Structural and Modal Analysis of Shock Absorber of Vehicle

Rahul Tekade[1], Chinmay Patil[2]

[1]M.E Advanced Manufacturing and Mechanical System Design, Department of Mechanical Engineering, Shri Sant Gajanan Maharaj College Of Engineering, Shegaon,Maharashtra,India

[2]Assistant Professor, Department of Mechanical Engineering, Shri Sant Gajanan Maharaj College Of Engineering, Shegaon,Maharashtra,India

Abstract

Safety and driving comfort for a car's driver are both dependent on the vehicle's suspension system. Safety refers to the vehicle's handling and braking capabilities. The comfort of the occupants of a car correlates to tiredness and ability to travel long distance with minimal annoyance. Shock absorbers are a critical part of a suspension system, connecting the vehicle to its wheels.

Essentially shock absorbers are devices that smooth out an impulse experienced by a vehicle, and appropriately dissipate or absorb the kinetic energy. Almost all suspension systems consist of springs and dampers, which tend to limit the performance of a system due to their physical constraints. Suspension systems, comprising of springs and dampers, are usually designed for passengers' safety, and do little to improve passenger comfort. To meet the current demands of high speed and safety we must designed and developed such a shock absorber which can sustain more and more vibrations and also improves the safety.

Keywords: Springs, dampers, **shock absorber**

1. INTRODUCTION

Shock absorbers are a critical part of a suspension system, connecting the vehicle to its wheels. The need for dampers arises because of the roll and pitches associated with vehicle manoeuvring, and from the roughness of roads. Shock absorbers are devices that smooth out an impulse experienced by a vehicle, and appropriately dissipate or absorb the kinetic energy. When a vehicle is travelling on a level road and the wheels strike a bump, the spring is compressed quickly. The compressed spring will attempt to return to its normal loaded length and, in so doing, will rebound past its normal height, causing the body to be lifted. The weight of the vehicle will then push the spring down below its normal loaded height. This, in turn, causes the spring to rebound again. This bouncing process is repeated over and over, a little less each time, until the up-and-down movement finally stops. If bouncing is allowed to go uncontrolled, it will not only cause an uncomfortable ride but will make handling of the vehicle very difficult.

A safe vehicle must be able to stop and manoeuvre over a wide range of road conditions. Good contact between the tires and the road will able to stop and manoeuvre quickly Suspension is the term given to the system of springs, shock absorbers and linkages that connects a vehicle to its wheels. Shock absorber is an important part of automotive suspension system which has an effect on ride characteristics. Shock absorbers are also critical for tire to road contact which to reduce the tendency of a tire to lift off the road. This affects braking, steering, cornering and overall stability. The removal of the shock absorber from suspension can cause the vehicle bounce up and down. It is possible for the vehicle to be driven, but if the suspension drops from the driving over a severe bump, the rear spring can fall out The design of spring in suspension system is very important.

In this project a shock absorber will be designed and a 3D model which will be created is by using Pro/Engineer. The model is also changed by changing the thickness of the spring. Structural analysis and modal analysis are done on the shock absorber by varying material for spring.. The analysis will consider the loads, bike weight, single person and 2 persons. Structural analysis will be done to validate the strength and modal analysis to determine the displacements for different frequencies for number of modes. Comparison will be done for two materials to verify best material for spring in Shock absorber.

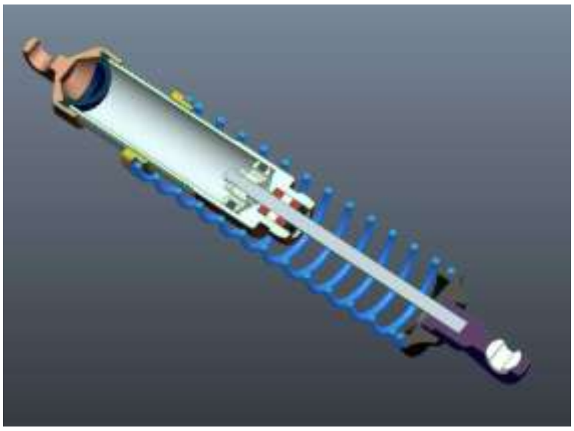


Figure 1.1 Shock absorbers with different dimensions

2 DESIGN OF HELICAL COMPRESSION SPRING FOR SHOCK ABSORBER

Material: ASTM A228 (modulus of rigidity) G = 184.48 $*10^3$ N/mm^2
Mean diameter of a coil D=60 mm
Diameter of wire d = 10 mm
Total no of coils n'= 7
Outer diameter of spring coil D0 = D +d =70mm
Inner diameter of spring coil Di = D +d =70 mm
No of active turns n= 5
Weight of bike = 150 Kgs
Let weight of 1 person = 75 Kgs
Total Weight (Wt) = Weight of bike + Weight of 1 persons
= 150+75 = 225Kgs

Rear suspension = 65%
65% of 225 = 146 Kgs
Considering dynamic loads it will be double
Wt = 292 Kgs = 2864 N
For single shock absorber weight (W) =Wt/2=2864 /2 =1432 N
We Know that, compression of spring (δ) = × × /×
C = spring index = = = 6 = 8 (δ) = × × × /× = 6.7
Solid length, Ls=n'×d=7*10=70
Free length of spring,
Lf = solid length+maximum compression + clearance between adjustable coils = 70+6.7+6.7*0.15 = 77.7
Spring rate, K = W/(δ) =1432/6.7 =213.73 Pitch of coil, P = 13
Stresses in helical springs: maximum shear stress induced in the wire τ = K× K = + . = × × + . = 0.97 τ = K× = 0.97 × × × × = 274 N/mm^2

3 CAD MODELLING

CAD/ CAE Software's used:

- PRO/E wildfire 4.0 - For 3D Component Design.
- Pro/Assembly - For Assembling Components
- ANSYS Workbench 11.0 - For FEM analysis

3.1 Introduction to PRO-E

Pro/ENGINEER is a parametric, feature based, solid modeling System. It is the only menu driven higher end software. Pro/ENGINEER provides mechanical engineers with an approach to mechanical design automation based on solid modeling technology and the following features.

3-D Modeling

The essential difference between Pro/ENGINEER and traditional CAD systems is that models created in Pro/ENGINEER exist as three-dimensional solids. Other 3-D modelers represent only the surface boundaries of the model. Pro/ENGINEER models the complete solid. This not only facilitates the creation of realistic geometry, but also allows for accurate model calculations, such as those for mass properties.

Parametric Design

Dimensions such as angle, distance, and diameter control Pro/ENGINEER model geometry. You can create relationships that allow parameters to be automatically calculated based on the value of other parameters. When you modify the dimensions, the entire model geometry can update according to the relations you created.

Feature-Based Modeling

You create models in Pro/ENGINEER by building features. These features have intelligence, in that they contain knowledge of their environment and adapt predictably to change. Each features asks the user for specific information based on the feature type. For example, a hole has a diameter, depth, and placement, while a round has a radius and edges to round.

Associativity

Pro/ENGINEER is a fully associative system. This means that a change in the design model anytime in the development process is propagated throughout the design, automatically updating all engineering deliverables, including assemblies, drawings, and manufacturing data. Associativity makes concurrent engineering possible by encouraging change, without penalty, at any point in the development cycle. This enables downstream functions to contribute their knowledge and expertise early in the development cycle.

Capturing Design Intent

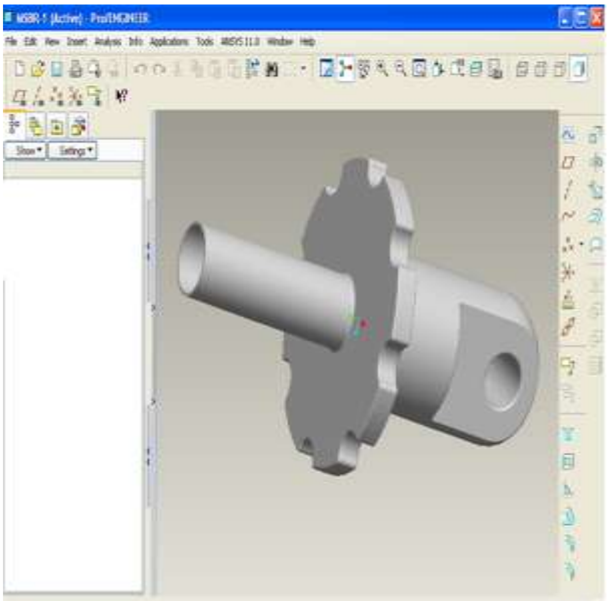
The strength of parametric modelling is in its ability to satisfy critical design parameters throughout the evolution of a solid model. The concept of capturing design intent is based on incorporating engineering knowledge into a model. This intent is achieved by establishing feature and part relationships and by the feature-dimensioning scheme. An example of design intent is the proportional relationship between the wall thickness of a pressure vessel and its surface area, which should remain valid even as the size of the vessel changes.

