Comparative Study of Effect of Mesh and Analysis of Lever Bracket

Sourav Jadhav[1]
Vishwakarma Institute of Information Technology, Pune

Atul Kulkarni[2]
Vishwakarma Institute of Information Technology, Pune

***

Abstract- This paper contains the design and analysis of lever bracket for the vehicle considering changes in structure of program element various software's like SIMSCALE, COMSOL Multiphysics, MATLAB, SOLLIDWORKS and most popular ANSYS. Finite Element Analysis (FEA) is a method used to calculate forces, moments, heat, fluid flow, resistance to acceleration, and the physical result. In short, method is used to predict the physical behavior of product. In this research paper, ANSYS is used for static structural analysis of lever bracket. FEA increased accuracy due to the analysis of any physical stress that might affect the design. The result of research paper shows change in Total Deformation, Difference in stress and strain. As the time passes the use FEA analysis in mechanical world is necessarily increased, hence the design should be analyzed so that we come to know what external factor would affect the design, so in this project analysis of Lever bracket we can define their stress strain etc.

Index Terms- Lever Bracket, Bracket, Meshing

I. Introduction

Bracket is Mechanical part. Any part that carried out function of fixing one part to another (usually large) part. Brackets are used to support beams, conduits, pipes etc. It is usually made of stainless steel or aluminum. The lever bracket is used in automobile sector to connect two parts. Brackets in general terms you can consider as a supporting system or a guiding system. In relatively 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 90deg you can use L bracket. There are different brackets available or we can manufacture it as per our requirements. It is fixed part in assembly.

II. Objective-

The objective is to design a Lever Bracket using Catia and carry out the finite element analysis (FEA) on the prepared model using ANSYS. Thus, we obtained the values of stress and strain, total deformation.

III. Modeling and Analysis

The work starts with designing the lever bracket in CATIA V5. Here CATIA V5 is used for modelling because it is easy to understand and supports 2D and 3D conversion including digital sketching. After the design analysis was done keeping in mind the material is stainless steel. The .igs file is imported from CATIA and use for analysis in ANSYS. Analysis is done in comparison between 4 types- Tetrahedrons linear and quadratic meshed model and Hex dominant linear and quadratic meshed model. While analyzing model in ANSYS different meshing methods to be done like Tetrahedron linear and Quadratic, Hexahedron Linear and Quadratic and the results were shown.

IV. Lever Bracket Design

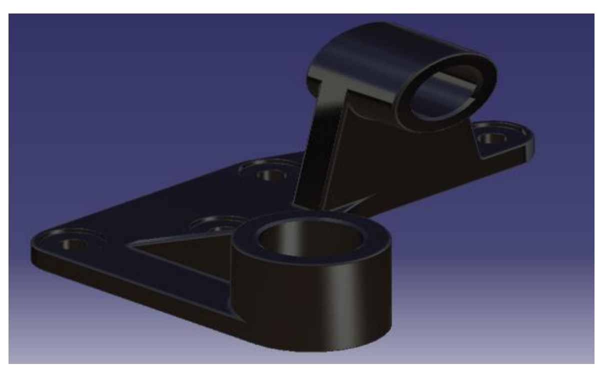

Fig. 1: Design of Lever bracket

V. Material

Based on the type of loading stainless steel is a perfect material for lever bracket and was decided that the desired material needed the following material properties:

1) Strong
2) Relatively ductile
3) Biocompatible
4) Relatively cheap
5) Reasonable corrosion resistance
6) Has lot of flexibility and strength to allow stable and thinner construction.

Details of "Structural Steel"	
Common Material Properties	
Density	7.85e-06 kg/mm³
Young's Modulus	2e+05 MPa
Thermal Conductivity	0.0605 W/mm·°C
Specific Heat	4.34e+05 mJ/kg·°C
Tensile Yield Strength	250 MPa
Tensile Ultimate Strength	460 MPa
Nonlinear Behavior	False
Full Details	Click To View Full D...
Statistics	
Assigned Bodies	1

