INTERNATIONAL JOURNAL OF ADVANCE RESEARCH, IDEAS AND INNOVATIONS IN TECHNOLOGY

ISSN: 2454-132X
Impact Factor: 6.078
(Volume 7, Issue 2 - V7I2-1500)
Available online at: https://www.ijariit.com

Analysis of toggle Lever with the help of FEA in Ansys

Atharva Anil Joshi
atharva.21810046@viit.ac.in
Vishwakarma Institute of Information Technology, Pune, Maharashtra

ABSTRACT

The subject of this project is the analysis of Toggle Lever. Toggle lever is used in pneumatic parts which are used to restrict the air supply as well as is used to locate the cylinder parts. In this paper, we modeled a 3D geometry of Toggle lever using Catia v5R21 and performed structural analysis on Ansys 2021 R1. We have done a comparative analysis with structural steel material of toggle lever. Finite element analysis (FEA) is software from Ansys that gives users the ability to automatize and design or customize simulations. FEA uses mathematical models to understand and quantify the effects of physical phenomena like forces, vibration, deformation, pressure, heat, and other physical properties affecting the simulations, thus reducing the needs of prototypes. Basically, finite elements are method which divides large problem into smaller one with the help of static structural in the finite elemental analysis, we can calculate deformation, stress, strain, and other values. FEA helps us to tell at what maximum and minimum value the deformation is taking place so that we can conclude at the particular value the product is going to break or bend. In this paper, static structural analysis is done on toggle lever by changing the elemental order such as linear and quadratic are used by interchanging the span angle centre which is coarse, and fine. It was observed that there was a difference in values of nodes and elements changed by corresponding linear and quadratic element order with coarse and fine span angle centre. It resulted that the number of nodes was greater in quadratic element order than in linear element order. In the static structural analysis, we observed that the element order having more number of nodes gives maximum deformation, stress, strain compared to the element having less number of nodes. Thus, quadratic element order gives maximum values and accurate result as compared to linear element order.

Keywords: *Toggle lever, Ansys Workbench, Static Structural Analysis, Mesh, Span Angle Centre, Element Order.*

1. INTRODUCTION

Finite Element Analysis is an analysis tool that solves complex linear and non-linear structural analysis problems in civil, mechanical, and aeronautical engineering. In Ansys, we can take a model with any type of geometry and shape and can do FEA of that model. FEA is beneficial to any project that requires strength, durability, or dynamic analysis. The main advantage of FEA is that we can give any material to any model and can change or add properties to the model very easily. For components that are too large and complex for testing, FEA helps to determine its dynamics and product characteristics. At the same time, it has some limitations. The analysis is done on a model but not on a real model, thus all the results are approximated and we cannot predict the difference between obtained results and real ones. A large amount of data is required as input for the mesh therefore it requires a longer time for execution.

2. GEOMETRY

The figure shows 3D geometry of Toggle Lever made with Structural Steel.

A. Details about Material properties (Structural steel):

Table 2: Details of Material Properties

Property Material Name	Density (g/cm3)	modulus of Elasticity (GPa)	Poisson ratio	Tensile Strength (MPa)
Structural Steel	7.85	200	0.3	460

The diagram below shows the fixed support of Toggle Lever:

Fig. 2.2: Fixed support is applied at blue part of Toggle Lever

	Total Deformation (mm)	Equivalent Stress (hbar)	Equivalent Strain (mm/mm)
Maximum	1.8634e-003	0.35386	1.8476e-005
Minimum	0	1.0329e-002	8.7157e-007
Average	4.2768e-004	9.6244e-002	5.2589e-006

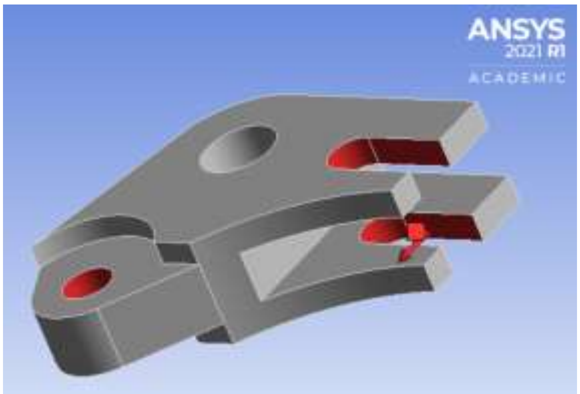

Fig. 2.3: Force of 1000N is applied at red part of Toggle Lever

3. ELEMENT ORDER

In this method, we can control whether we have to use the first-order or linear element which has nodes only at the corner or edges while a second-order or quadratic element has mid-side nodes as well as nodes at the corner. A first-order element requires a linear equation while a second-order requires a quadratic equation to solve it.