Combining Features into Parts

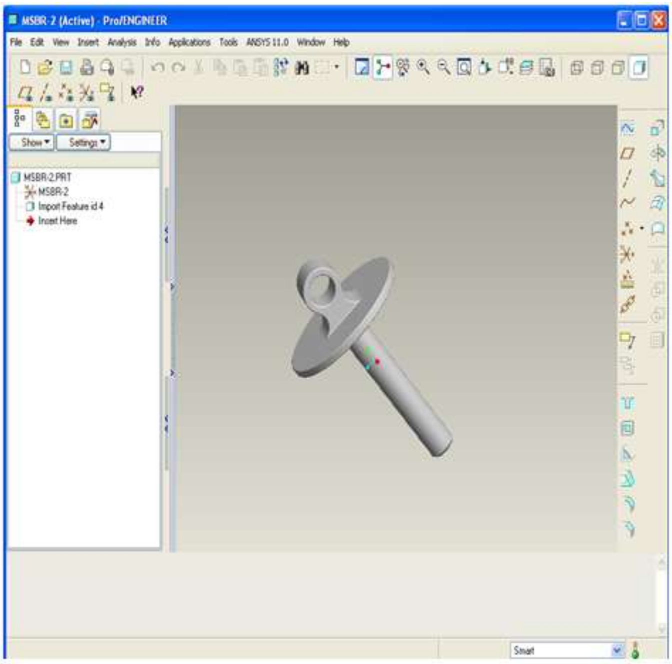
The various types of Pro/ENGINEER features serve as building blocks in the progressive creation of solid parts. Certain features, by necessity, precede others in the design process. The features that follow rely on the previously defined features for dimensional and geometric references. The progressive design of features can create relationships between features already in the design and subsequent features in the design that reference them. The following figure illustrates the progressive design of features

3.2 Model of Shock Absorber

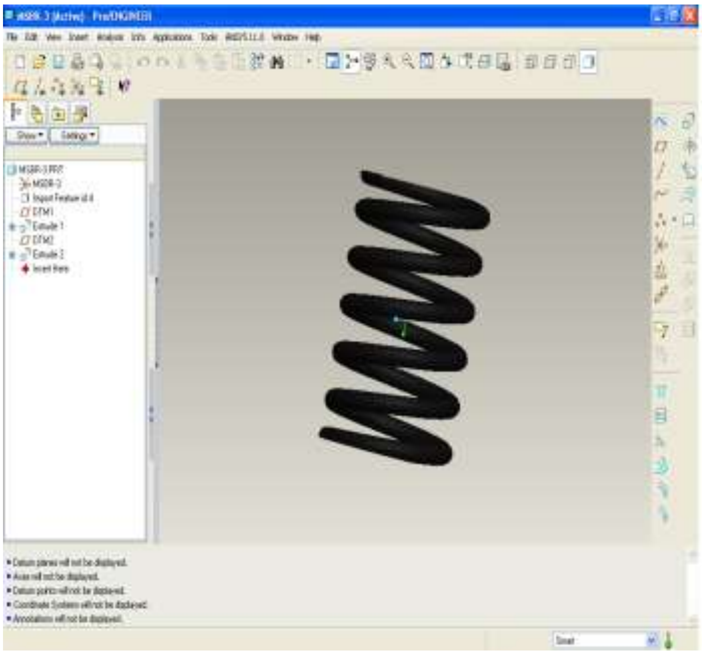
I] Bottom Part



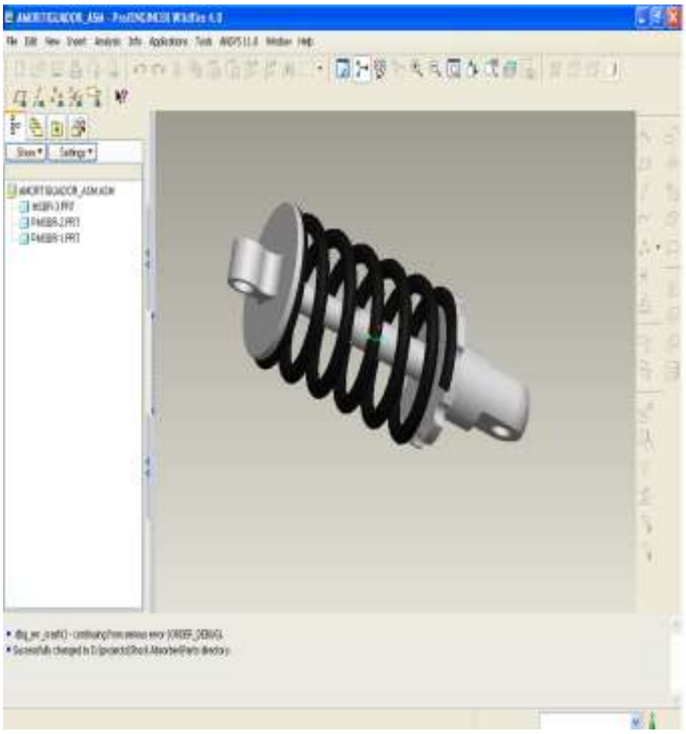
Ii] Top Part



iii] Helical Spring



Iv] Total Assembly



4 ANALYSIS

4.1 Finite Element Analysis

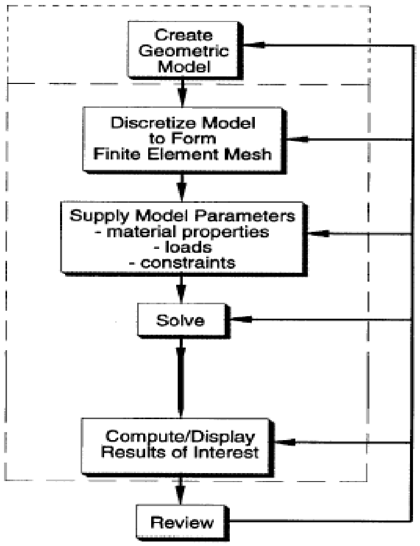
- Fem Introduction

The finite element method (FEM), sometimes referred to as *finite element analysis* (FEA), is a computational technique used to obtain approximate solutions of boundary value problems in engineering. Simply stated, a boundary value problem is a mathematical problem in which one or more dependent variables must satisfy a differential equation everywhere within a known domain of independent variables and satisfy specific conditions on the boundary of the domain. Boundary value problems are also sometimes called *field* problems. The field is the domain of interest and most often represents a physical structure.

The *field variables* are the dependent variables of interest governed by the differential equation. The *boundary conditions* are the specified values of the field variables (or related variables such as derivatives) on the boundaries of the field. Depending on the type of physical problem being analysed, the field variables may include physical displacement, temperature, heat flux, and fluid velocity to name only a few.

A General Procedure for Finite Element Analysis

Certain steps in formulating a finite element analysis of a physical problem are common to all such analyses, whether structural, heat transfer, fluid flow, or some other problem. These steps are embodied in commercial finite element software packages (some are mentioned in the following paragraphs) and are implicitly incorporated in this text, although we do not necessarily refer to the steps explicitly. The steps are described as follows.



Overall Steps in FEA Solution

- Pre-processing

The pre-processing step is, quite generally, described as defining the model and includes

- Define the geometric domain of the problem.
- Define the element type(s) to be used.
- Define the material properties of the elements.
- Define the geometric properties of the elements (length, area)
- Define the element connectivity's (mesh the model).
- Define the physical constraints (boundary conditions).
- Define the loadings.

The pre-processing (model definition) step is critical. In no case is there a better example of the computer-related axiom "garbage in, garbage out." A perfectly computed finite element solution is of absolutely no value if it corresponds to the wrong problem.

- Solution

During the solution phase, finite element software assembles the governing algebraic equations in matrix form and computes the unknown values of the primary field variable(s). The computed values are then used by back substitution to compute additional, derived variables, such as reaction forces, element stresses, and heat flow. As it is not uncommon for a finite element model to be represented by tens of thousands of equations, special solution techniques are used to reduce data storage requirements and computation time. For static, linear problems, a wave front solver, based on Gauss elimination, is commonly used.

