INTERNATIONAL JOURNAL OF ADVANCE RESEARCH, IDEAS AND INNOVATIONS IN TECHNOLOGY

IJARIIT

ISSN: 2454-132X

Impact Factor: 6.078

(Volume 7, Issue 2 - V7I2-1460)

Available online at: https://www.ijariit.com

Analysis of sprocket with the help of FEA in Ansys

Vivek Patil

vivek.21810112@viit.ac.in

Vishwakarma Institute of Information Technology, Pune, Maharashtra

ABSTRACT

A sprocket, sprocket-wheel or chainwheel is a profiled wheel with teeth, or machine gear-pieces, that network with a chain, track or other punctured or indented material. The name 'sprocket' applies by and large to any wheel whereupon spiral projections draw in a chain disregarding it. The sprocket is a very basic part in the transmission of energy and development in most bicycle; they exist in various estimations, teeth number and are made of different materials. All around sprockets are made of smooth steel.

Keywords— *Sprocket, ANSYS Workbench, Static Structural Analysis, Mesh.*

1. INTRODUCTION

A sprocket is a profiled wheel with teeth, or machine gear-pieces, that work with a chain, track or other penetrated or indented material. The name 'sprocket' applies usually to any wheel whereat extended projections keep company with a sequence ignoring it. Sprockets are used as a piece of bicycles, bicycles, cars, followed vehicles, and other equipment either to send rotational development between two shafts where cog wheels are inadmissible or to give straight development to a track, tape, etc.

Limited Element Analysis is an investigation apparatus that settles complex straight and non-direct primary examination issues in common, mechanical, and aeronautical designing. In Ansys, we can take a model with a calculation and shape and can do FEA of that model. FEA is helpful to any project that requires strength, toughness, or dynamic investigation. The primary benefit of FEA is that we can give any material to any model and can change or add properties to the model without any problem. For parts that are excessively huge and complex for testing, FEA assists with deciding its elements and item attributes.

2. GEOMETRY

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

Fig. 2.1: Geometry of Sprocket 3D model

A. Details about Material properties (Structural steel):

TABLE 2.0: DETAILS OF MATERIAL PROPERTIES

Property → Material Name ↓	Density (kg/m^-3)	Tensile Yield Strength (GPa)	Compressive Yield Strength (GPa)	Tensile Ultimate Strength (GPa)
Structural Steel	7850	0.25	0.25	0.46

The diagram below shows the fixed support of Sprocket

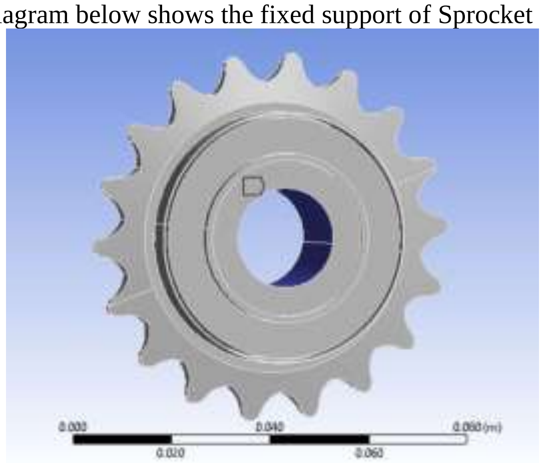


Fig. 2.2: Fixed support is applied at labelled part of Sprocket

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

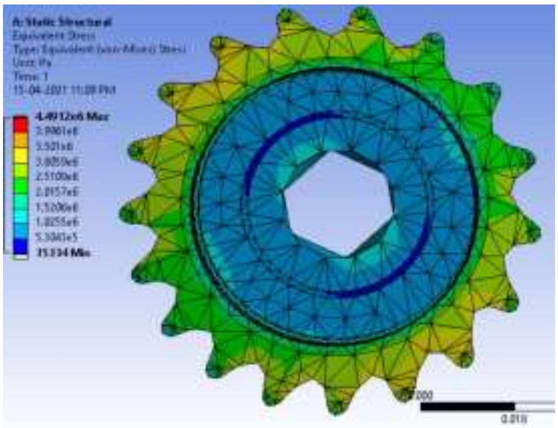


Fig: Equivalent Stress

3. ELEMENT ORDER

In this technique, we are able to decide whether we've to use first-order or linear part that has nodes solely at the corner edges whereas a second-order or quadratic part has mid-side nodes additionally as nodes at the corner.

