International Journal of Research Publication and Reviews

Journal homepage: www.ijrpr.com ISSN 2582-7421

Effect of Element Order and Span Angle of Static Structural and Modal Analysis on Shock Absorber of Vehicle Using Ansys

Aniket Salvi [a], ***Revati Mathpati*** [b]

[a]*Vishwakarma Institutes of Information Technology,Pune,Evershine Gharkul Vidyanagar Karad Satara Maharashtra,415110,India*

[b]*Vishwakarma Institutes of Information Technology,Pune,Hostel Chowk Latur Maharashtra,413510,India*

A B S T R A C T

Wellbeing and driving comfort for a vehicle's driver are both dependent on the vehicle's suspension framework. Wellbeing suggests to the vehicle's taking care of and slowing down capacities. The comfort of the inhabitants of a vehicle corresponds to sleepiness and capacity to travel significant distance with insignificant disturbance. Basically, shock absorbers are gadgets that smooth out a drive experienced by a vehicle, and suitably disperse or ingest the motor energy. Basically, in this paper we have shown the effect of element order and span angle of static structural analysis on the shock absorber and the modal analysis to check the criteria of failure. We have taken a shock absorber of a vehicle. For the 3d modeling and assembly of shock absorber we have used solid works 20. The analysis part is done by using ANSYS 2021. The static structural analysis is done on the shock absorber to check the effect of element order and shape angle. Modal analysis is done to check the harmonic frequency of the absorber, how the components amplitude will vary according to the different frequencies. From the analysis we can conclude that while meshing by changing the span angle center number of nodes and number of elements increases from coarse then medium and highest in fine in linear as well as quadratic element order also number of nodes and number of elements are more in quadratic order than in linear. from the harmonic frequency we can conclude that if the natural frequency of the absorber matches with the forced calculated frequency i.e.,274.7 Hz at 1200N.then resonance will take place and component will fail.

Keywords: Ansys, Solid works, Mesh, Springs, dampers, shock absorber

Introduction

Shock absorbers are a basic piece of a suspension framework, associating the vehicle to its wheels. The requirement for dampers emerges on account of the roll and pitches related with vehicle and from the unpleasantness of streets. shock absorbers are gadgets that smooth out a drive experienced by a vehicle, and fittingly scatter or ingest the motor energy. At the point when a vehicle is going on a level street and the wheels strike a knock, the spring is packed rapidly. The packed spring will endeavor to get back to its typical stacked length and, in this manner, will bounce back past its ordinary stature, making the body be lifted. The heaviness of the vehicle will at that point push the spring down underneath its ordinary stacked stature. This, thusly, makes the spring bounce back once more. This skipping interaction is rehashed again and again, somewhat less each time, until the here and there development at long last stops. On the off chance that bobbing is permitted to go uncontrolled, it won't just motivation an awkward ride however will make treatment of the vehicle troublesome.

A protected vehicle should have the option to stop by a wide scope of street conditions. Great contact between the tires and the street will ready to stop and rapidly Suspension is the term given to the arrangement of springs, shock absorbers and linkages that associates a vehicle to its wheels. shock absorber is a significant piece of car suspension framework which affects ride qualities. shock absorbers are additionally basic for tire to street contact which to diminish the propensity of a tire to take off the street. This influence slowing down, guiding, cornering and in general soundness. The expulsion of the shock absorber from suspension can cause the vehicle ricochet all over. It is workable for the vehicle to be driven, however in the event that the suspension drops from the driving over an extreme knock, the back spring can drop out the plan of spring in suspension framework is vital.