- Post processing

Analysis and evaluation of the solution results is referred to as post processing. Postprocessor software contains sophisticated routines used for sorting, printing, and plotting selected results from a finite element solution. Examples of operations that can be accomplished include:

- Sort element stresses in order of magnitude.
- Check equilibrium.
- Calculate factors of safety.
- Plot deformed structural shape.
- Animate dynamic model behaviour.
- Produce colour-coded temperature plots.

4.2 Introduction To Ansys

ANSYS has developed product lines that allow you to make the most of your investment and choose which product works best in your environment. ANSYS is a Finite Element Analysis (FEA) code widely used in the Computer-Aided Engineering (CAE) field.

ANSYS FEA software is widely recognized as the leading fully integrated suite of Computer Aided Engineering tools and technologies, providing the most innovative and powerful simulation solutions to satisfy the ever-growing needs of organizations worldwide. ANSYS solutions are broad and highly integrated, beginning with very strong advanced technology in all key areas including Structural, CFD, Thermal, Electromagnetic, Dynamics, and Meshing.

- Specific Capabilities of ANSYS:

- Structural Analysis

It is used to determine displacements stresses, etc. under static loading conditions. ANSYS can compute both linear and nonlinear static analyses. Nonlinearities can include plasticity, stress stiffening, large deflection, large strain, hyper elasticity, contact surfaces, and creep.

4.3 Static Structural Analysis

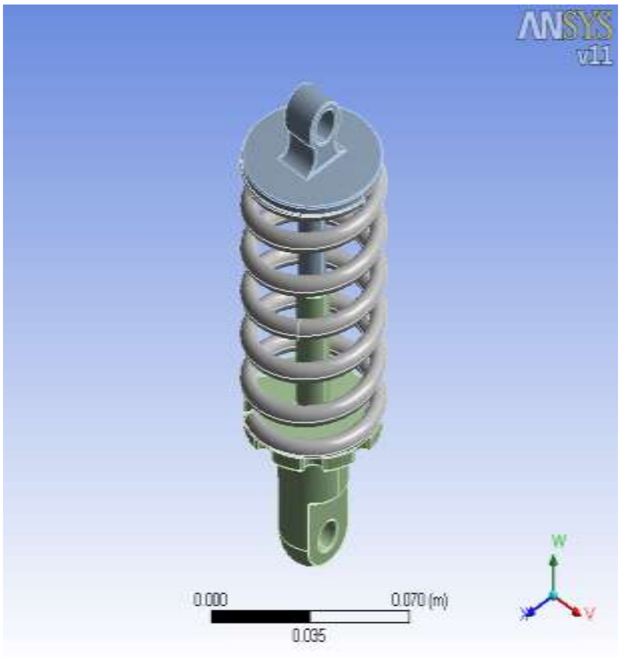
- Structural Analysis Of Existing Material ((67SiCr5 [Din 17221 Spring Steel Grade]))

CASE I: Considering the weight of 1 person (When W=1432 N)

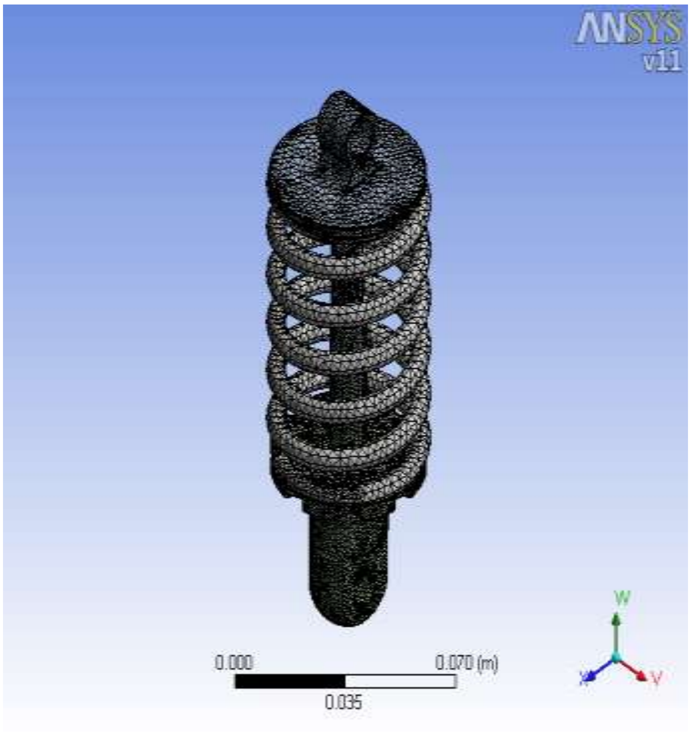
Material :((67SiCr5 [DIN 17221 SPRING STEEL GRADE]))
Properties: Young's Modulus (EX): 2100000000 Pa
Poisson's Ratio (PRXY): 0.27

Density: 7700 Kg/m^3

Step 1: Importing Shock Absorber Model in Ansys Workbench and Assigning the Material (Material Properties)

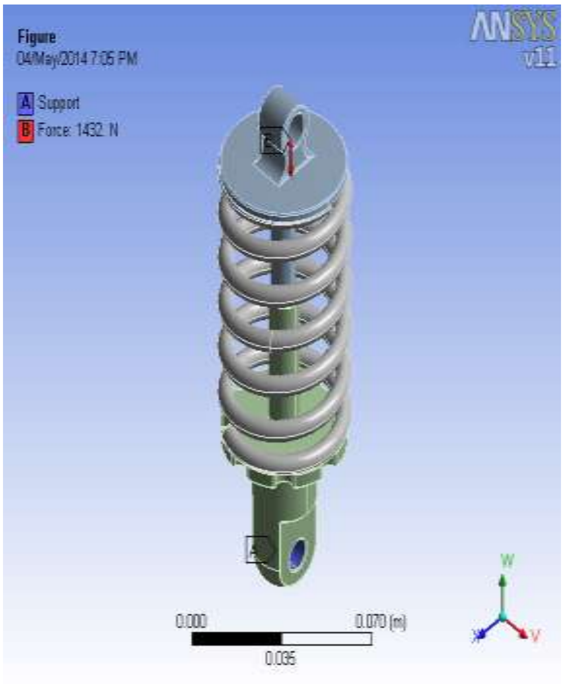


Step 2: Applying the Mesh (Meshing)



Step 3: Applying the Loads and Boundary Conditions

The following figure shows that the loading and boundary condition on a shock absorber, this load is applied according to the design of spring, Also the boundary condition is applied at fixed position of a model which is consider as a supports, The load is applied considering weight of one person at top part of shock absorber model.



Step 4: Obtaining results after applying loads

Figure shows the total deflection in the shock absorber when the load of 1432 N is applied for the existing material 67 SiCr5.

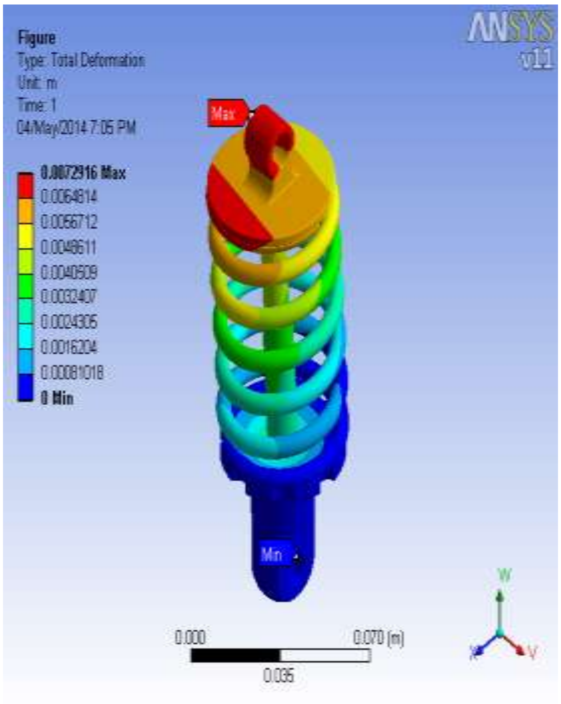


Fig 4.1:- Total Deflection for load of 1 person

Figure shows the Maximum Shear stress in the shock absorber when the load of 1432 N is applied for the existing material 67SiCr5. Red colour shows the Maximum stress and blue colour shows minimum stress generated.