The main aim of the area angle center is employed to boost the curvature of the mesh within the model. The mesh is going to be divided into sickle-shaped regions till the only part have a span angle.

4. STATIC STRUCTURAL ANALYSIS OF SPROCKET

With the help of static structural analysis, we have applied physical properties to Sprocket. We have applied fixed support of 1000N is applied to the outer part of the sprocket. In the study, we have done a comparison between the total deformation, equivalent stresses(von-mises) and equivalent elastic strain on a Sprocket 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 Sprocket:
4.A] Case 1

Element of order: Linear	
Span center angle: Coarse	
No. of nodes	2386
No. of elements	7326

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

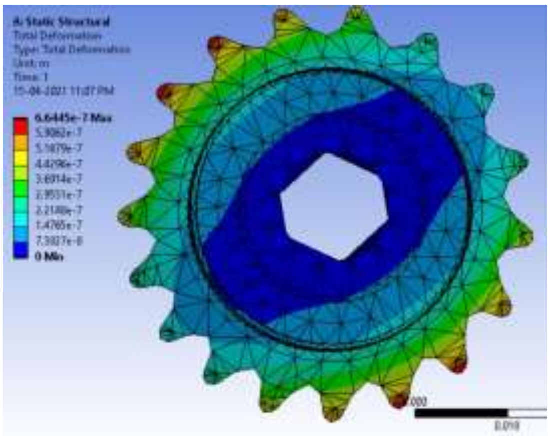


Fig: Total Deformation

	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

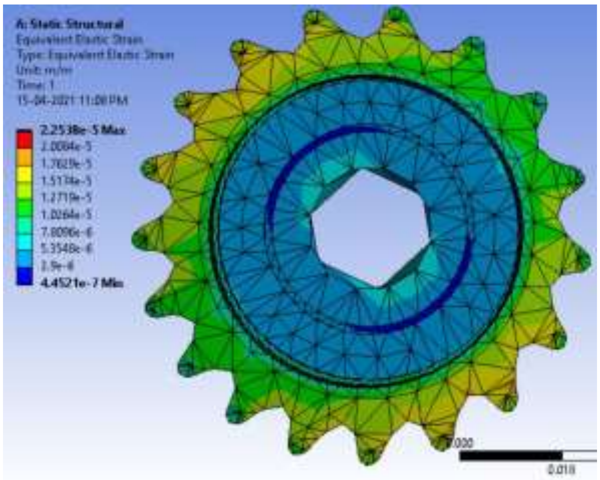


Fig: Equivalent Elastic Strain

4.B] Case 2

Element of order: Linear	
Span center angle: Fine	
No. of nodes	2512
No. of elements	7688

Total Deformation, Equivalent Stress, Equivalent Strain diagrams of Case 2 are given below considering above table:

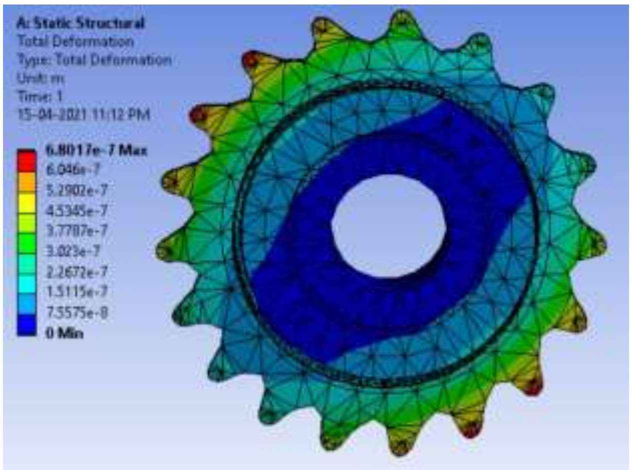


Fig: Total Deformation

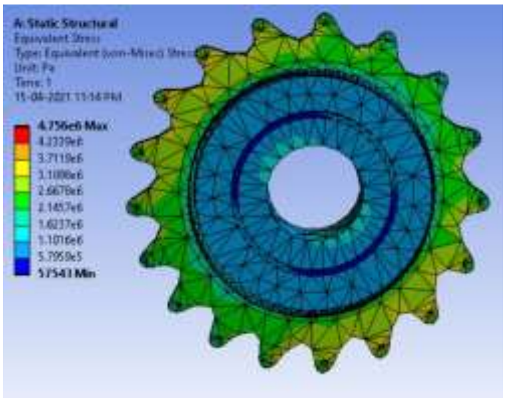
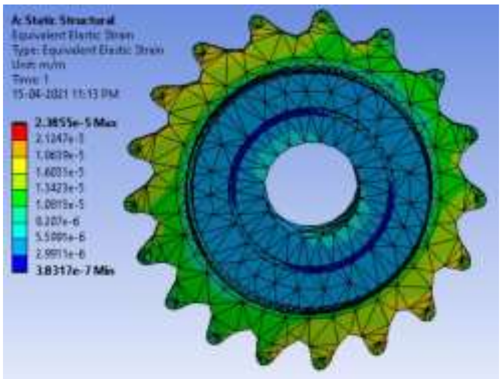


Fig: Equivalent Stress



	Total Deformation (m)	Equivalent Stress (Pa)	Equivalent Strain (m/m)
Maximum	7.9693e-007	6.4575e+006	3.2597e-005
Minimum	0	20402	3.0815e-007
Average	2.2479e-007	1.7869e+006	9.6368e-006

Fig: Equivalent Elastic Strain

	Total Deformation (m)	Equivalent Stress (Pa)	Equivalent Strain (m/m)
Maximum	6.8017e-007	4.756e+006	2.3855e-005
Minimum	0	57543	3.8317e-007
Average	2.0504e-007	1.5603e+006	8.0313e-006

5.A] Case 1

Element of order: Quadratic	
Span center angle: Coarse	
No. of nodes	11442
No. of elements	5788

Total Deformation, Equivalent Stress, Equivalent Strain diagrams of Case 1 are given below considering above.

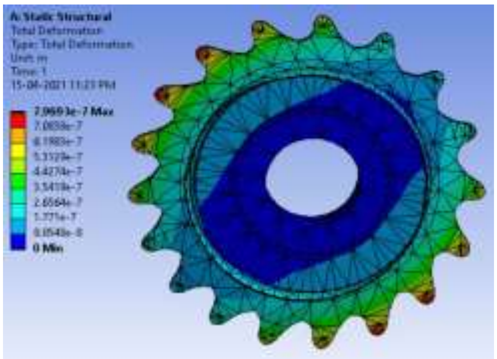


Fig: Total Deformation

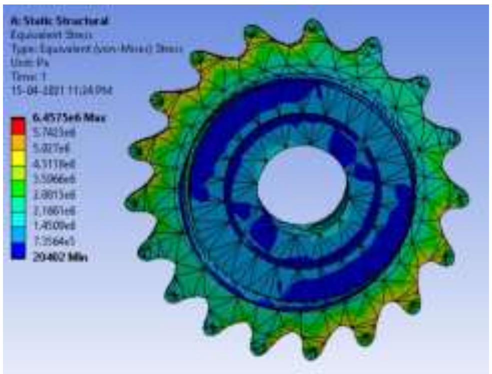


Fig: Equivalent Stress

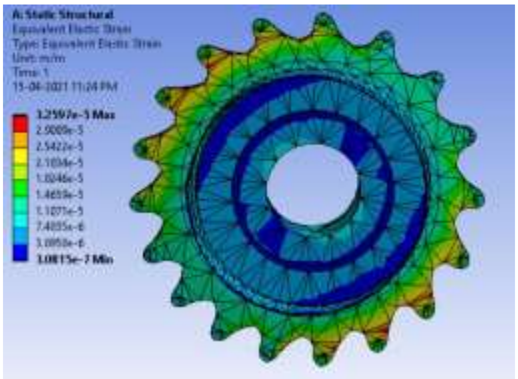


Fig: Equivalent Elastic Strain

5.B] Case 2

Element of order: Quadratic	
Span center angle: Fine	
No. of nodes	15356
No. of elements	7666

