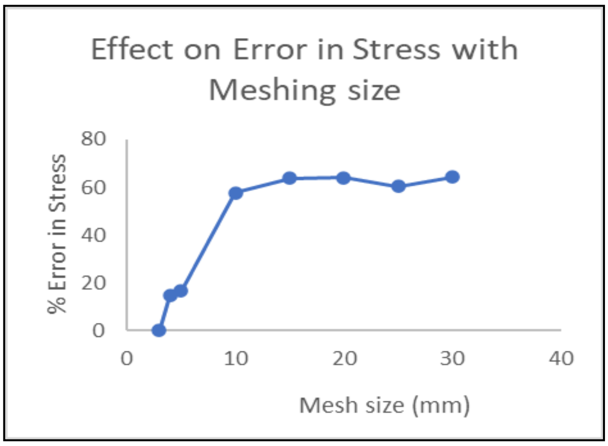
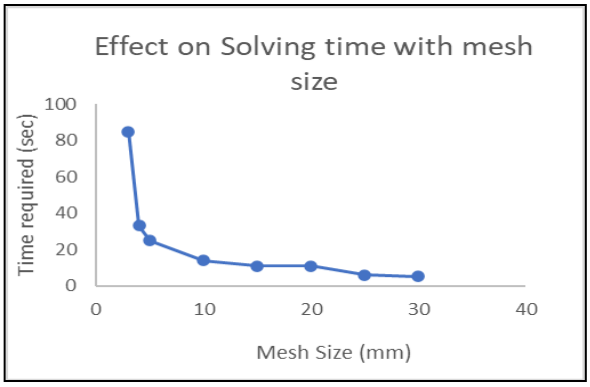


Fig 26 (30 mm mesh size)



Graph 5



Graph 6

6.2 Deformation

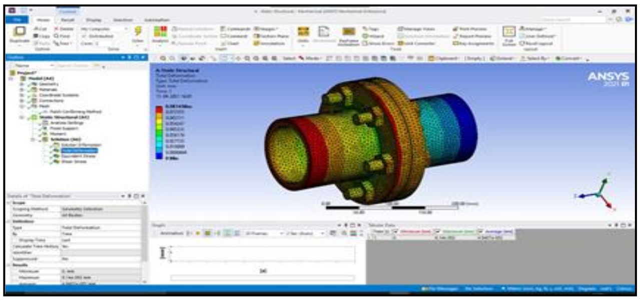


Fig 27 (3 mm mesh size)

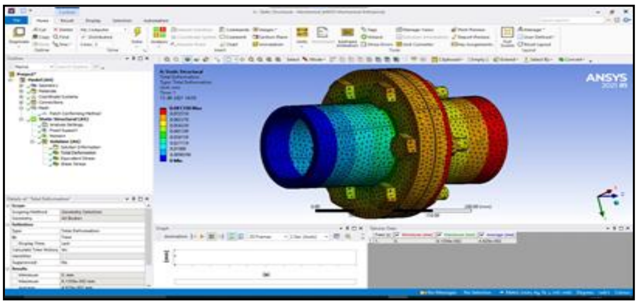


Fig 28 (4 mm mesh size)

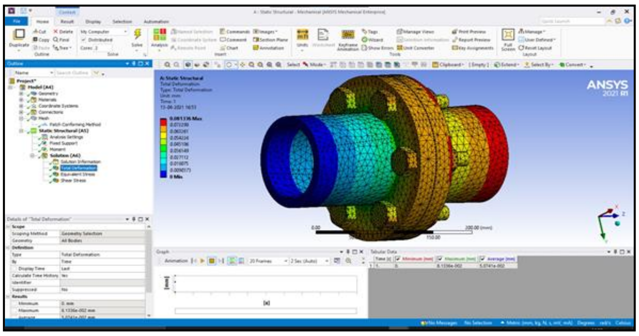


Fig 29 (5 mm mesh size)