The purpose of study is to compare the Element Order in FiniteElement Meshes for Static Structural Analysis for shock absorber System as shown in fig1. The material used for the shock absorber is spring steel which has Compressive Ultimate Strength 0, Compressive Yield Strength 2.5e+008, Tensile Yield Strength 2.5e+008, Tensile Ultimate Strength 4.6e+008, Young's Modulus 2e+011, Poisson's Ratio 0.3, Bulk Modulus 1.6667e+011 and Shear Modulus 7.6923e+010, density 7850 kg/m^3.The base part has volume of 2.1495e-005 m^3 and mass is 0.16874 kg. The spring has a volume of 2.321e-005 m^3mand mass is 0.1822 kg. The nut has a volume of 8.0044e-006 $m^3$and mass of 6.2835e-002 kg. And for the top we have a volume of 4.7662e-005 m^3 and mass of 0.37415 kg.

1.1 Methodology

The 3d parts for the shock absorber i.e top part, nut, spring, base are completed in solid works. Assembly is also done in solid works.

After that complete assembly is saved in. igs format, we have taken a static structural analysis and assembly is imported in the geometry section of Ansys. spring steel is the material added in the engineering data section. Meshing is done using mesh command and we have taken default element size. Different element orders are considered like coarse, fine and medium to check the results or variations in total deformation and equivalent stress.
After that we have used modal for modal analysis from ANSYS, and same geometry is imported without applying the force. we have just provided the fixed support at base section of shock absorber. Here we checked the frequency variation vs total deformation. We got some frequencies and after that to check the harmonic frequency we have used harmonic response section from ANSYS and we applied 1200 N load at the opposite side of fixed support, from the different frequencies we got a plot of frequency vs amplitude and frequency vs phase angle.

1.2 Cad modelling

CAD/ CAE Software's used:

- Solid works 20 - For 3D Component Design.
- Solid works 20 - For Assembling Components
- ANSYS Workbench 21- For FEM analysis

1.3 Partts of the assembly

1) Top part:

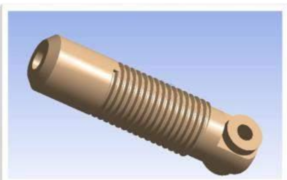


Fig-1, Top part

2) Nut:

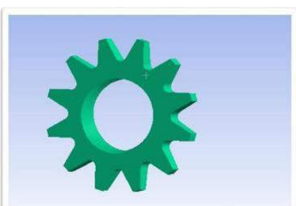


Fig-2, Nut

2) Spring:

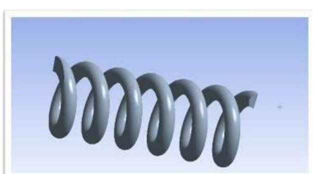


Fig-3, Spring

3) Base:

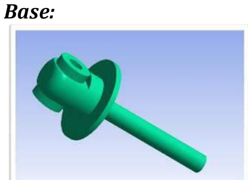


Fig-4, Base

1.4 Shock absorber Assembly:

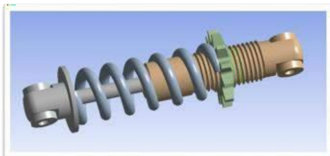


Fig-5,Assembly

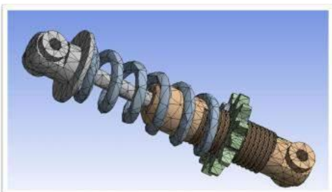


Fig-6 , Meshedpart

1.5 Finite Element Analysis:

The finite element method (FEM), at times called to as finite element analysis (FEA), is a computational procedure used to get inexact arrangements of boundary value problems in designing. Basically expressed, a boundary value is a numerical problem where at least one ward factors should fulfil a differential condition wherever inside a known area of autonomous factors and fulfil explicit conditions on the limit of the space. Boundary value problems are additionally at times called field problems. The field is the space of interest and frequently addresses an actual design. The field factors are the reliant factors of interest administered by the differential condition. The limit conditions are the predefined estimations of the field factors (or related factors like subordinates) on the limits of the field. Contingent upon the sort of actual problem being broke down, the field factors may incorporate actual removal, temperature, heat transition, and liquid speed to give some examples.
An Overall Method for Finite Element Analysis Certain means in defining a finite element analysis of an actual problem are regular to all such examinations, regardless of whether primary, heat move, liquid stream, or some other problem. These means are encapsulated in business limited component programming bundles (some are referenced in the accompanying passages) and are certainly consolidated in this content, despite the fact that we don't really suggest to the means expressly. The steps for the same are given below