VI. Meshed Model and Comparison

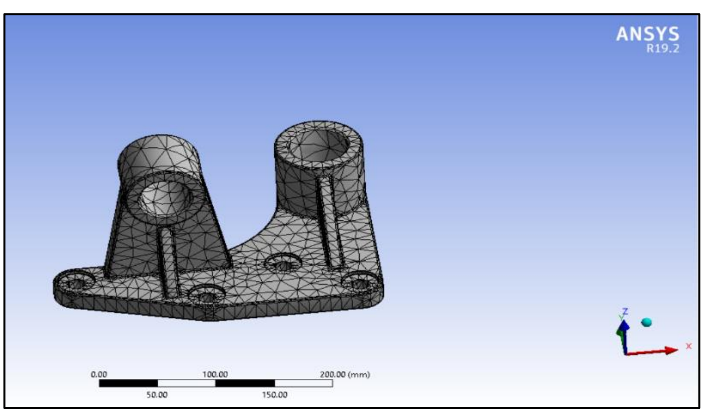

Fig 2. Meshed Model

Stainless Steel

Here, we make use of unstructured tetrahedron mesh since there are a lot of curvatures. Also, the uniform mesh is not useful in this case. So Tetrahedron is best for discretizing complex geometries but the computational time to too large for tetrahedron mesh.

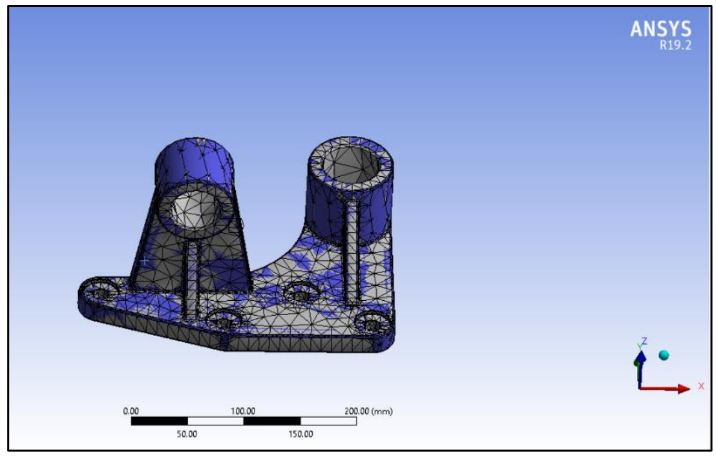

Fig. 3 Tetrahedrons meshed model

Full part tetrahedron Mesh
No of Nodes – 26,976
No of Elements – 14660

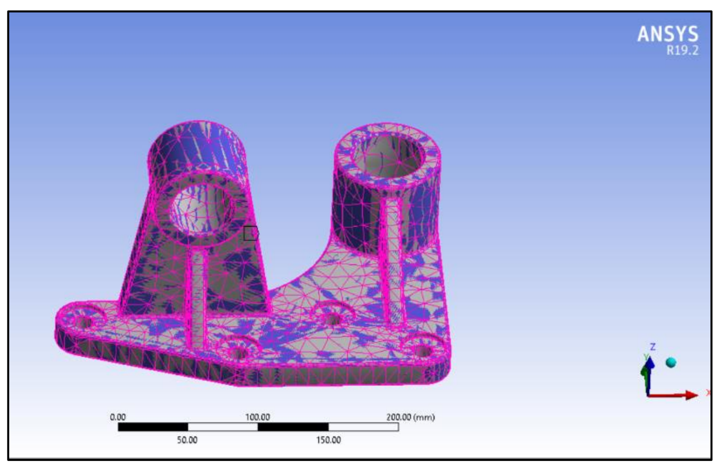

Fig. 4 Hex dominant meshed model

Full part Hex dominant Mesh
No of Nodes – 13280
No of Elements – 23898

Here we can observe that combination of Tetrahedron at the nonlinear part and hexahedron at the linear part is much more accurate in meshing. Also, better as compared to full body Tetrahedron or hex dominant mesh.

VII. Analysis

Total Deformation:

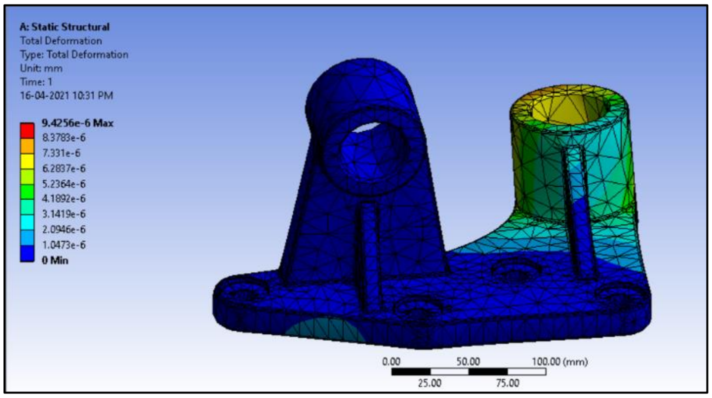

Fig. 5 Total Deformation in Tetrahedral Linear

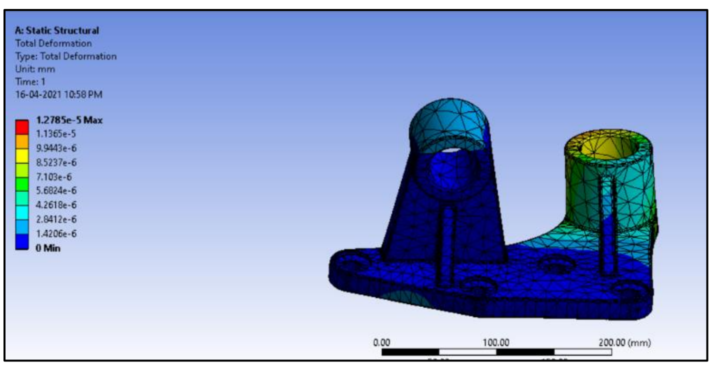

Fig. 6 Total Deformation in Tetrahedral Quadratic

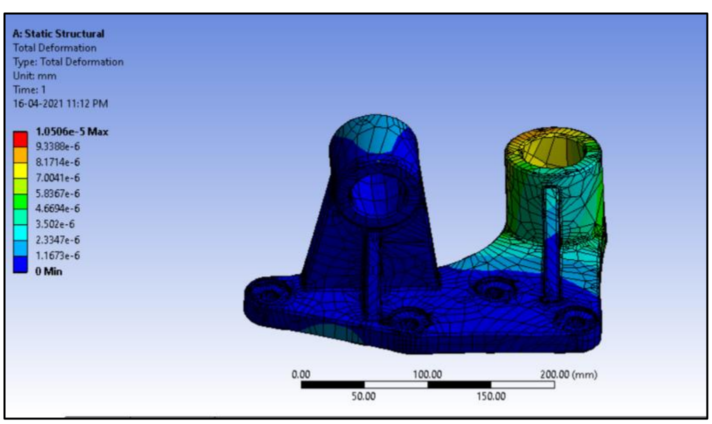

Fig. 7 Total Deformation in Hex Dominant Linear

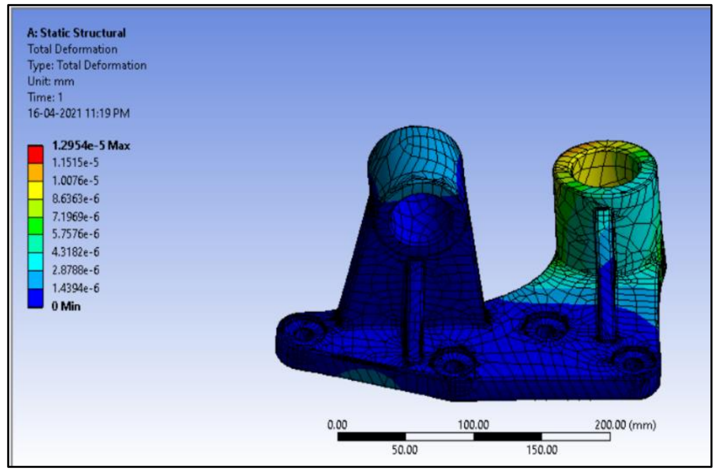

Fig. 8 Total Deformation in Hex dominant Quadratic

Equivalent Stress (MPa):