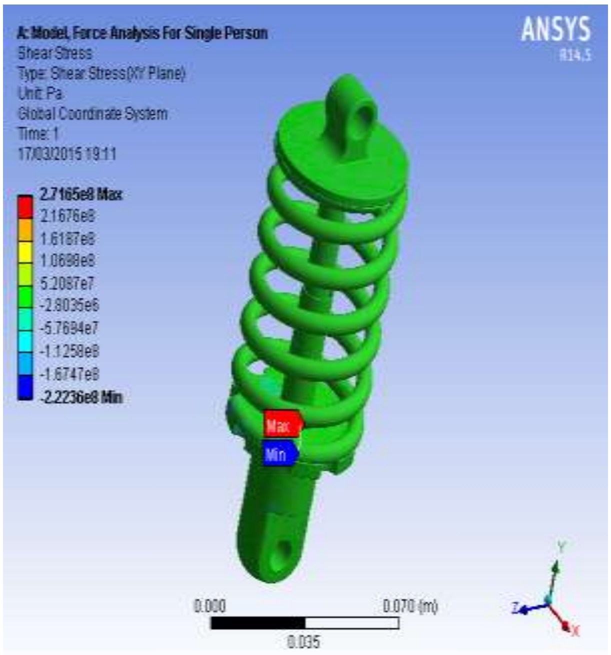


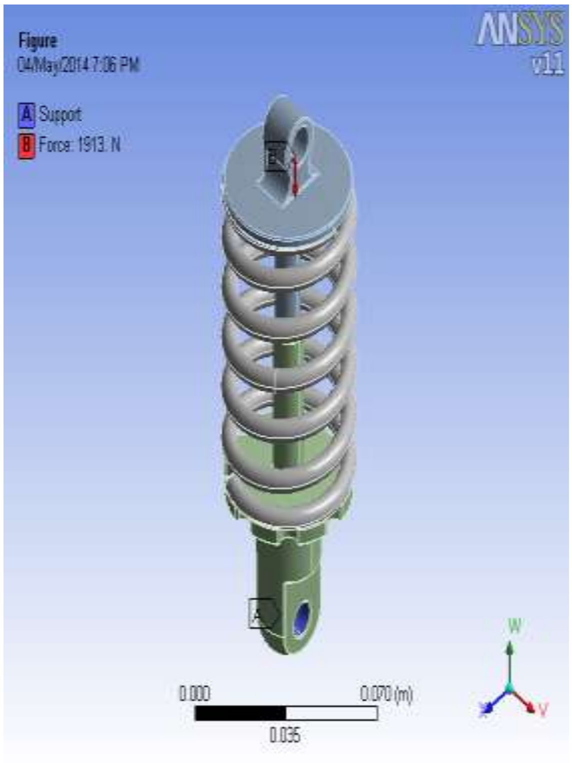
Fig 4.2:- Maximum Shear Stress for load of 1 person

CASE II: Considering the weight of 2 persons (When W=1913 N)

Material :((67SiCr5 [DIN 17221 SPRING STEEL GRADE]))

Properties: Young's Modulus (EX): 2100000000 Pa
Poisson's Ratio (PRXY): 0.27
Density: 7700 Kg/m^3

Step 1: Importing Shock Absorber Model in Ansys Workbench and Assigning the Material (Material Properties)
Applying the Loads and Boundary Conditions:
The following figure shows that the loading and boundary condition on a shock absorber, this load is applied according to the design of spring, Also the boundary condition is applied at fixed position of a model which is consider as a supports, The load is applied considering weight of two persons at top part of shock absorber model.



Step 2: Obtaining results after applying loads

Figure shows the total deflection in the shock absorber when the load of 1913 N is applied for the existing material 67SiCr5.

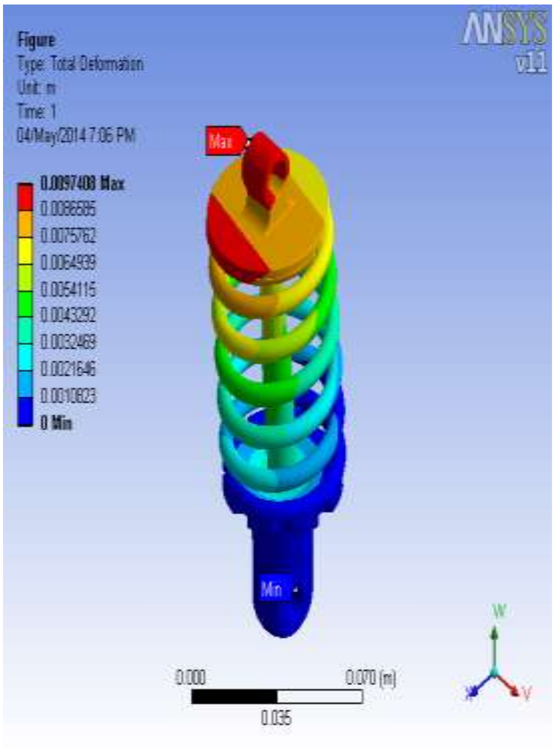


Fig 4.3:- Total Deflection for load of 2 persons

Following Figure shows the Maximum Shear stress in the shock absorber when the load of 1913 N is applied for the existing material 67SiCr5. Red colour shows the Maximum stress and blue colour shows minimum stress generated.

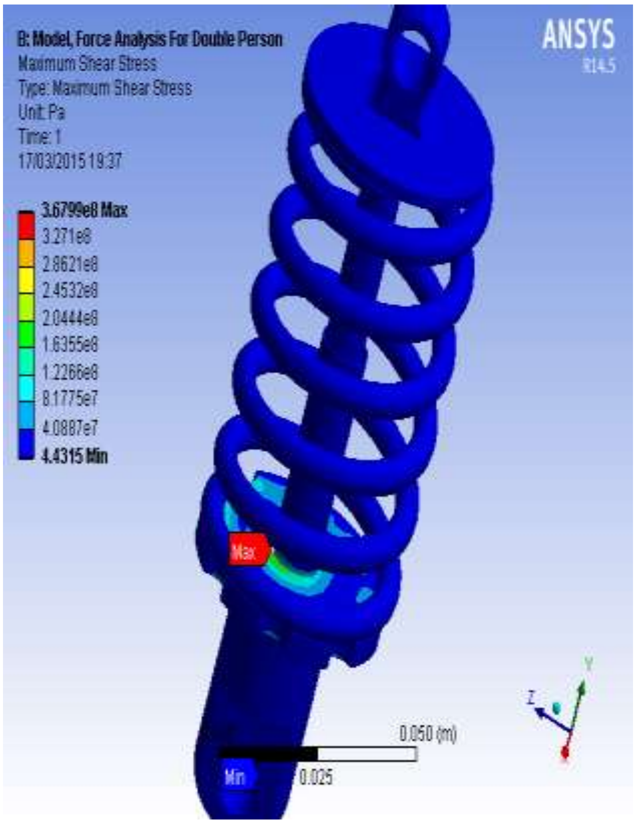


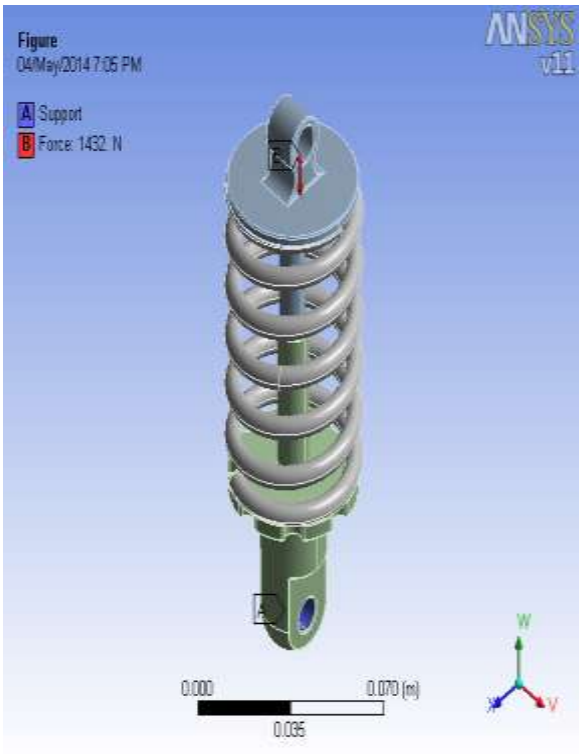
Fig 4.4:- Maximum Shear Stress for load of 2 persons

- Structural Analysis Of Suggested Material ((ASTM A228 [High Carbon Spring Wire]))

CASE I: Considering the weight of 1 person (When W=1432 N)

Material:(ASTM A228 [HIGH CARBON SPRING WIRE])

Properties: Young's Modulus (EX): 20850000000
Poisson's Ratio (PRXY): 0.31
Density: 7860 Kg/m^3
Step 1: Importing Shock Absorber Model in Ansys Workbench and Assigning the Material (Material Properties)
Applying the Loads and Boundary Conditions:
The following figure shows that the loading and boundary condition on a shock absorber, this load is applied according to the design of spring, Also the boundary condition is applied at fixed position of a model which is consider as a supports, The load is applied considering weight of one person at top part of shock absorber model.