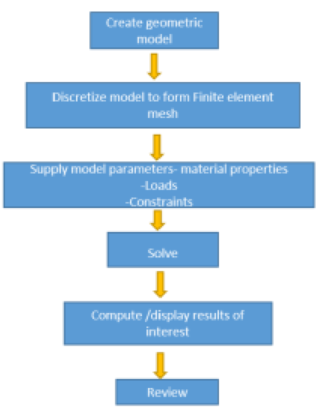


Fig-7, overall steps in FEA

1.6 Span angle centre:

It sets the goal for curvature based local refinement. On the boundaries the mesh will subdivide in curved regions until the individual element span this angle.

Coarse: 91 - 60

Medium: 75 - 24

Fine: 36 – 12

1.7 Static Structural Analysis of the shock absorber

Static Structural Analysis is used to determine the effect of loads on the physical bodies. Applied mechanics, Mathematics, Material Science is used to compute the deformation, stresses, forces, reactions and stability of the object. It plays a vital role in engineering design as the results of the analysis of object are used to verify its fitness and effectiveness for use.The base of the shock absorber was treated as a fixed support and a force of 1200 N was applied. Graph of force is shown in graph1. In the study we have compared the total deformation and equivalent stresses on a shock absorber by changing element order from linear to quadratic with a time lapse of 1 second.

Graph-1, Force representation

Modal analysis:

Modal analysis is broadly used to depict the powerful properties of a structural as far as the modular boundaries, common recurrence, damping factor,modular mass and mode shape. The examination might be done either tentatively or numerically. In numerical modular investigation, one endeavours to ncouple the primary conditions of movement so that each uncoupled condition can be addressed independently. At the point when careful arrangements are nrealistic, mathematical approximations, for example, limited component and limit component strategies are utilized.

Graph-2, mode and frequency graph

2 Results:

2.1 Static Structural Results

Element Order- linear

Span Angle Centre- Coarse (Equivalent stress is shown in fig8. and total deformation is shown in fig9

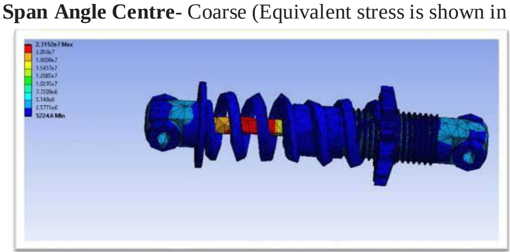


Fig8. Equivalent stress

After meshing it was observed that number of nodes present are 3711 and number of elements are 11699. Also, after structural analysis the results obtained are in table 1.

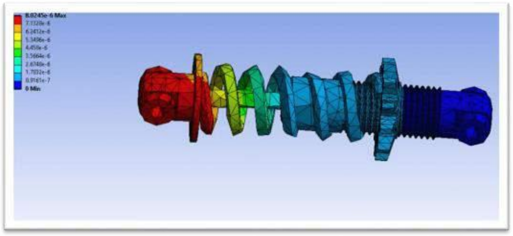


Fig9. Total Deformation

Table -1

	Equivalent Stress	*Total Deformation*
Minimum	5224.6 Pa	0. m
Maximum	2.3152e+007 Pa	8.0245e-006 m
Average	1.9246e+006 Pa	1.7423e-006m

Span Angle Centre- Medium (Equivalent stress is shown in fig10. and total deformation is shown in fig11.)