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

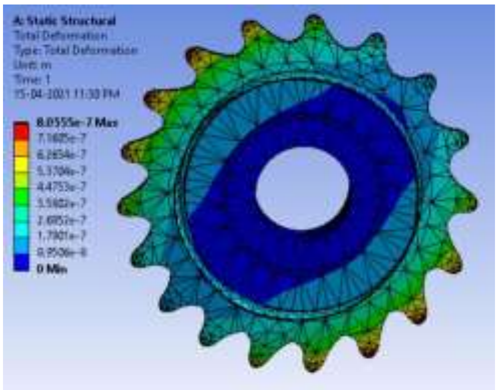


Fig: Total Deformation

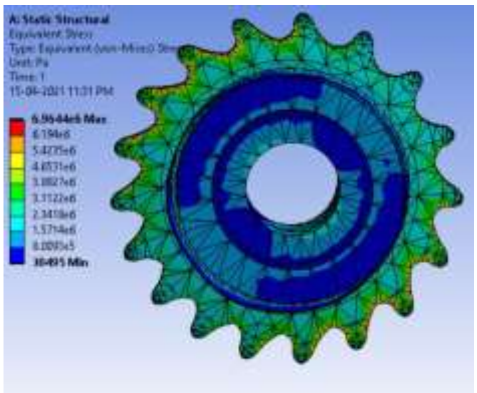


Fig: Equivalent Stress

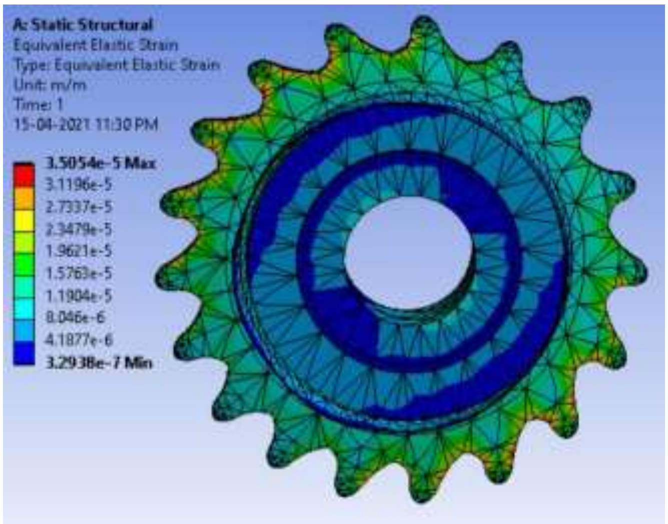


Fig: Equivalent Elastic Strain

	Total Deformation (m)	Equivalent Stress (Pa)	Equivalent Strain (m/m)
Maximum	8.0555e-007	6.9644e+006	3.5054e-005
Minimum	0	30495	3.2938e-007
Average	2.787e-007	1.993e+006	1.048e-005

5. RESULT

As compared to Linear Element the Total Deformation, Equivalent Stress, Equivalent Strain for Quadratic Element is higher. From this we can conclude that, the meshing of quadratic element order gives a more accurate result as there are more number of nodes in quadratic.

6. CONCLUSION

We get values of the maximum deformation, stress, strain that can be applied on the sprocket. We conclude that the FEA of the sprocket is safe as there is no braking and deformation of the material and, 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.

7. 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] Design and Analysis of Chain Sprocket using Reverse Engineering K. Chinna Maddaiah, Y Diwakar Reddy, Assistant Professor, Department of Mechanical Engineering, MLR Institute of Technology, Hyderabad.