Step 2: Obtaining results after applying loads

Figure shows the total deflection in the shock absorber when the load of 1432 N is applied for the suggested material ASTM A228.

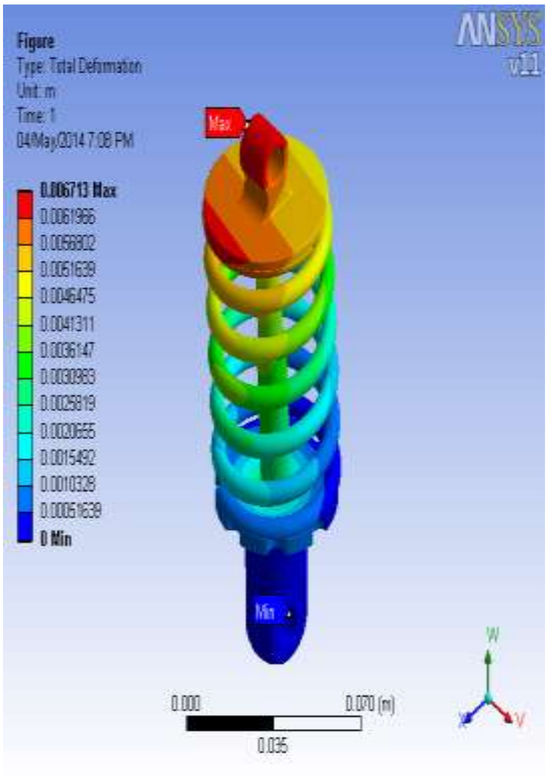


Fig 4.5:- Total Deflection for load of 1 person

Figure shows the Maximum Shear stress in the shock absorber when the load of 1432 N is applied for the suggested material ASTM A228. Red colour shows the Maximum stress and blue colour shows minimum stress generated.

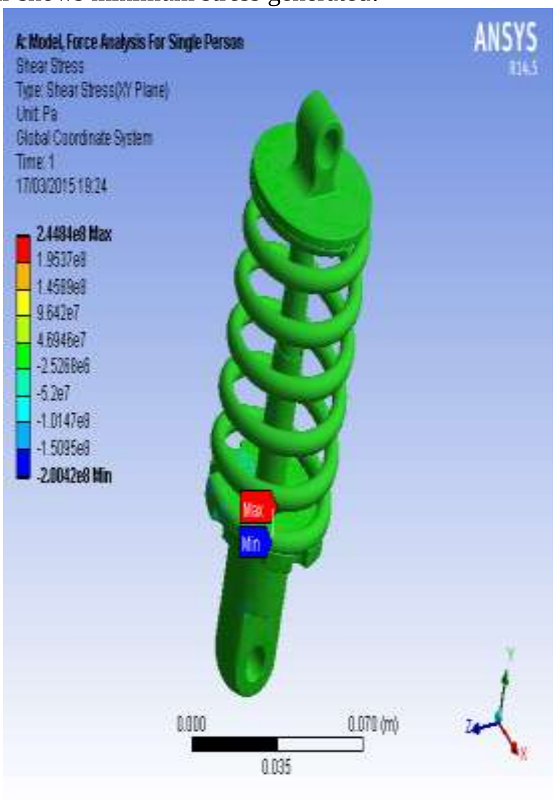


Fig 4.6:- Maximum Shear Stress for load of 1 person

CASE II: Considering the weight of 2 persons (When W=1913 N)

Material :(ASTM A228 [HIGH CARBON SPRING WIRE])
Properties: Young's Modulus (EX): 20850000000 Pa
Poisson's Ratio (PRXY): 0.31

Density: 7860 Kg/m^3

Step 1: Importing Shock Absorber Model in Ansys Workbench and Assigning the Material (Material Properties)

Applying the Loads and Boundary Conditions:
The following figure shows that the loading and boundary condition on a shock absorber, this load is applied according to the design of spring, Also the boundary condition is applied at fixed position of a model which is consider as a supports, The load is applied considering weight of one person at top part of shock absorber model.

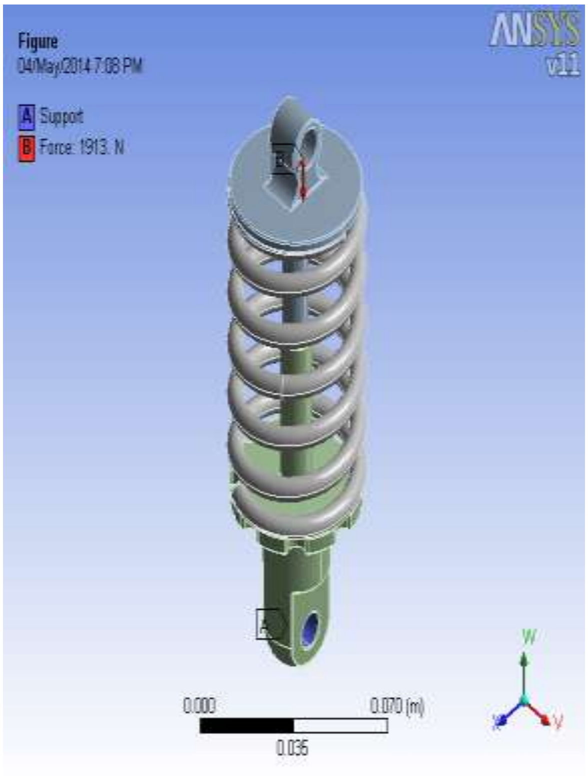


Fig. 4.7 Loading and Boundary Condition

Step 2: Obtaining results after applying loads
Figure shows the total deflection in the shock absorber when the load of 1913 N is applied for the suggested material ASTM A228.

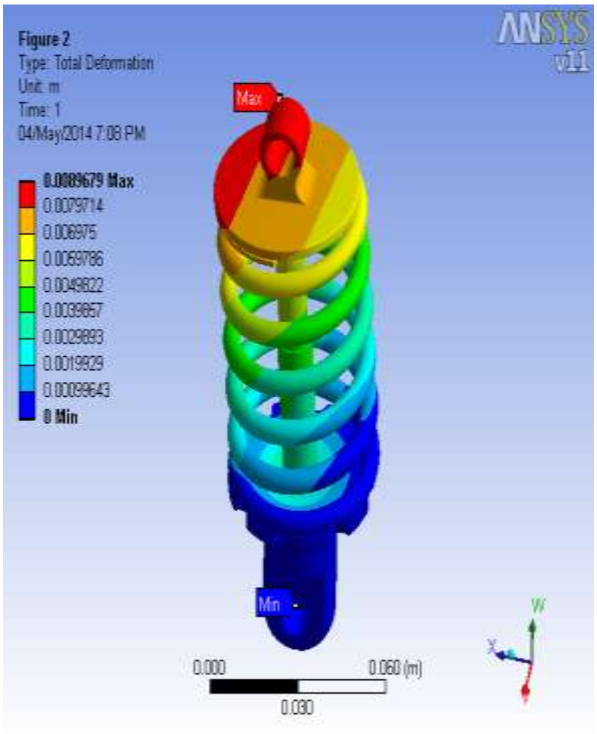


Fig 4.8:- Total Deflection for load of 2 persons

Figure shows the Maximum Shear stress in the shock absorber when the load of 1913 N is applied for the suggested material ASTM A228. Red colour shows the Maximum stress and blue colour shows minimum stress generated.