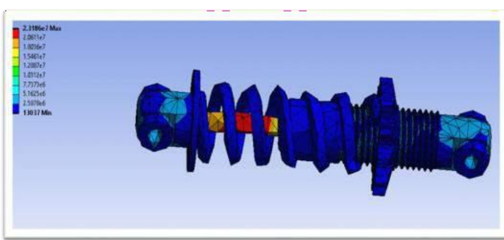


Fig10. Equivalent stress

After meshing it was observed that number of nodes present are 3758 and number of elements are 11786. Also, after structural analysis the results obtained are in Table 2

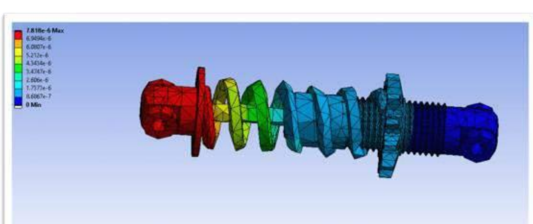


Fig-11. Total Deformation

Table-2

	Equivalent Stress	*Total Deformation*
Minimum	13037 Pa	0. m
Maximum	2.3186e+007 Pa	7.818e-006 m
Average	1.9129e+006 Pa	1.7529e-006 m

Span Angle Centre- Fine (Equivalent stress is shown in fig12. and total deformation is shown in fig13.)

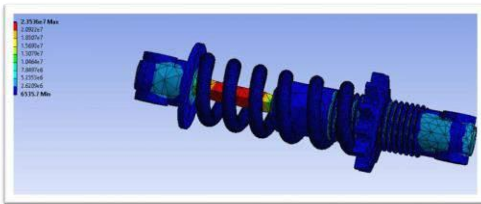


Fig12.Equivalentstress

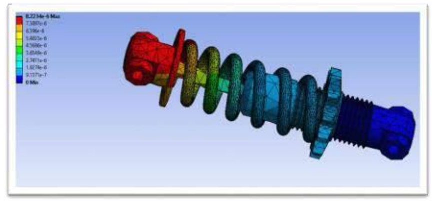


Fig13. Total Deformation

After meshing it was observed that number of nodes present are 11481and number of elements are 38808. Also after structural analysis the results obtained are in table 3

Table -3

	Equivalent Stress	*Total Deformation*
Minimum	6535.7 Pa	0. m
Maximum	2.3536e+007 Pa	8.2234e-006 m
Average	9.8942e+005 Pa	2.5155e-006 m

Element Order- quadratic

Span Angle Centre- Coarse (Equivalent stress is shown in fig14. and total deformation is shown in fig15.)

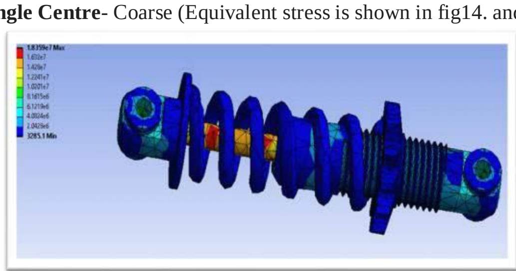


Fig-14, Equivalent stress

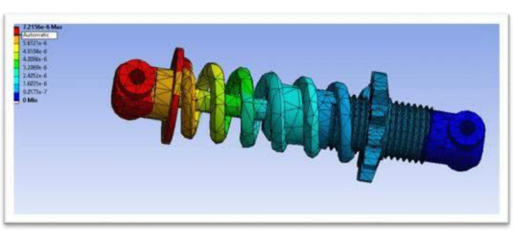


Fig15. Total Deformation

After meshing it was observed that number of nodes present are 22335 and number of elements are 11659.Also, after structural analysis the results obtained are in table 4

Table -4

	Equivalent Stress	*Total Deformation*
Minimum	3285.1 Pa	0. m
Maximum	1.8359e+007 Pa	7.2156e-006 m
Average	2.1747e+006 Pa	1.7404e-006 m

Span Angle Centre- medium (Equivalent stress is shown in fig16. and total deformation is shown in fig17.)