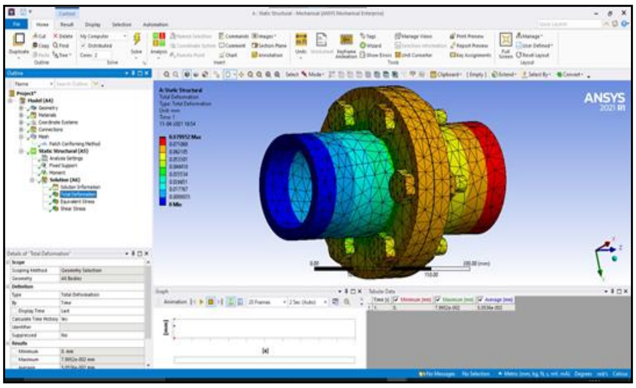


Fig 30 (10 mm mesh size)

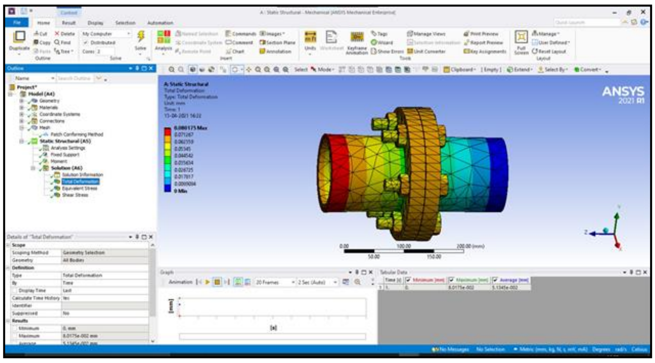


Fig 31 (15 mm mesh size)

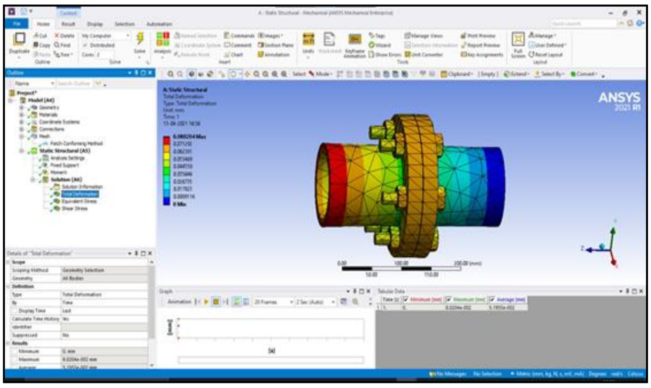


Fig 32 (20 mm mesh size)

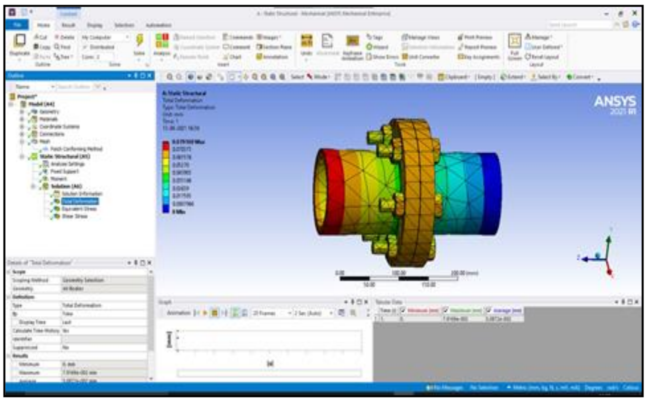


Fig 33 (25 mm mesh size)

