International Research Journal of Innovations in Engineering and Technology (IRJIET)

ISSN (online): 2581-3048

Volume 5, Issue 5, pp 10-14, May-2021

https://doi.org/10.47001/IRJIET/2021.505002

Comparative Analysis of Tetrahedrons and Hexahedrons for Finite Element Mesh during Static Structural Analysis of Guide Bracket

[1]Sakshi Suryawanshi, [2]Abhishek Hange

[1,2]Department of Mechanical Engineering, Vishwakarma Institute of Information Technology, India

***Abstract* - In mechanical engineering a bracket is any simple component for fixing one part to another, usually larger part. To enable the outstretched arm and to support a greater weight, a bracket will have a third arm running diagonally between the horizontal and vertical arms, or the bracket may be a solid triangle. A simple rigid structure and the shape of an L, one arm is fixed to a vertical surface and the other one is projecting horizontally to support a shelf or other weight. This paper describes the analysis of mechanical component Guide Bracket in finite element mesh by the various methods. We have designed the guide bracket in Solid works software and are imported to Ansys workbench for finite element analysis (FEA). In FEA, the structural analysis is performed and the analysis is done. We applied some constraints like stress, deformation of different materials, toughness, elasticity and other attributes. There are different standards, styles, and shapes of brackets. Each bracket style has its specific requirement for mounting a component, or being suitable with a particular surface shape. We can design and manufacture any kind of metal bracket as per our need (so long as our design fits within our tooling requirements).**

Keywords: Finite element analysis, Static structural analysis, guide bracket.

I. INTRODUCTION

Brackets in general terms that considers a supporting system or a guiding system. In moderately long or vertical components brackets can be used to support and in mechanisms and machines brackets are guiding systems i.e. to join components at 90 degrees where L brackets are used. There are different brackets available or we can manufacture it as per our requirement. Ansys software is used to analyze computer models of structures, electronics, or machine components for analyzing strength, elasticity, strain, toughness, elasticity, temperature distribution, electromagnetism, fluid flow, and other attributes. It allows students to locate the problems in their designs. It will show you whether a product will break, wear out or work the way it was designed. It is also used for structural analysis to find out significant stress and receive data input from other tools such as fluid dynamic system and kinematic analysis etc. Each system and any software have its own advantages and disadvantages. In FEA, we have to go through the various phases of preprocessing, solution and post processing. The preprocessing is likely equal to seventy percent of the total effort it contains various steps have to give the input data, geometry, material properties and support conditions; after that the solution runner or solution program is five percent of total effort; all the boundary conditions effects, calculations and stress analysis is done. Then the third phase post-processing comes into the picture and has twenty-five percent of total effort. It completes the last phase by repeating the Analysis, refine mesh and verify the result. Therefore, these are all phases done in FEA. The brackets are made up of various materials. There are different types of brackets available in the market mainly L-Bracket, U-Bracket, Z-Bracket, Offset-Bracket and Gusset-Bracket.

II. METHODOLOGY

CAD model

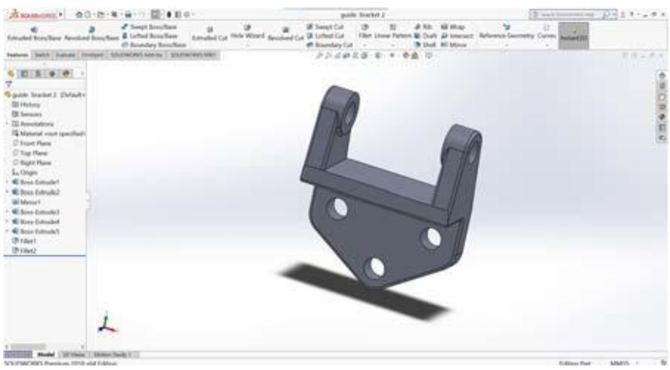

Figure 1 Solidworks design of guide bracket

To start with the analysis, the first step is to design a CAD model with accurate dimensions. This design of a guide bracket is done in Solidworks 2018. Once the designing is done, the file is saved to igs. Extension and is imported to Ansys workbench for further analysis. For this we are using ansys 19.2 version.