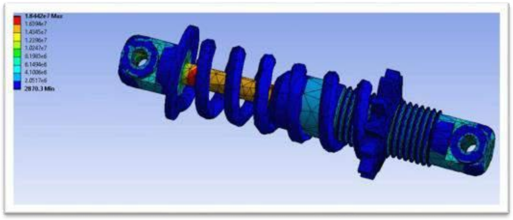


Fig16. Equivalent stress

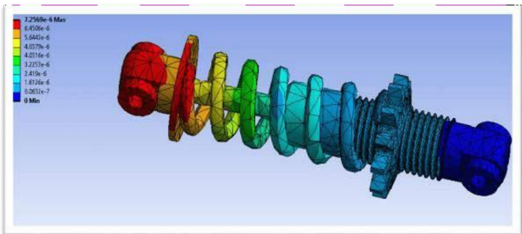


Fig17. Total Deformation

After meshing it was observed that number of nodes present are 25587 and number of elements are 13396.Also, after structural analysis the results obtained

Table -5

	Equivalent Stress	*Total Deformation*
Minimum	3285.1 Pa	0. m
Maximum	1.8359e+007 Pa	7.2156e-006 m
Average	2.1747e+006 Pa	1.7404e-006 m

Span Angle Centre- fine (Equivalent stress is shown in fig18. and total deformation is shown in fig19.)

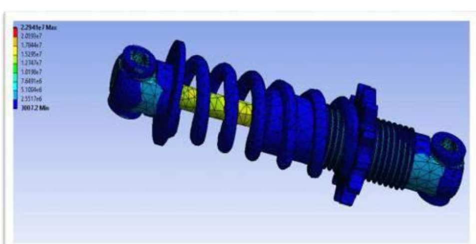


Fig18. Equivalent stress

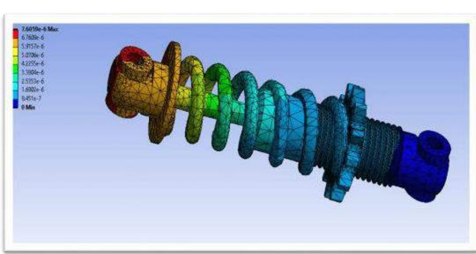


Fig19. Total Deformation

After meshing it was observed that number of nodes present are 65671 and number of elements are 36260.Also, after structural analysis the results obtained in table 6.

Table -6

	Equivalent Stress	*Total Deformation*
Minimum	3007.2 Pa	0. m
Maximum	2.2941e+007 Pa	7.6059e-006 m
Average	2.1976e+006 Pa	2.0479e-006 m

2.2 Modal analysis results:

Mode shape-(mode =1, frequency = 275.1)

Table -7

Minimum deformation	**0. m**
Maximum deformation	**2.7824 m**
Average	**0.66672 m**

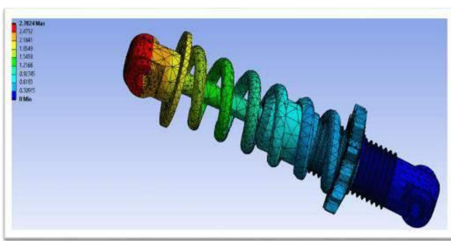


Fig20. Total Deformation

Mode shape-(mode =2, frequency = 295.32)

Table -8

Minimum deformation	**0. m**
Maximum deformation	**2.8882 m**
Average	**0.64544 m**

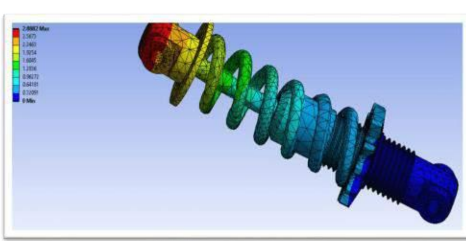


Fig21. Total Deformation

Mode shape-(mode =3, frequency = 764.81)

Table -9

Minimum deformation	**0. m**
Maximum deformation	**2.6087 m**
Average	**0.82583 m**