The main aim of the Space angle center is used to improve the curvature of the mesh in the model. The mesh will be divided into curved regions until the single element have a span angle. There are three types of angles which will be all over on mesh model: 1. Coarse: 91 – 60 degrees, 2. Medium: 75 – 24 degrees, 3. Fine:36 – 12 degrees.

4. ANALYSIS OF TOGGLE LEVER

In this study, meshing is an important factor in this analysis, by changing the amount of mesh on a model we get an accurate result of the analysis. From the result, it is clear that if we increase the amount of mesh on a model the accuracy of the result will be that perfect resulting in an increase in execution time for analysis.

5. STATIC STRUCTURAL ANALYSIS OF TOGGLE LEVER

With the help of static structural analysis, we have applied physical properties to Toggle Lever. We have applied fixed support to the middle hole and a force of 1000N is applied to the upper hole and lower slot. In the study, we have done a comparison between the total deformation, equivalent stresses(von-mises) and equivalent elastic strain on a Toggle Lever by changing element order from linear to quadratic and also space angle center with a time-lapse of 1 second.

Following are the analysis of Toggle lever:

5.A] Case 1

Element of order: linear	
Span center angle: coarse	
No. of nodes	505
No. of elements	1384

Total Deformation, Equivalent Stress, Equivalent Strain diagrams of Case1 are given below considering above table:

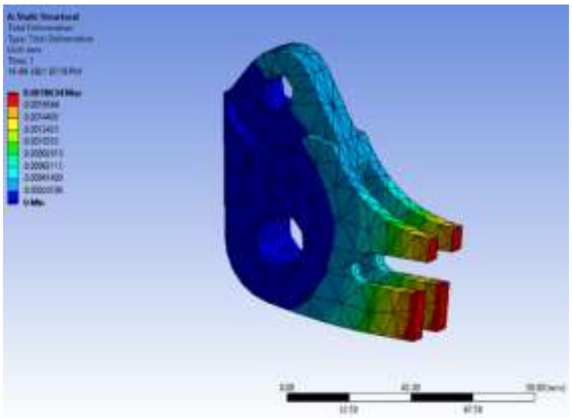

Fig 5.1: Shows Total Deformation

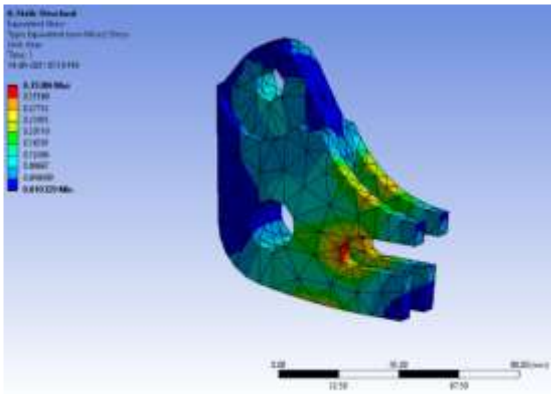

Fig 5.2: Shows Equivalent Stress

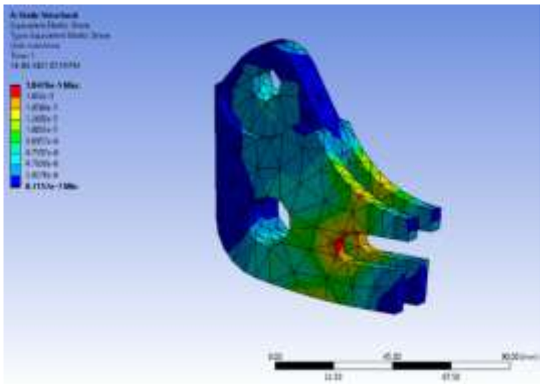

Fig 5.3: Shows Equivalent Elastic Strain

B] Case 2

Element of order: linear	
Span center angle: Coarse	
No. of nodes	610
No. of elements	1689