International Research Journal of Innovations in Engineering and Technology (IRJIET)

ISSN (online): 2581-3048

Volume 5, Issue 5, pp 10-14, May-2021

https://doi.org/10.47001/IRJIET/2021.505002

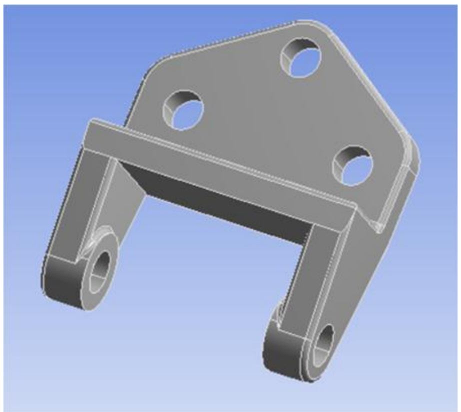

Figure 2: Guide Bracket

To commence with the analysis following steps are done.

1) Open Static Structural window from analysis system.
2) Go to engineering data and select the required material.
3) Select geometry of the component from the browser.
4) Select Model

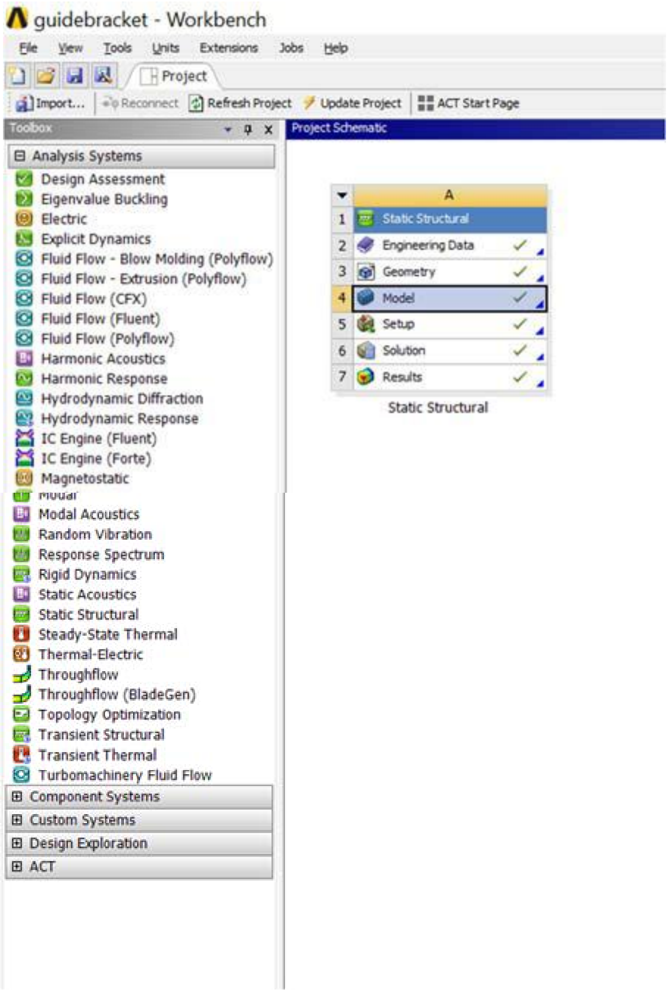

Figure 3: Static Structural

Thereafter, a new window of static structural mechanical is opened. The left side window gives all the content like geometry, material, and coordinate system which is selected. Further, for mesh select the Mesh option and the overall component (Ctrl + A). Right click the mesh option to generate mesh. Once Meshing is done, the component is ready for static structural analysis. The green ticks describe the accuracy of the analysis as shown in fig. 4.

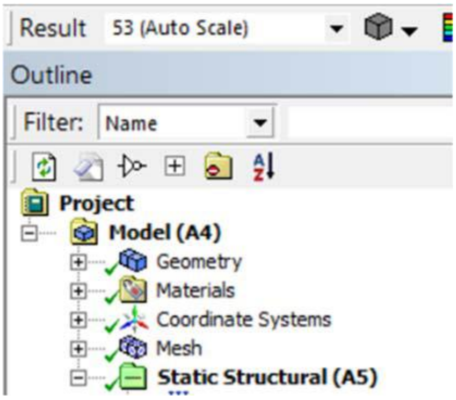

Figure 4: Analysis Window in Static Structural Mechanical

For the FEA of guide bracket we have chosen Stainless Steel (304L) (austenitic). Following are the properties of Stainless Steel 30L.