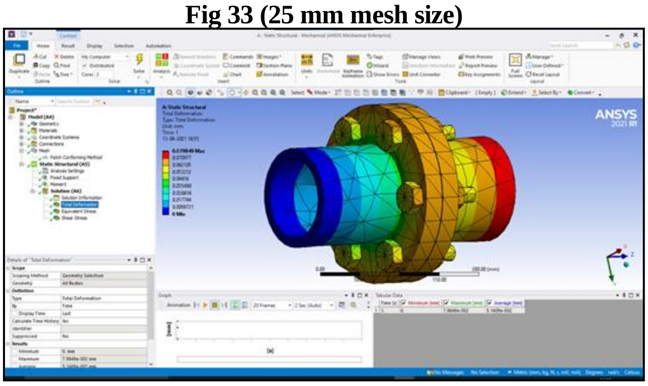
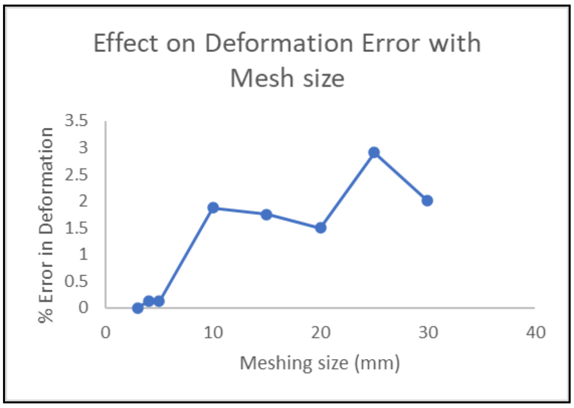


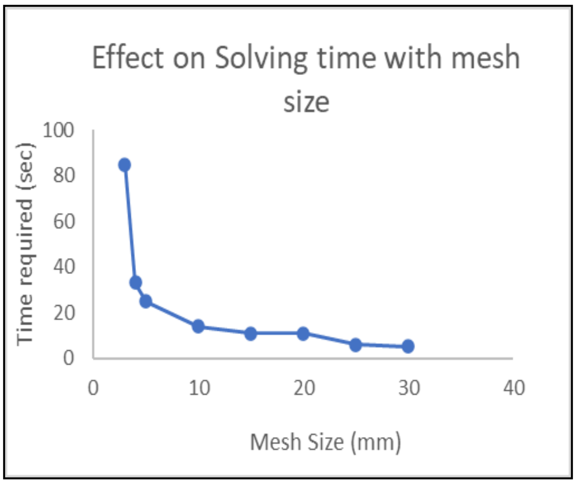
Fig 34 (30 mm mesh size)

Table 5: Deformation result and error

Mesh Size	Deformation (mm)	% Error in Deformation	Time required (sec)
3	0.0814	0	85
4	0.0813	0.12300123	33
5	0.0813	0.12300123	25
10	0.0799	1.877346683	14
15	0.08	1.75	11
20	0.0802	1.496259352	11
25	0.0791	2.907711757	6
30	0.0798	2.005012531	5



Graph 7



Graph 8

7. RESULTS AND DISCUSSION

It is observed from the result table of both the components that the values of equivalent stress and deformation differ according to the mesh size.

We can observe that as the mesh size varies result changes.

Percentage Error in equivalent stress is far more than deformation. According to FEA theory, deformation prediction is more accurate than stress.

8. CONCLUSION

Accuracy in result increases with decrease in mesh size, but computation time in also increased exponentially. Error in stress is due to stress concentration in component. Mesh size of 10 mm for these components shows a balance between Error and computation time.

9. REFERENCES

[1] C. S. Kim, K. S. Hong, and M. K. Kim, Nonlinear robust control of a hydraulic elevator, Control Engineering Practice, 13 (6) (2005) 789-803.

[2] Weibing Liu, MamtiminGeni, and Lie Yu, “Effect of Mesh Size of Finite Element Analysis in Model Analysis Periodic Symmetric Struts Support,Key Engineering Materials” Vol.462-463,2011,pp 1008-1012.

[3] Victor Debnath, Bikramjit Debnath “Deflection And Stress Analysis Of Beam On Different Elements Using Ansys APDL”,IJMET,vol.5,june 2011,pp 70-79.

[4] Nitin S. Godbole, Practicle finite element analysis, First edition, Finite to Infinite, 2008.