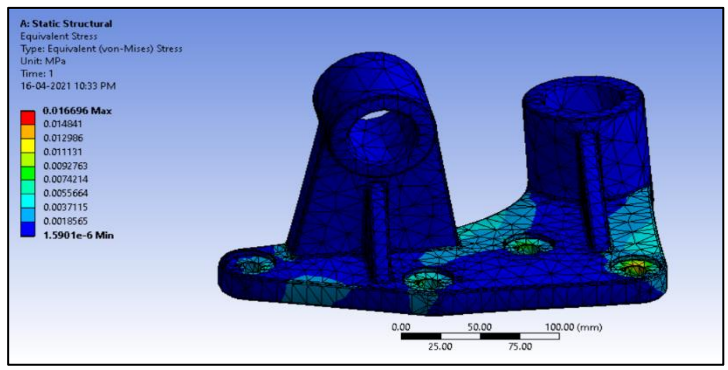

Fig. 9 Equivalent Stress in Tetrahedral Linear

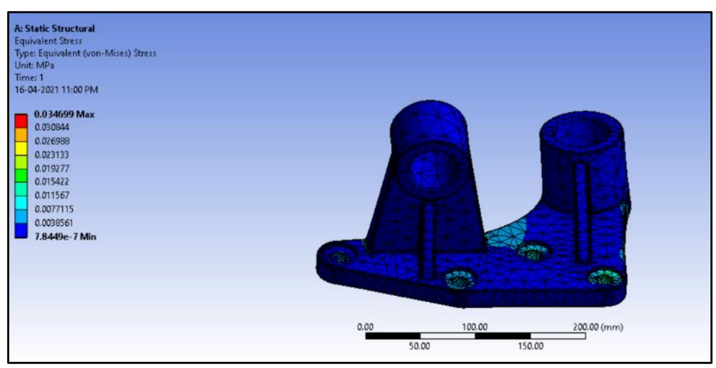

Fig. 10 Equivalent Stress in Tetrahedral Quadratic

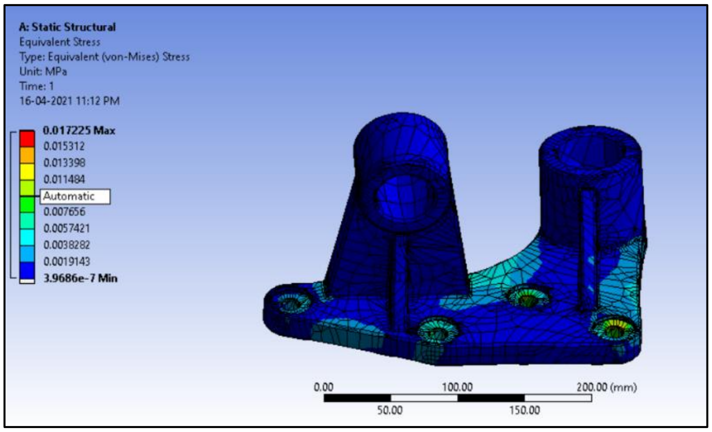

Fig. 11 Equivalent Stress in Hex Dominant Linear

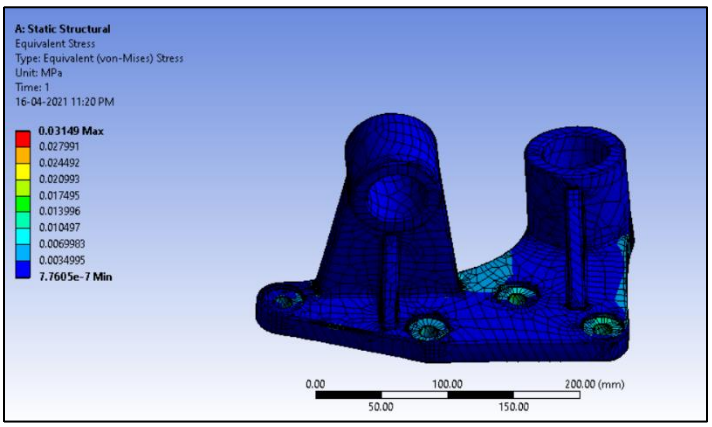

Fig. 12 Equivalent Stress in Hex Dominant Quadratic

Equivalent Strain:

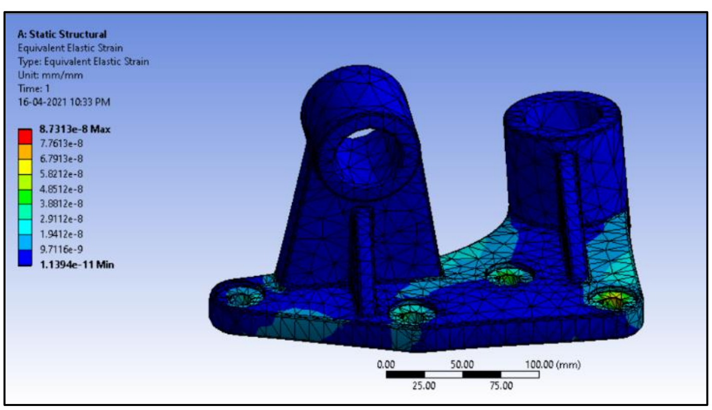

Fig. 13 Equivalent Strain in Tetrahedral Linear

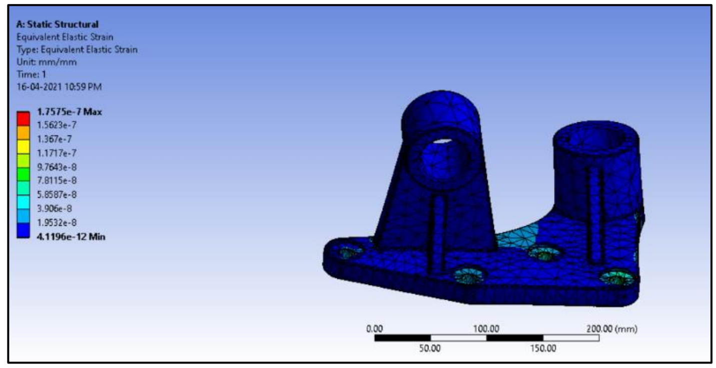

Fig. 14 Equivalent Strain in Tetrahedral Quadratic

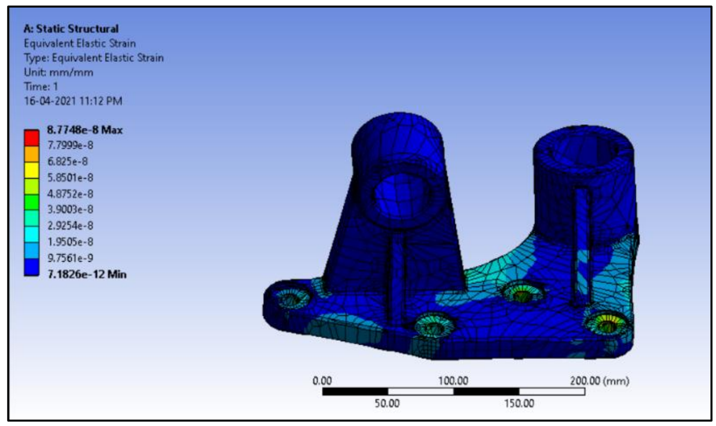

Fig. 15 Equivalent Strain in Hex Dominant Linear

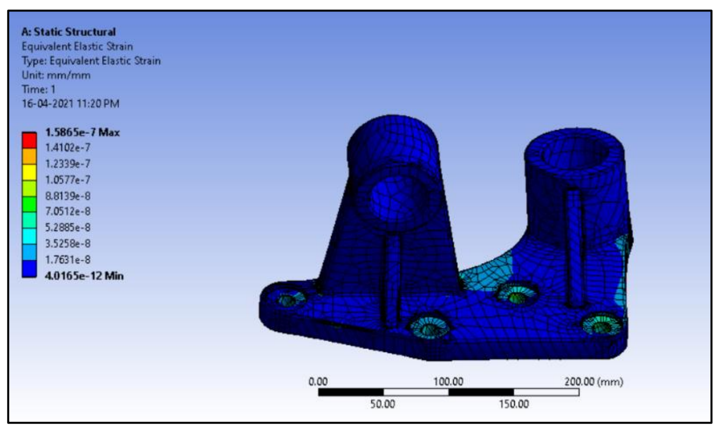

Fig. 16 Equivalent Strain in Hex Dominant Quadratic

VIII. Result

Type	Min	Max
Tetrahedral Linear	0.	9.4256e-006
Tetrahedral Quadratic	0.	1.2785e-005
Hex Dominant Linear	0.	1.0506e-005
Hex dominant Quadratic	0.	1.2954e-005

It is observed the tetrahedral linear has maximum deformation 9.4256e-006 as compare to other.

Type	Min	Max
Tetrahedral Linear	1.5901e-006	1.6696e-002
Tetrahedral Quadratic	7.8449e-007	3.4699e-002
Hex Dominant Linear	3.9686e-007	1.7225e-002
Hex dominant Quadratic	7.7605e-007	3.149e-002

According to result Tetrahedral Quadratic has maximum deformation 7.8449e-007 MPa as compare to other and Tetrahedral Linear has minimum deformation 1.5901e-006 MPa.

Type	Min	Max
Tetrahedral Linear	1.1394e-011	8.7313e-008
Tetrahedral Quadratic	4.1196e-012	1.7575e-007
Hex Dominant Linear	7.1826e-012	8.7748e-008
Hex dominant Quadratic	4.0165e-012	1.5865e-007

According to result Tetrahedral Quadratic has maximum Strain 7.8449e-007 as compare to other and Tetrahedral Linear has minimum strain 1.1394e-011.

IX. Conclusion

Hence the FEA of Lever bracket is done and is proved as safe since none of the deformations exceed the given safety values. Comparison by defining different parameters have completed successfully. However, in the bracket module, it was noticed that it is advisable to perform the analysis in hex-dominant. If the manufacturing company uses hex-dominant analysis, there will be no problems like error or broken part in lever bracket.

X. References

1. Material and design, volume 67, 15 February 2015 - study of automobile brake bracket formed by casting–forging integrated forming technology.

2. International Journal of Naval Architecture and Ocean Engineering, Volume 5, Issue 3, September 2013, Pages 376-391.

3. Computer Methods and Programs in Biomedicine Volume 197, December 2020, 105748 Finite element analysis.