Properties	Stainless Steel 304L
Tensile Yield Strength	2.43e+08 Pa
Tensile Ultimate Strength	5.46e+08
Hardness	215 max HB
Density	7900 kg/m^3
Young's Modulus	1.98e+11Pa
Thermal Conductivity	15 W/m deg C
Specific Heat	510 J/kg deg C

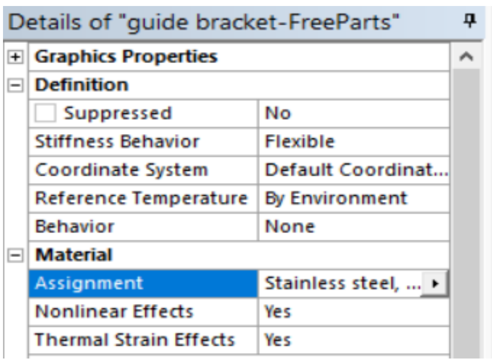

Figure 5: Geometry Details

ISSN (online): 2581-3048
Volume 5, Issue 5, pp 10-14, May-2021
https://doi.org/10.47001/IRJIET/2021.505002

After generating Mesh, we have to proceed with static structural analysis. Here we have to check the total deformation, strain, stress etc. of the given body. Select fixed support and then give the required forces and direction to the body.

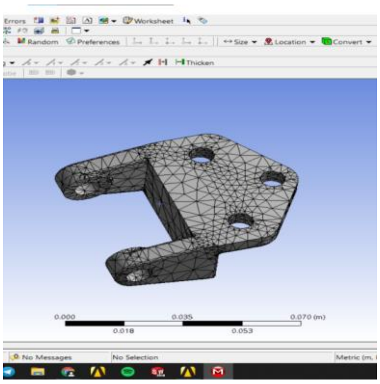

Figure 6: Meshing

Force Applied to both hanging ends is 1000 N. It is along Y direction and is pointing outwards (red color) as shown in fig. 6.

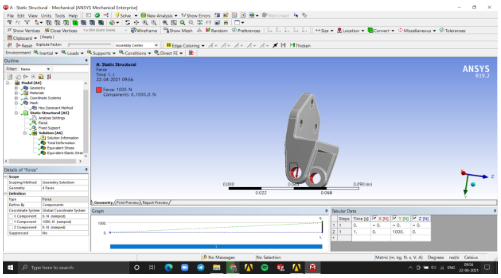

Figure 7: Forces

For the guide bracket, we have performed the FEA by comparing the values of deformation, stress and strain in the same body in tetrahedron method and hex dominant method.

For Proceeding with these two methods the steps are similar.

Mesh < Insert < Method< select the body < apply geometry <tetrahedron method < Tetrahedron < Generate Mesh

Tetrahedron Method

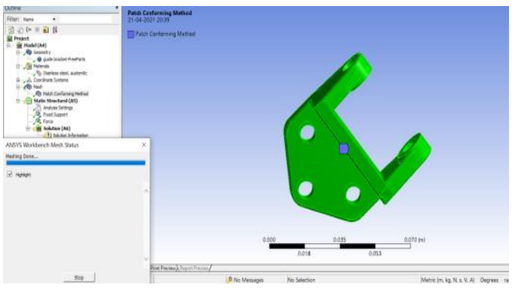

Figure 8: Tetrahedron meshing

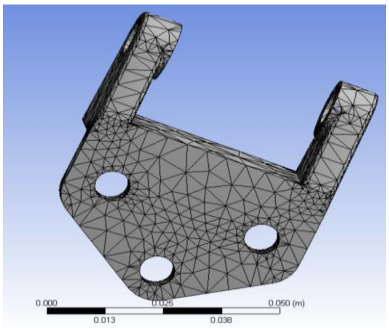

Figure 9: Tetrahedron Mesh

Statistics	
Nodes	12498
Elements	6821

The structural analysis of the guide bracket is done further by means of deformation, stress and strain.

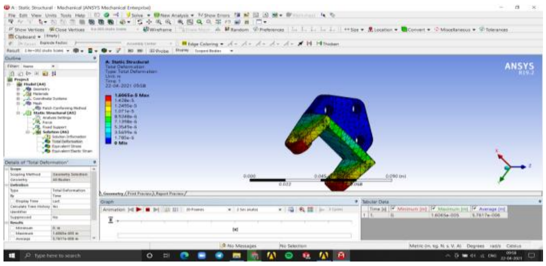

Figure 10: Total deformation in tetrahedron method

The red colour describes the maximum deformation and the blue minimum deformation.