Total Deformation, Equivalent Stress, Equivalent Strain diagrams of Case2 are given below considering above table:

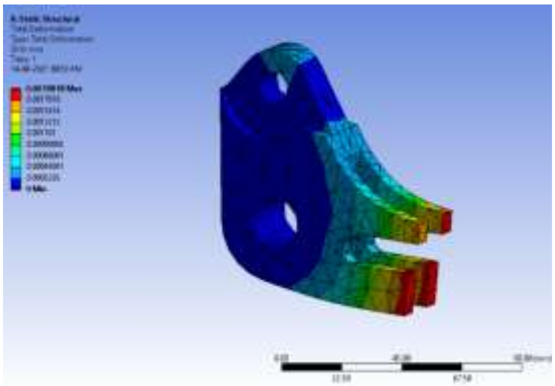

Fig 5.4: Shows Total Deformation

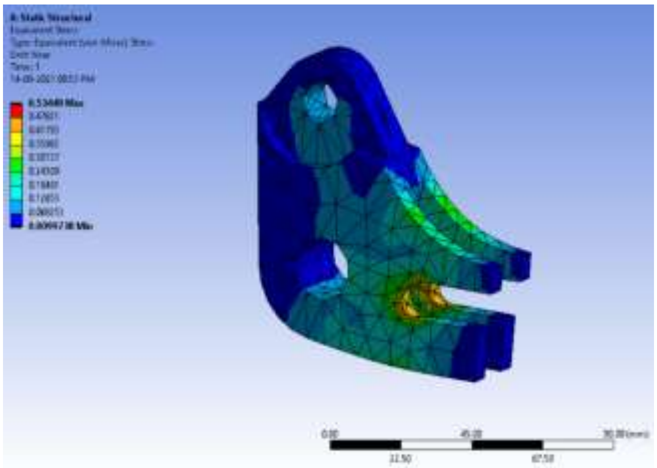

Fig 5.5: Shows Equivalent Stress

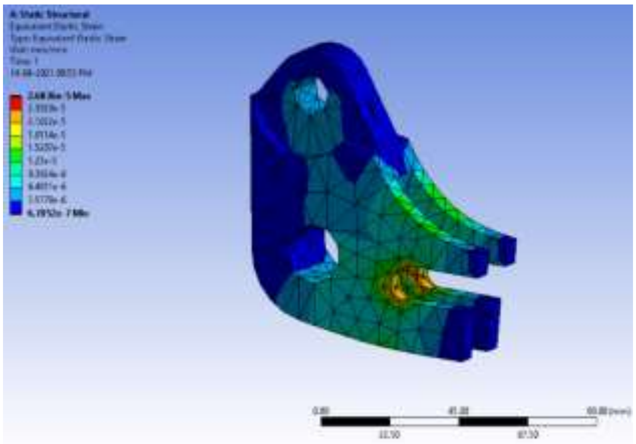

Fig 5.6: Shows Equivalent Elastic Strain

	Total Deformation (mm)	Equivalent Stress (hbar)	Equivalent Strain (mm/mm)
Maximum	2.5445e-003	0.64761	3.271e-005
Minimum	0	4.2971e-003	3.103e-007
Average	5.159e-004	0.13005	7.2301e-006

	Total Deformation (mm)	Equivalent Stress (hbar)	Equivalent Strain (mm/mm)
Maximum	1.9818e-003	0.53449	2.6836e-005
Minimum	0	9.9738e-003	6.7052e-007
Average	4.1671e-004	0.12622	6.7076e-006

6.A] Case 1

Element of order: Quadratic	
Span center angle: Coarse	
No. of nodes	2560
No. of elements	1236

Total Deformation, Equivalent Stress, Equivalent Strain diagrams of Case1 are given below considering above.

