INTERNATIONAL JOURNAL OF ADVANCE RESEARCH, IDEAS AND INNOVATIONS IN TECHNOLOGY

IJARIIT

ISSN: 2454-132X
Impact Factor: 6.078
(Volume 7, Issue 2 - V7I2-1376)
Available online at: https://www.ijariit.com

Comparative analysis of hexahedral and tetrahedral mesh of guide bracket

Ashay Katrojwar
ashay.21810435@viit.ac.in
Vishwakarma Institute of Information Technology, Pune, Maharashtra

ABSTRACT

In the following paper an effort has been made to do the comparative analysis of hex mesh and tetrahedral mesh of a guide bracket to decide which mesh type to prefer in order to get accurate result in considerably less time. To analyze whether there is significant impact of using hex dominant mesh and tetrahedral mesh. The bracket used in analysis is modelled using solid works 2018 and the finite element analysis is done using "ANSYS 2020 R2 student workbench". From the finite element analysis, I witnessed that there was significant difference in processing time of two types of mesh and other parameters such as stress and strain were also having noticeable increase with respect to results in tetrahedral mesh. The results will thus help to gain knowledge on which mesh is best for future meshing on various models.

***Keywords**: ANSYS, FEA, Tetrahedral, Hexahedral, Mesh*

1. INTRODUCTION

The finite element analysis is computational method which uses mathematical models to obtain solution by discretization of model in various small divisions called elements. FEA help us to analyze flaws in our current design so that we can recuperate our design. The finite element analysis basically involves three steps that are pre-processing, processing and post processing. Processing involves developing of a 3d model or assembly, discretization of part by performing suitable meshing and providing boundary conditions to our model such as forces, stress and pressure. Processing involves use of mathematical and numerical models to get required solutions for various parameters provided by the user. Post Processing involves displaying of results and graphs with user friendly contour plots which helps to identify maximum, minimum and average of various solutions. There are numerous software's available for finite element analysis such as Ansys, LS Dyna, Abaqus etc. The FEA software used in this paper is Ansys. Ansys has huge variety of tools available for meshing such as mesh sizing, mesh refinement, face meshing, contact sizing and many more. Ansys by default generates tetrahedral mesh on use of generate mesh function. In general, tetrahedral mesh require less time for processing of results but the result obtained after analysis by using tetrahedral mesh are not as accurate as obtained by hex dominant mesh. The results obtained by using Finite element analysis are approximate results as FEA uses numerical methods which provide approximate solution of a problem.

2. METHODOLOGY

2.1 Modelling

The component analyzed in following paper is guide bracket. The guide bracket is modelled using 3d modelling software solid works 2018.

Fig. 1: Guide Bracket

2.2 Material Properties

The material properties provided to "guide bracket" for analysis are of structural steel due to its good physical properties. Some physical properties of structural steel are described below:

Density	7850 kg/m³
Youngs Modulus	2e+11 Pa
Thermal Conductivity	60.5 W/m·°C
Bulk Modulus	1.6667e+11
Poisson's Ratio	0.3
Shear Modulus	7.6923e+10
Thermal Conductivity	60.5 W/m*C
Specific Heat	434 J/kg·°C
Tensile Yield Strength	2.5e+08 Pa
Tensile Ultimate Strength	4.6e+08 Pa

2.3 Meshing

The model was meshed in ANSYS 2020 R2 academic workbench. The component was meshed using hexahedral elements and tetrahedron elements by using quadratic element order. The number of elements and nodes involved are displayed :

For tetrahedral:

node	8058
Elements	4194

For hexahedral:

node	12173
Elements	3677

2.4 Boundary Conditions

After meshing of the guide bracket the boundary conditions were defined which involved defining fixed support on the bracket and application of 2.5 KN load on negative y axis.

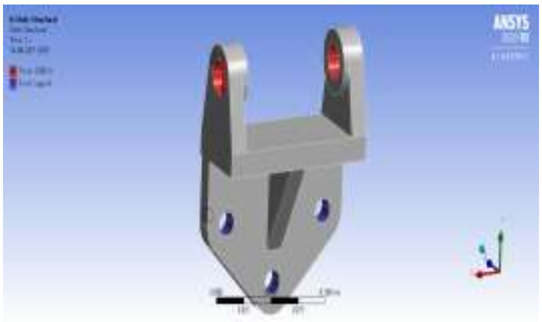

Fig. 1: Boundary Conditions

3. RESULTS AND DISCUSSIONS

For "*tetrahedral*" we obtained following results

Total Deformation	1.955e-005 m
Equivalent Elastic Strain	8.6936e-005
Equivalent Stress	1.7381e+007Pa

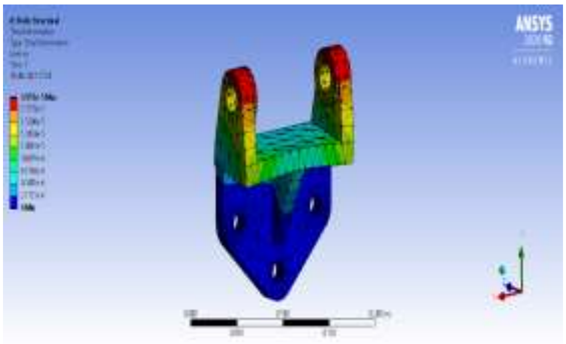

Fig. 3: Total Deformation (Tetrahedral)