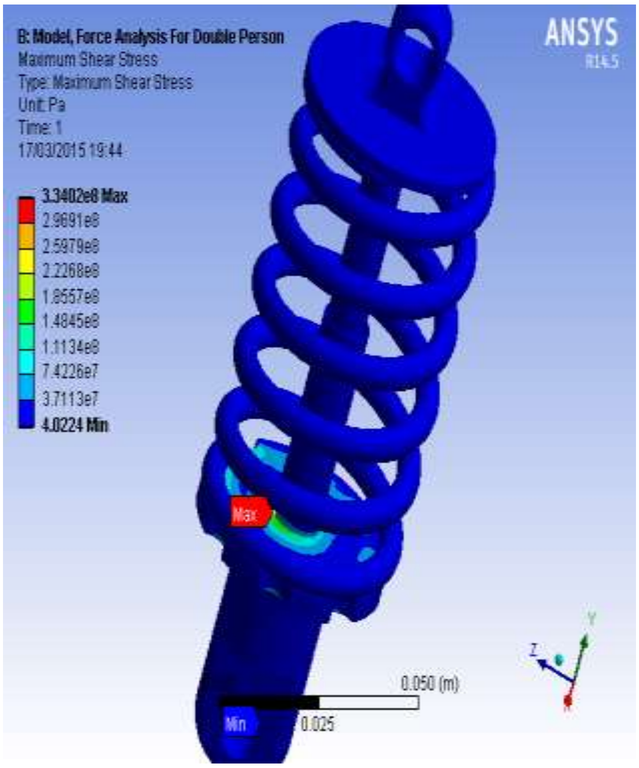


Fig 4.9:- Maximum Shear Stress for load of 2 person

4.4 Modal Analysis

- A modal analysis is typically used to determine the vibration characteristics (natural frequencies and mode shapes) of a structure or a machine component while it is being designed. It can also serve as a starting point for another, more detailed, dynamic analysis, such as a harmonic response or full transient dynamic analysis. Modal analyses, while being one of the most basic dynamic analysis types available in ANSYS, can also be more computationally time consuming than a typical static analysis. A reduced solver, utilizing automatically or manually selected master degrees of freedom is used to drastically reduce the problem size and solution time

- Modal Analysis Results For Existing Material ((67sicr5 [Din 17221 Spring Steel Grade]))

The following bar chart indicates the frequency at each calculated mode:

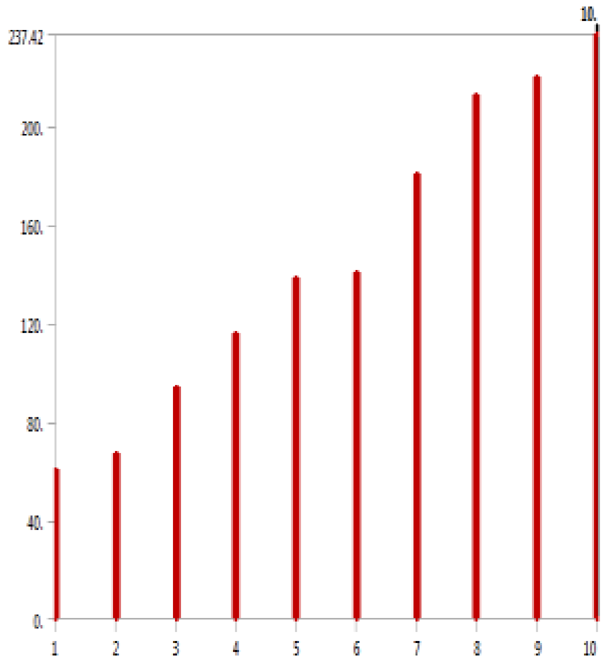


Fig 4.10 Frequency of shock absorber for existing material

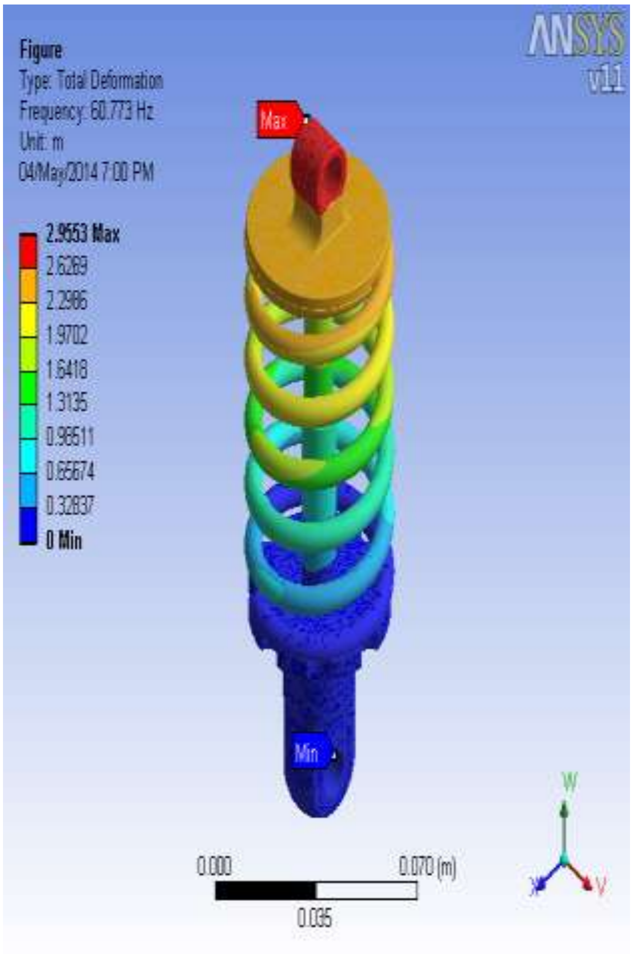


Fig 4.11 :- First Mode Shape

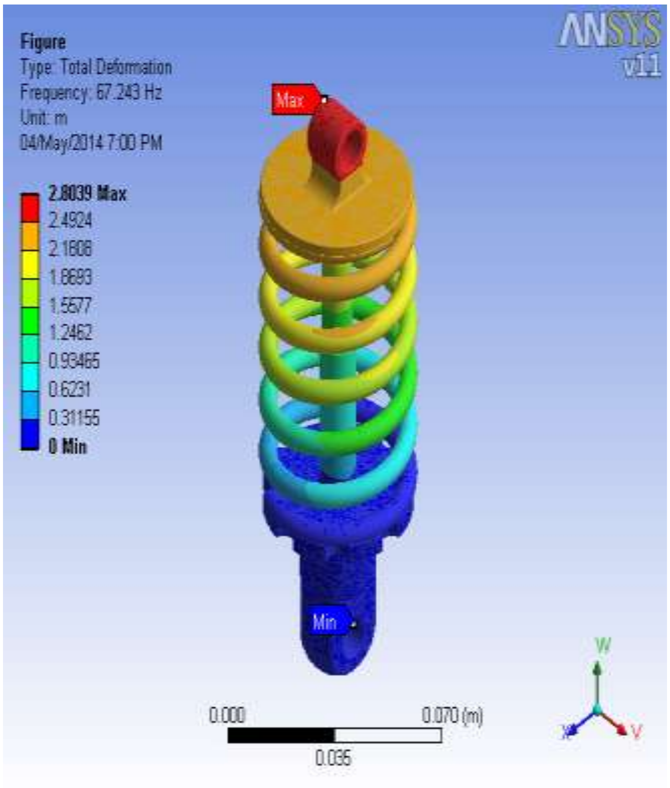


Fig 4.12:- Second Mode Shape

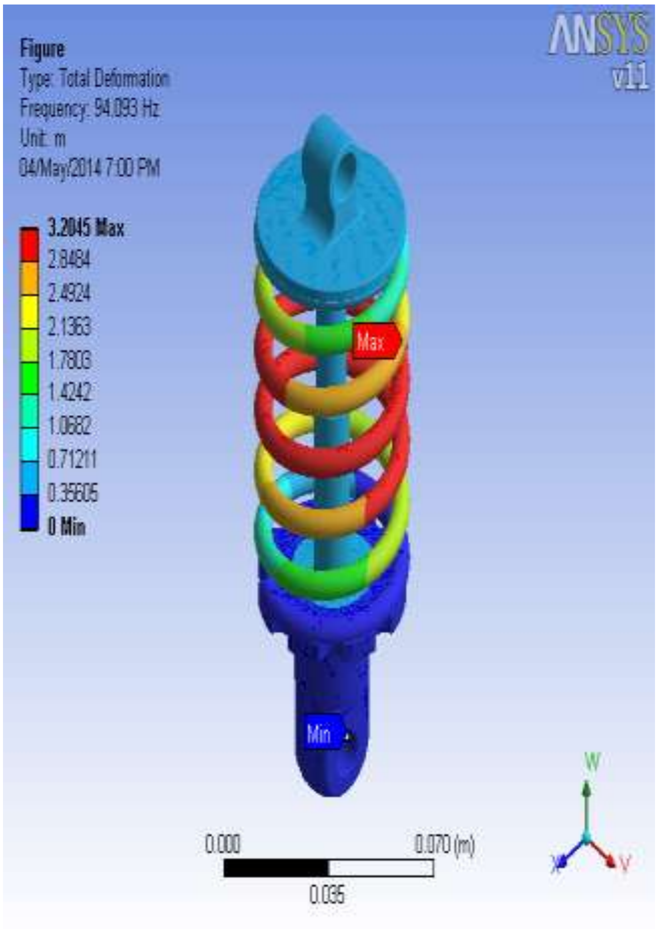


Fig 4.13 :- Third Mode Shape

TABLE 1
Model > Modal > Solution

Mode	Frequency [Hz]
1.	60.773
2.	67.243
3.	94.093
4.	115.63
5.	138.68
6.	140.72
7.	180.52
8.	212.66
9.	219.91
10.	237.42