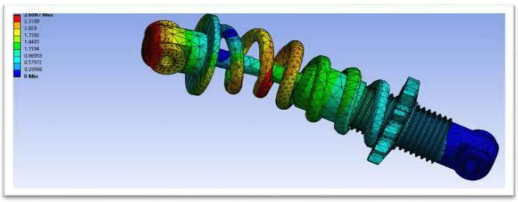


Fig 22. Total Deformation

Mode shape-(mode =4, frequency = 786.43)

Table -10

Minimum deformation	**0. m**
Maximum deformation	**3.1494 m**
Average	**0.78735 m**

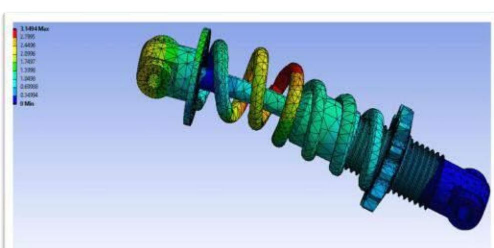


Fig23. Total Deformation

Mode shape-(mode =5, frequency = 888.86)

Table -11

Minimum deformation	**0. m**
Maximum deformation	**4.6304 m**
Average	**0.46526 m**

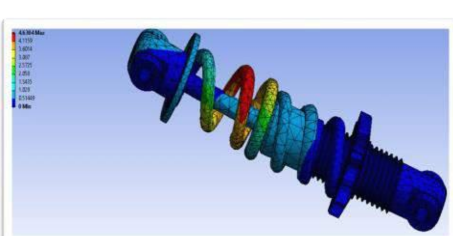


Fig24. Total Deformation

Mode shape-(mode =6, frequency =927.43)

Table -12

Minimum deformation	**0. m**
Maximum deformation	**5.4541 m**
Average	**0.33079 m**

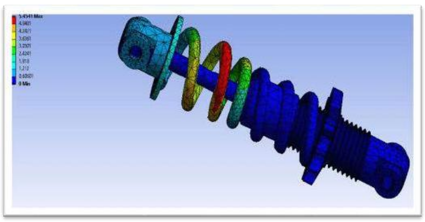
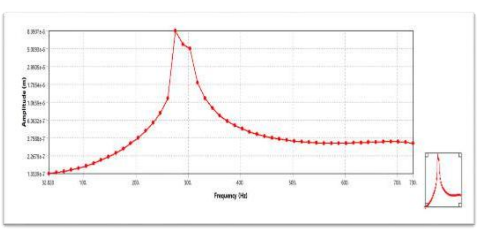


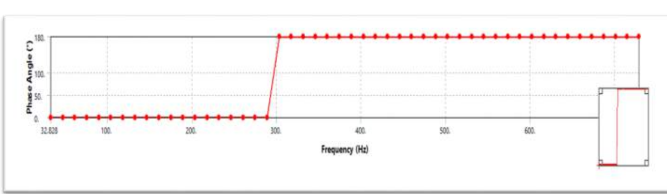
Fig25. Total Deformation

2.1 *frequency response (Harmonic) results:*

Harmonic frequency response refers to a sinusoidal external force of a certain frequency applied to a system. The response of a system to harmonic excitation is a very important topic because it is encountered very commonly and also covers the concept of resonance. Resonance occurs when the external excitation has the same frequency as the natural frequency of the system. It leads to large displacements and can cause a system to exceed its elastic range and fail structurally. A popular example, that many people are familiar with, is that of a singer breaking a glass by singing. Harmonic excitation is also commonly observed in systems that contain a rotating mass for example tires, engines, rotors, etc



Graph -3, Amplitude vs frequency



Graph – 4 phase angle vs frequency

From the above data in graph-3, we got that the frequency for which the amplitude reach at the max value is 274.7 and the corresponding amplitude is 8.39e-06 as mentioned in the table-13. The graph for amplitude vs frequency is plotted for frequency ranges from 18.7 to 730(Hz). Graph-4 shows the graph between phase angle and frequency. From the table 16 we can see that the phase changes take place at a point where the maximum amplitude has reached. (0 to 180)