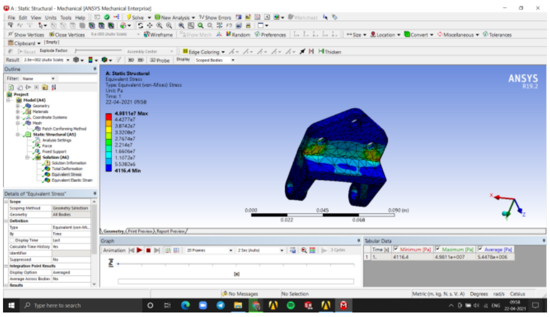

Figure 11: Equivalent stress in tetrahedron method

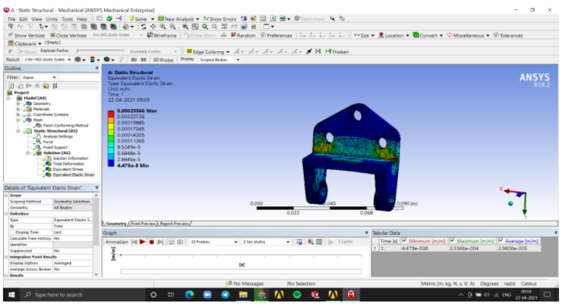

Figure 12 Equivalent strain in tetrahedron method

International Research Journal of Innovations in Engineering and Technology (IRJIET)

ISSN (online): 2581-3048

Volume 5, Issue 5, pp 10-14, May-2021

https://doi.org/10.47001/IRJIET/2021.505002

Analysis for tetrahedron method

Analysis Type	Min	Max
Total deformation	0	1.6065 e-005
Equivalent Stress	4116.4	4.9811 e+007
Equivalent elastic strain	4.479 e-008	2.5566 e-004

The above table gives the values obtained from various analysis types in tetrahedron method.

Hex dominant method

For the hex dominant method the same steps are followed the values are obtained accordingly.

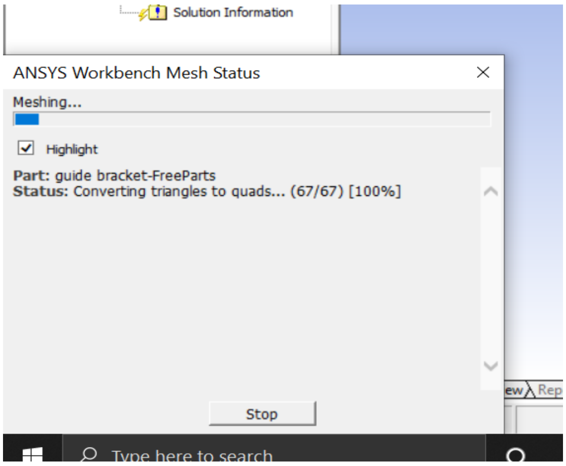

Figure 13: Mesh Window

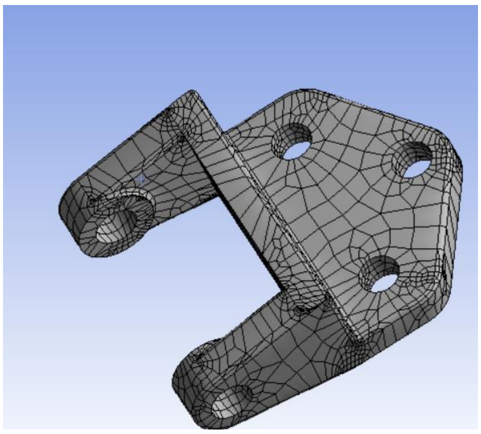

Figure 14: Hex dominant method Mesh

The structural analysis of the guide bracket in hex dominant method is done further by means of deformation, stress and strain.