Table 4.1: Frequency at ten mode shapes for existing material

- Modal Analysis Results For Suggested Material ((ASTM A228 [High Carbon Spring Wire]))

The following bar chart indicates the frequency at each calculated
mode

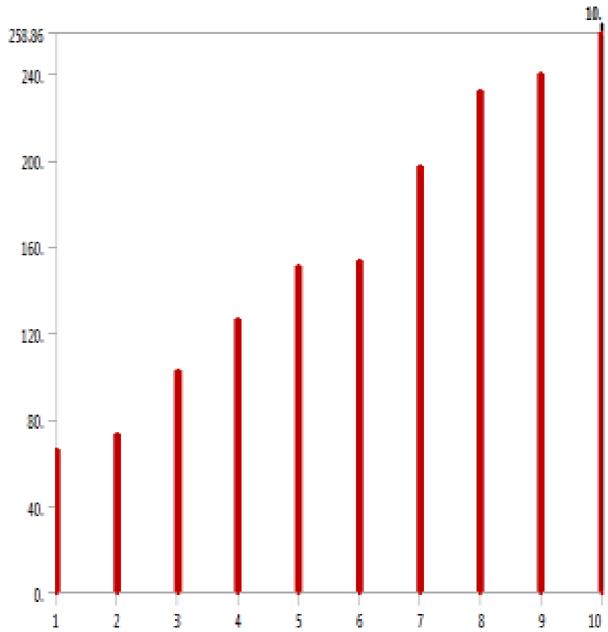


Fig 4.14. Frequency of shock absorber for suggested material

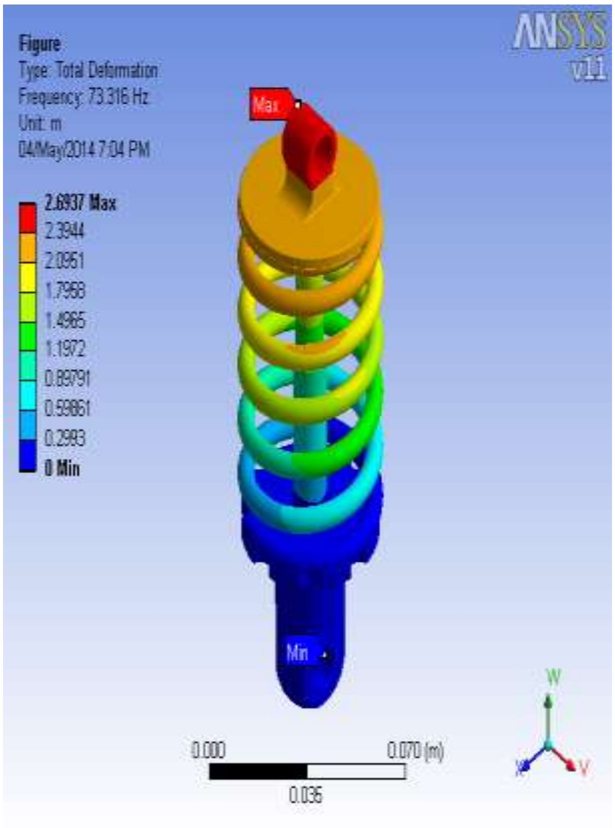


Fig 4.16:- Second Mode Shape

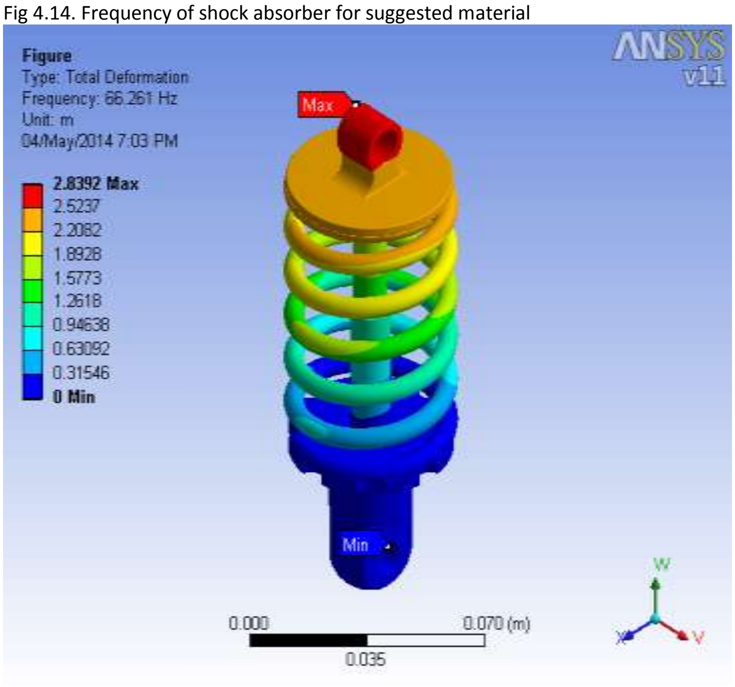


Fig 4.15 :- First Mode Shape

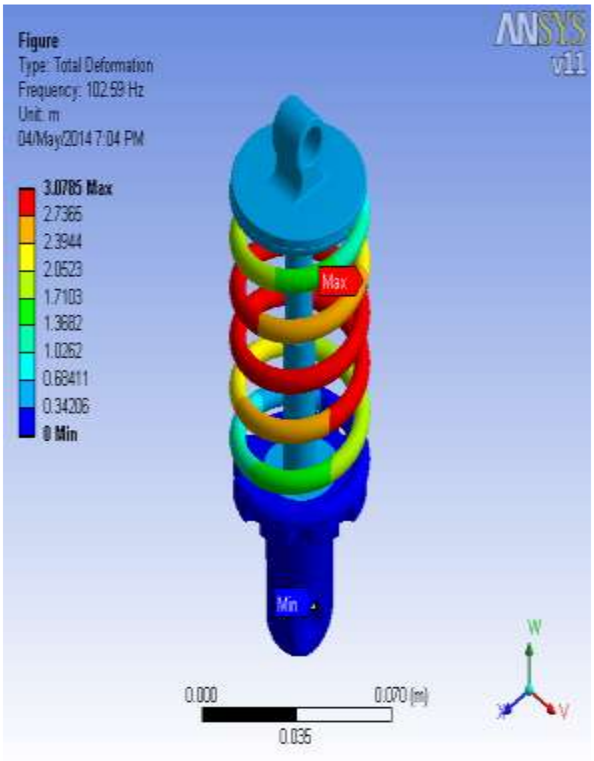


Fig 4.17:- Third Mode Shape

TABLE 2
Model > Modal > Solution

Mode	Frequency [Hz]
1.	66.261
2.	73.316
3.	102.59
4.	126.07
5.	151.21
6.	153.43
7.	196.82
8.	231.86
9.	239.77
10.	258.86

Table 4.2 : Frequency at ten mode shapes for suggested material

5 RESULTS AND DISCUSSION

Deflection Result Table

Total Deflection in mm

Material	Analytical		Ansys Workbench	
	1 Person Load [W=1432 N]	2 Persons Load [1913 N]	1 Person Load [W=1432 N]	2 Persons Load [1913 N]
1) Existing Material [67SiCr5] Spring Steel	7.23	9.66	7.29	9.74
2) Suggested Material [ASTM A228] Spring Steel	6.70	8.90	6.71	8.96

Table 5.1: Reduction in deflection

Graph

Reduction in Deflection (mm)