Table - 13

frequency	amplitude	phase angle
32.828	1.35E-07	0
47.056	1.39E-07	0
61.284	1.44E-07	0
75.512	1.51E-07	0
89.74	1.59E-07	0
103.97	1.70E-07	0
118.2	1.83E-07	0
132.42	2.00E-07	0
146.65	2.20E-07	0
160.88	2.46E-07	0
175.11	2.79E-07	0
189.34	3.22E-07	0
203.56	3.81E-07	0
217.79	4.62E-07	0
232.02	5.84E-07	0
246.25	7.84E-07	0
260.48	1.18E-06	0
274.7	8.39E-06	0
288.93	5.67E-06	0
303.16	5.03E-06	180
317.39	1.88E-06	180
331.62	1.19E-06	180
345.84	8.92E-07	180
360.07	7.24E-07	180
374.3	6.18E-07	180
388.53	5.45E-07	180
402.76	4.92E-07	180
416.98	4.53E-07	180
431.21	4.23E-07	180
445.44	3.99E-07	180
459.67	3.81E-07	180
473.9	3.66E-07	180
488.12	3.54E-07	180
502.35	3.45E-07	180
516.58	3.38E-07	180
530.81	3.33E-07	180
545.04	3.29E-07	180
559.26	3.27E-07	180
573.49	3.25E-07	180
587.72	3.25E-07	180
601.95	3.26E-07	180
616.18	3.28E-07	180
630.4	3.30E-07	180
644.63	3.33E-07	180
658.86	3.37E-07	180
673.09	3.40E-07	180
687.32	3.42E-07	180
701.54	3.42E-07	180
715.77	3.38E-07	180
730	3.23E-07	180

3. Conclusion:

In this project we have designed a shock absorber. We have modelled the shock absorber by using solid works 20. To validate the strength of our design, we have done structural analysis and modal analysis on the shock absorber using spring steel as a material . In this paper we have compared the static structural analysis of a shock absorber by taking the order of the element linear and quadratic. The primary motive of higher order element is it gives the results more accurately. From the analysis we can conclude that while meshing by changing the span angle center number of nodes and number of elements increases from coarse then medium and highest in fine in linear as well as quadratic element order also number of nodes and number of elements are more in quadratic order than in linear.

When the force of 1200 N was applied to shock absorber by changing span angle center, the maximum equivalent stress was least in coarse then in medium and highest in fine. Also the maximum total deformation was highest in fine then in medium and least in coarse in both of the element order. Therefore, from the above results we can conclude that the accuracy increases as we go from lower order of element to higher order of element.

from the harmonic frequency we can conclude that if the natural frequency of the absorber matches with the forced calculated frequency i.e., 274.7 Hz at 1200N.then resonance will take place and component will fail

ACKNOWLEDGEMENTS

I would like to thank my college Vishwakarma institutes of information technology Pune and Associate professor Dr. Atul Kulkarni for providing good knowledge of how to make the research paper.

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] Charles.G.Martin and Dr. A. Arokkiaswamy, Finite Element Modal Analysis and Mesh Optimization of a Typical Turbo Fan Engine – Fan Hub Frame, t, Vol. 1 Issue 3, March 2016.

[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] Steven E. Benzley, Ernest Perry, Karl Merkley, Brett Clark, A Comparison of All Hexagonal and All Tetrahedral Finite Element Meshes for Elastic and Elasto-plastic Analysis.

[5] Bamin Khomami.e' Kapil K. Talwar, and Herambh K. Ganpule, A comparative study of higher- and lower-order finite element techniques for computation of viscoelastic flows.

[6] P.-Y. Rohan, C. Lobos, M. A. Nazari, Pascal Perrier, Y. Payan, Finite element models of the human tongue: a mixed-element mesh approach.

[7] I A Magomedov1 and Z S Sebaeva2, Comparative study of finite element analysis software packages