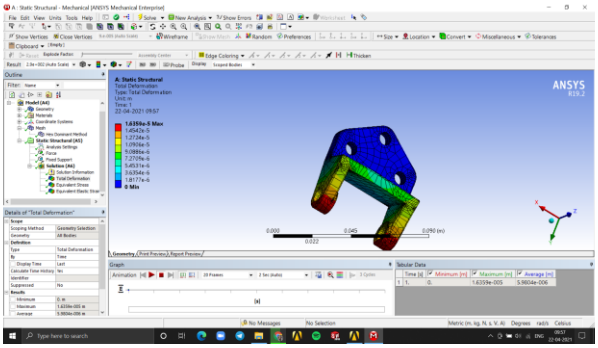

Figure 15: Total deformation in hex dominant method

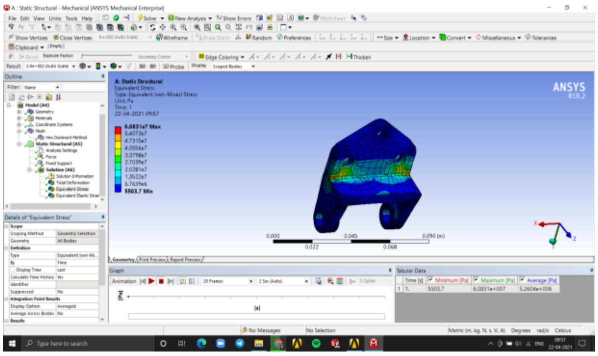

Figure 16: Equivalent stress in hex dominant method

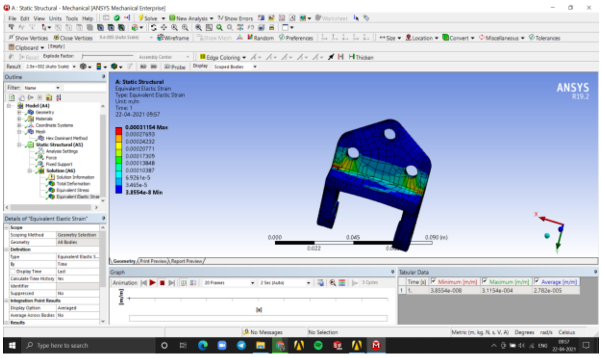

Figure 17: Equivalent strain in hex dominant method

Analysis for hex dominant method

Analysis Type	Min	Max
Total deformation	0	1.6359 e-005
Equivalent Stress	5509.7	6.0831 e+007
Equivalent elastic strain	3.8554 e-008	3.1154 e-004

The structural analysis of the guide bracket in hex dominant method is done further by means of deformation, stress and strain.

ISSN (online): 2581-3048
Volume 5, Issue 5, pp 10-14, May-2021
https://doi.org/10.47001/IRJIET/2021.505002

We can now easily compare the difference of the stress, strain and the deformation values of the component between tetrahedron and hex dominant method.

III. CONCLUSION

1. There is a slight variation of values when FEA is done by comparing the fixed supports in ansys 19.2.
2. The FEA domain is used for analysing the variation of values in the design parameters of the given component.
3. The total deformation of tetrahedron and hex dominant method when compared is negligible.
4. The minimum stress value differs a lot where the minimum tetrahedron stress value is less than that of the hex dominant stress value.
5. Whereas the maximum value ranges close in both the methods.
6. There is a difference in the equivalent strain values also, where the minimum strain of hex dominant is more than that of tetrahedron and maximum value of hex dominant method is less than tetrahedron.

REFERENCES

[1] https://www.protocase.com/blog/
[2] https://www.solidworks.com/domain/design-engineering
[3] https://www.researchgate.net/publication/284184006_Modeling_and_Analysis_of_Guide_Rail_Brackets_and_Attaching_Parts

AUTHORS BIOGRAPHY

Sakshi Suryawanshi, College: Vishwakarma Institute of Information and Technology, Branch: Mechanical Engineering, Year: Third Year Mail id:sakshi.21810905@viit.ac.in

Abhishek Hange, College: Vishwakarma Institute of Information and Technology, Branch: Mechanical Engineering, Mail id:abhishek.21810920@viit.ac.in

Citation of this Article:

Sakshi Suryawanshi, Abhishek Hange, “Comparative Analysis of Tetrahedrons and Hexahedrons for Finite Element Mesh during Static Structural Analysis of Guide Bracket” Published in *International Research Journal of Innovations in Engineering and Technology - IRJIET*, Volume 5, Issue 5, pp 10-14, May 2021. Article DOI https://doi.org/10.47001/IRJIET/2021.505002

****