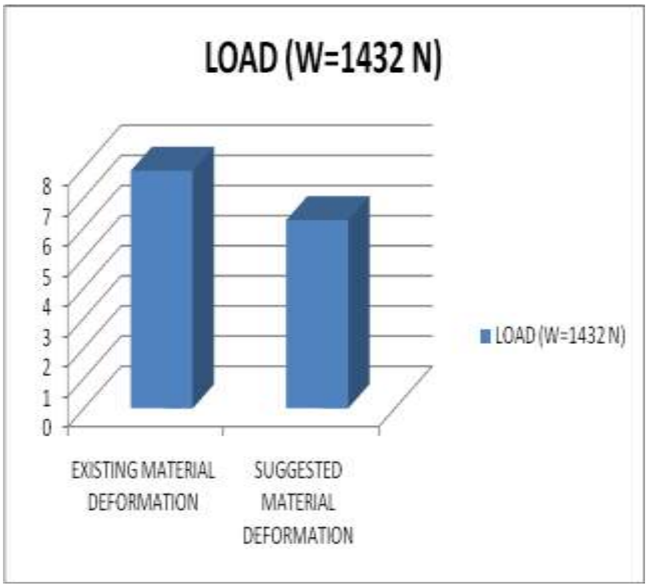


Fig 5.1. Reduction in deflection for load of single person

Fig 5.2. Reduction in deflection for load of two persons

Shear Stress Result Table:

Maximum Shear Stress in N/mm^2

Material	Analytical		Ansys Workbench	
	1 Person Load [W=1432 N]	2 Persons Load [1913 N]	1 Person Load [W=1432 N]	2 Persons Load [1913 N]
1) Existing Material [67SiCr5] Spring Steel	274	366	271	367
2) Suggested Material [ASTM A228] Spring Steel	274	366	244	334

Table 5.2: Reduction in deflection

Graphs

Reduction in stress (N/mm^2)

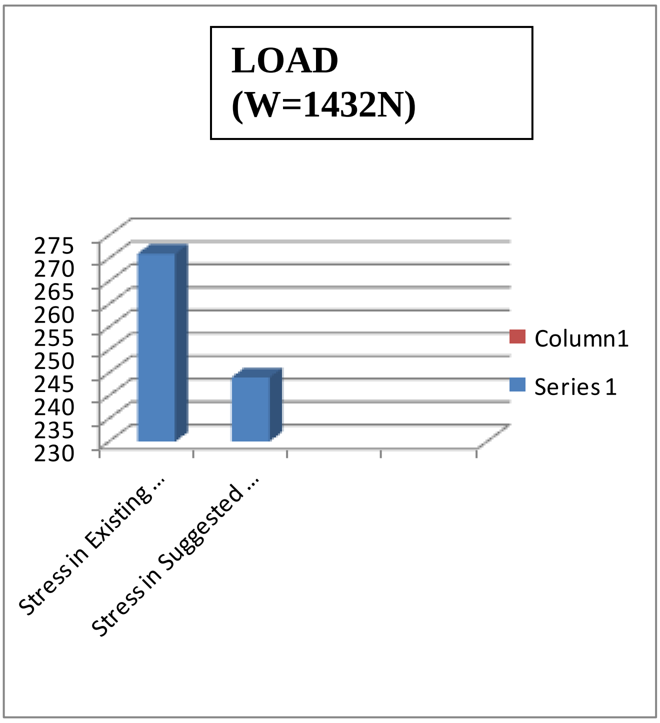


Fig 5.3 Reduction in Stress for load of single person

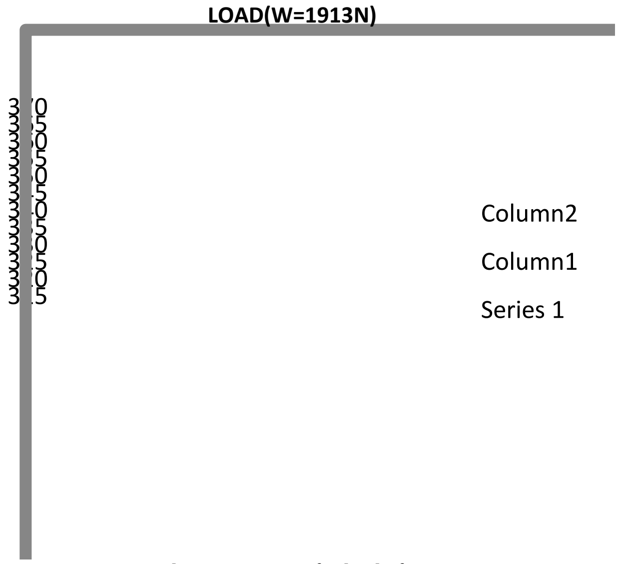


Fig 5.4 Reduction in Stress for load of two persons

Natural Frequency Result Table:

Natural Frequency in Hz

TABLE 3
Model > Modal > Solution

Material	Existing Material 67SiCr5	Suggested Material ASTM A228
Mode	Frequency [Hz]	Frequency [Hz]
1.	60.773	66.261
2.	67.243	73.316
3.	94.093	102.59
4.	115.63	126.07
5.	138.68	151.21
6.	140.72	153.43
7.	180.52	196.82
8.	212.66	231.86
9.	219.91	239.77
10.	237.42	258.86

Table 5.3: Frequency at ten mode shapes for both material

Graphs

Natural Frequency

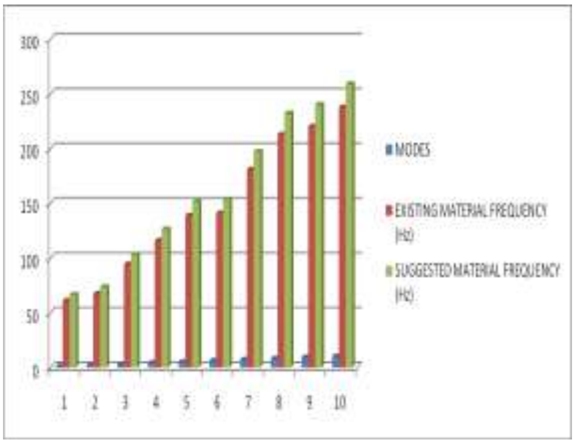


Fig 5.5. Increase in Natural Frequency

CONCLUSION

1. In this project we have designed a shock absorber. We have modeled the shock absorber by using 3D parametric software Pro/Engineer.

2. To validate the strength of our design, we have done structural analysis and modal analysis on the shock absorber. We have done analysis by changing existing material ((67SiCr5 [din 17221 spring steel grade])) to suggested material ((ASTM A228 [high carbon spring wire]))
3. By observing the analysis results, the analyzed stress values are less than their respective yield stress values. So our design is safe.

4. By comparing the results for both materials, the total deflection value is less for ASTM A228 than 67SiCr5. So stiffness is more for Spring Steel.

5. By comparing the results for both materials, the Natural Frequency is more for ASTM A228 than 67SiCr5.

6. So we can conclude that as per our analysis using ASTM A228 [high carbon spring wire] for spring is best.

REFERENCES

1) John C. Dixon, "The Shock Absorber Handbook", 2nd Ed., John Wiley & Sons Ltd, England, 2007
2) Mr. SudarshanMartande, Design and Analysis of Shock Absorber, IJAIEM, Volume 2, Issue 3, March 2013 ISSN 2319 – 4847
3) M.S.M.Sani, M.M. Rahman, M.M.Noor, Study on Dynamic Characteristics of Automotive Shock Absorber System, Malaysian Science and Technology Congress, MSTC08, 16~17 Dec, KLCC, Malaysia, 2008.
4) PiotrCzop, Paweliwa, A COMPUTATIONAL FLUID FLOW ANALYSIS OF A DISC VALVE SYSTEM, Journal of KONES Powertrain and Transport, Vol. 18, No. 1 2011
5) Reybrouck K.G., A Non Linear Parametric Model of an Automotive Shock Absorber, SAE Technical Paper Series 940869, 1994
6) Talbots M.S., An Experimentally Validated Physical Model of a High Performance Automotive Damper, M.S. Thesis, Purdue University, Lafayette, IN, 2002
7) Pinjarla.Poornamohan, 'DESIGN AND ANALYSIS OF A SHOCK ABSORBER', IJRET DEC 2012, Volume: 1 Issue: 4 578 – 592
8) Achyut P. Banginwar ,Nitin D. Bhusale, Kautuk V. Totawar , Design and analysis of shock absorber using FEA tool , *Volume 10, Issue 2 (February 2014), PP.22-28*
9) R. S. Khurmi and J. K. Gupta, „Machine Design‟ , Fourteen Edition, 2012, Eurasia Publishing House, pp. 820-833.