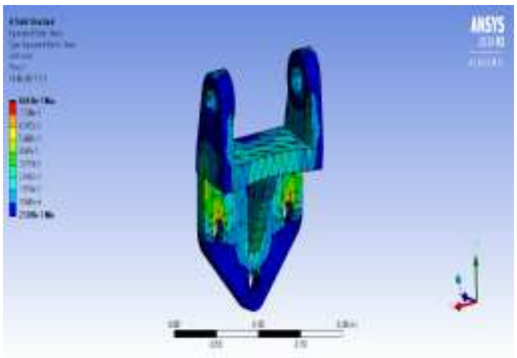

Fig. 4: Equivalent Elastic Strain (Tetrahedral)

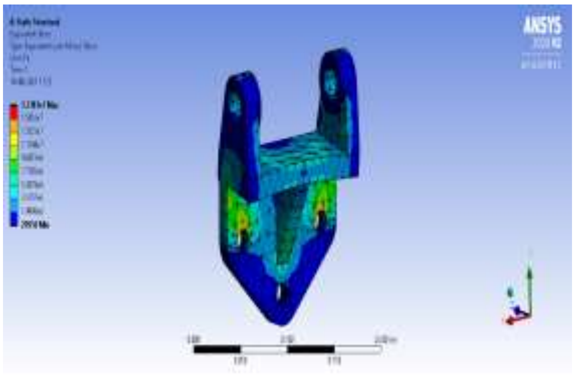

Fig. 5: Equivalent Stress (Tetrahedral)

For "*hexahedral*" mesh we obtained following results

Total Deformation	2.0059e-005 m
Equivalent Elastic Strain	9.5451e-005
Equivalent Stress	1.9083e+007 Pa

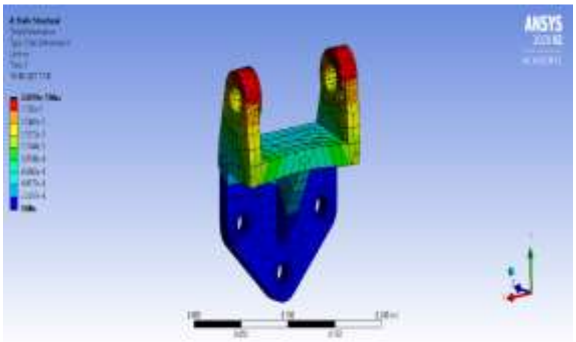

Fig. 6: Total Deformation (Hex)

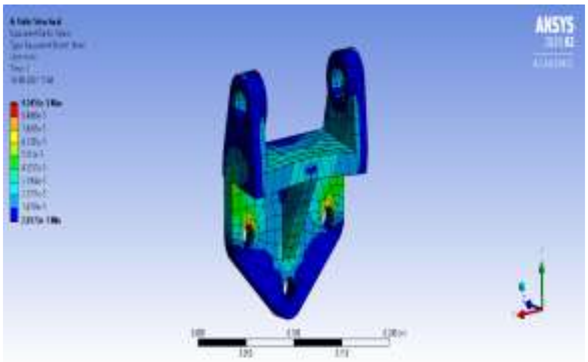

Fig. 7: Equivalent Elastic Strain (Hex)

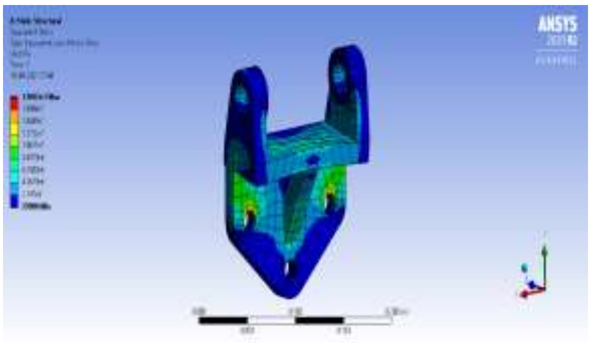

Fig. 8: Equivalent Stress (Hex)

The processing time of hex dominant mesh was more than tetrahedral mesh. It was noticed that there was no increase in elements in tetrahedral mesh and hex dominant mesh, but the number of nodes when bracket is meshed using hex dominant mesh rather than tetrahedral mesh increased by huge amount of 51%.

The total deformation obtained by hex dominant mesh is 3% more than that obtained by using tetrahedral mesh, but this difference is not so significant.

The equivalent elastic strain in hex dominant mesh was 10% more than tetrahedral mesh which is quite on the higher side.

The equivalent stress in hex dominant mesh was 9% more than that of tetrahedral mesh.

4. CONCLUSIONS

Hence comparative analysis of tetrahedral mesh and hexahedral mesh on guide bracket by using Ansys 2020 R2 workbench was performed. From the following analysis we found that the tetrahedral mesh takes less time for processing of results than hex dominant mesh due to large number of nodes.The results obtained by hex dominant mesh were overall slightly greater than that of tetrahedral mesh which is correct as hex dominant mesh involves a greater number of nodes and greater number of nodes depicts high accuracy. Thus a conclusion is made that for analyzing model requiring higher accuracy must be meshed using hexahedral mesh and model requiring quick results must be meshed using tetrahedral mesh.

5. REFERENCES

[1] Finite element modelling and simulation with ANSYS workbench by Xiaolin Chen-Yijun Liu

[2] Shashikant T. More, Dr R.S Bindu Effect of Mesh size on finite element analysis of plate structure.

[3] The finite element method in engineering, fifth edition by Singiresu S. Rao

[4] Finite element analysis by S.S. Bhavikatii