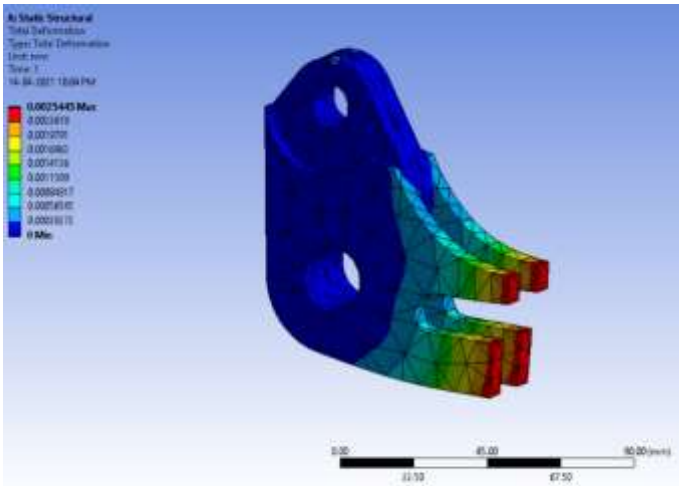

Fig 6.1: Shows Total Deformation

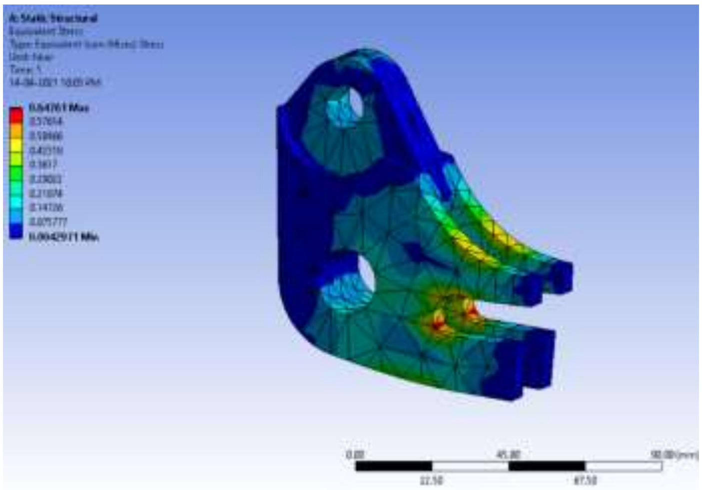

Fig 6.2: Shows Equivalent Stress

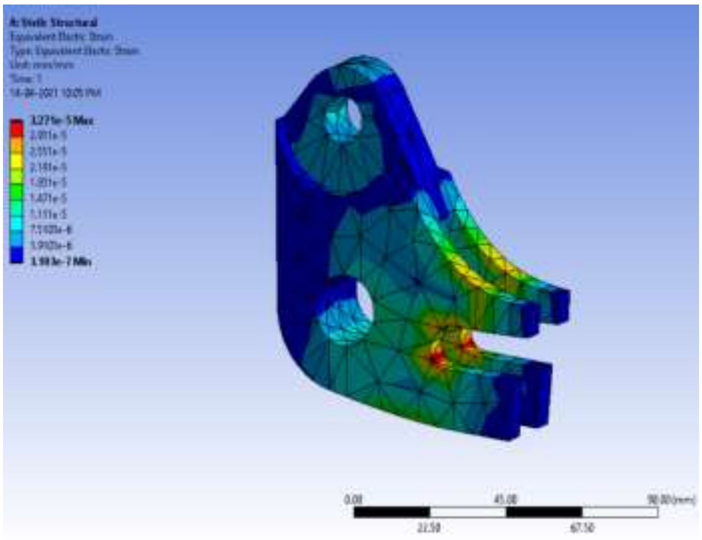

Fig 6.3: Shows Equivalent Elastic Strain

6.B] Case 2

Element of order: Quadratic	
Span center angle: Fine	
No. of nodes	4074
No. of elements	2021

Total Deformation, Equivalent Stress, Equivalent Strain diagrams of Case2 are given below considering above table:

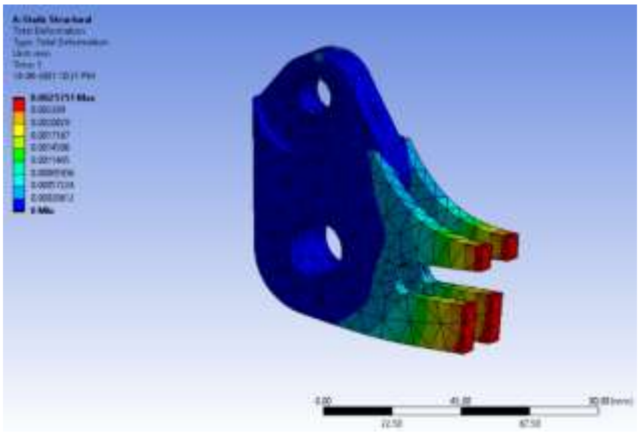

Fig 6.4: Shows Total Deformation

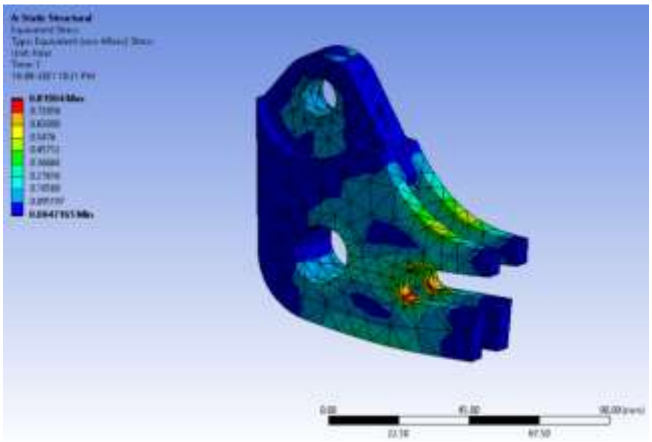

Fig 6.5: Shows Equivalent Stress

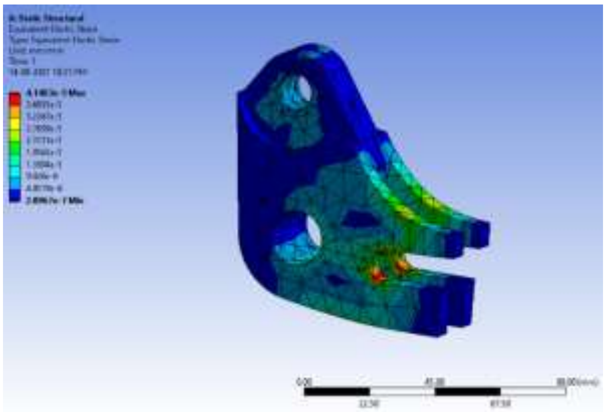

Fig 6.6: Shows Equivalent Elastic Strain

	Total Deformation (mm)	Equivalent Stress (hbar)	Equivalent Strain (mm/mm)
Maximum	2.5751e-003	0.81904	4.1403e-005
Minimum	0	4.7165e-003	2.8967e-007
Average	4.2933e-004	0.1491	8.0071e-006

6. RESULT

The Total Deformation, Equivalent Stress, Equivalent Strain for Quadratic Element was higher compared to Linear Element. From this, it is cleared that, the meshing of quadratic element order gives a more accurate result as there is more number of nodes in quadratic. Hence, the analysis of toggle lever has more accurate results.

7. CONCLUSION

We conclude that the FEA of the toggle lever is safe as there is no braking and deformation of the material and we get what is the maximum deformation, stress, strain that would be applied on the toggle lever. It is clear that in the mesh as the number of nodes of the model increases the accuracy of the result increases. Thus, the combination of quadratic element at coarse and fine element order was found more accurate result as compared to linear element.

8. REFERENCES

[1] Shashikant T. More, Dr. R. S. Bindu, Effect of Mesh Size on Finite Element Analysis of Plate Structure, Volume 4, Issue 3, May 2015.

[2] Modal analysis of cantilever beam Structure Using Finite Element analysis and Experimental Analysis S. P. Chaphalkar1 , Subhash. N. Khetre2 , Arun M. Meshram

[3] Teseo Schneidera, Yixin Hua , Xifeng Gaob , Jérémie Dumasc , Denis Zorina , Daniele Panozzoa, A Large Scale Comparison of Tetrahedral and Hexahedral Elements for Finite Element Analysis.

[4] Finite Element Analysis of Traction gear using ANSYS Praveen Silori, Amir Shaikh, Nithin Kumar KC, Tushar Tandon

[5] Finite element ansys analysis of the mechanical behavior of a bicycle, authors: K-H Chang, K-L Lee

