DESIGN, ANALYSIS AND OPTIMISATION OF SHOCK ABSORBER

KARAN CHOUGULE[1], ADITYA PARTE[1], MILIND DEEPAK HAVAL[1], MANIK RAJANGALE[2]

[1]Student, Dept. of Mechanical Engineering, Maratha Mandal Engineering College, Belagavi
[2]Assistant Professor, Dept. of Mechanical Engineering, Maratha Mandal Engineering College, Belagavi

---

Abstract:- *A suspension system or shock absorber is a mechanical device designed to smooth out or damp shock impulse, and dissipate kinetic energy. The shock absorbers duty is to absorb or dissipate energy. In this project a shock absorber is designed and a 3D model is created using CATIA. Structural analysis and modal analysis are done on the shock absorber by varying material for spring, Spring Steel and Phosphor Bronze. Structural analysis is done to validate the strength and modal analysis is done to determine the displacements for different frequencies for number of modes. Comparison is done for two materials to verify best material for spring in Shock absorber. Modeling is done in CATIA and analysis is done in ANSYS.*

1. INRODUCTION

A shock absorber or damper is a mechanical device designed to smooth out or damp shock impulse, and dissipate kinetic energy. When a vehicle is traveling 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. The design of spring in suspension system is very important.

1.1 Description

Pneumatic and hydraulic shock absorbers commonly take the form of a cylinder with a sliding piston inside. The cylinder is filled with a fluid (such as hydraulic fluid) or air. Though is fluid-filled piston/cylinder combination is a dashpot.

1.2 Explanation

The shock absorbers duty is to absorb or dissipate energy. One design consideration, when designing or choosing a shock absorber, is where that energy will go. In most dashpots, energy is converted to heat inside the viscous fluid. In hydraulic cylinders, the hydraulic fluid will heat up, while in air cylinders, the hot air is usually exhausted to the atmosphere. In other types of dashpots, such as electromagnetic ones, the dissipated energy can be stored and used later. In general terms, shock absorbers help cushion cars on uneven roads.

1.3 Applications

Shock absorbers are an important part of auto mobile and motorcycle suspensions, aircraft landing gear, and the supports for many industrial machines. Large shock absorbers have also been used in structural engineering to reduce the susceptibility of structures to earthquake damage and resonance. A transverse mounted shock absorber, called a yaw damper, helps keep railcars from swaying excessively from side to side and are important in passenger railroads, commuter rail and rapid transit systems because they prevent railcars from damaging station platforms. The success of passive damping technologies in suppressing vibration amplitudes could be ascertained with the fact that it has a market size of around $ 4.5 billion.

Fig: 1.3.1 Rear shock absorber and spring

1.4 Modal analysis:-

Modal analysis provides an overview of the limits of the response of a system. For example, for a particular input (like an applied load of certain amplitude and frequency) what are the limits of the system response (like when and what is the maximum displacement).

Every system can be described in terms of a stiffness matrix that connects the displacements (or system response) and forces (or system inputs). These frequencies are known as natural frequencies of the system and given by the eigenvectors of the stiffness matrix. These frequencies are also known as the resonant frequencies.

When the frequency of loading is the same as averaged vibrational frequency of the atoms in the structure, the energy is transferred with minimum loss. In other words, in simple terms, one can think of it as two waves (one being the external load and another being that of the internal atomic structure) that are being superimposed. When the frequencies are the same, they tend to add up. Greater the frequency safer will be the component.

There are several examples where a prior accurate modal analysis could have prevented loss to lives and property – e.g. allowing a better bridge design in case of an earthquake etc.

2. LITRATURE REVIEW

This literature review is carried out to understand the present practices and theories in shock absorber design and analysis. Kim (1993) performed an analysis of a twin tube damper with focus on implementation into a vehicle suspension system. Kim's model included chamber compliance and fluid compressibility which yielded a differential equation for the chamber pressures that was solved using the Runga Kutta Method. Incorporating damping data into a quarter car model, the frequency response of the sprung mass and tire deflection were calculated numerically.

Ferdek et al. (2012) designed a physical and mathematical model for a twin-tube hydraulic shock absorber, using oil as the working medium. To analyze the model, methods of numerical integration were incorporated. The basic characteristics of the damping force were also obtained. The velocity sensitive and nominally symmetric hydraulic dampers were considered.(2010) presented the model of a complete system, consisting of a variable damping shock absorber and a servo-hydraulic tester, used to evaluate the vibration levels produced by a shock absorber. Also, same group (2011) gives the configurations of a typical valve system including three basic regimes of operation, which correspond to the amount of oil flowing through a valve cavity. The aim of this work was to propose a finite element fluid flow model, which can be used in order to reduce the velocity of fluid flow through a cavity of a shock absorber valve.

Lee (2005) suggested a new mathematical model of displacement sensitive shock absorber to predict the dynamic characteristics of automotive shock absorber. The performance of shock absorber is directly related to the vehicle behaviors and performance, both for handling and ride comfort. Poornamohan (2012) designed shock absorber for 150 cc bike using CATIA. The model was also changed by changing the thickness of the spring. They have performed the structural analysis and modal analysis on the shock absorber by varying material for spring, to steel and Beryllium Copper. Martande (2013), focused on to develop new correlated methodologies that will allow engineers to design components of shock absorbers by using FEM based tools. The different stress and deflection values in shock absorber components have been obtained using FEA tools and compared with analytical solutions. Percentage error is calculated and it is found that percentage error is less than 15%.

3. CATIA MODEL DESIGN

3.1 Introduction to CATIA

CATIA is one of the world's leading high-end CAD/CAM/CAE software packages. CATIA (computer aided three dimensional interactive application) is a multi-platform PLM/CAD/CAM/CAE commercial software suite developed by Dassault systems and marketed worldwide by IBM. CATIA is written in the C++ programming language.

CATIA provides open development, architecture through the use of interfaces, which can be used to customize or develop applications. The applications in programming interfaces supported visual basic and C++ programming languages.

Commonly referred to as 3D product Lifecycle management (PLM) software suite, CATIA supports multiple stages of product development. The stages range from conceptualization, through design (CAD) and manufacturing (CAM), until analysis (CAE). Each work bench of catiaV5 refers and each stage of product development for different products. CATIA V5 features a parametric solid/surface-based package which uses NURBS as the core surface representation and has several work benches but provide KBE (knowledge based engineering) support.

3.2 Fundamentals of CATIA:

The Part Design Workbench is used to create Solid geometry using a Feature based approach. In general the features are produced from sketches created in the Sketcher workbench. The specification tree contains all the features created along with the sketch used to define them. All the Solid features are contained within a node called a Part Body. They also contain wireframe sketches that are used to create the features. As you create features they are added to the tree in order of creation.

There may be multiple Part bodies within a CAT Part which can be Boo learned together in order to form complex solid models. Part bodies can be added to the Specification Tree by selecting Body from the Insert drop down menu when in the Part Design Workbench. The Part body can then be renamed by editing its properties.

3.3 EXPERIMENTAL PROCEDURE:

Aim: To design a 3D model of a shock absorber by using CATIA V5 R20. **Equipment:** Intel Core 2 duo computer installed with CATIA V5 R19. **Software**:CATIA V5 R19.

Step 1: Click Start in Menu bar >Mechanical Design > Sketcher.

Step 2: Select XY Sketch plane.

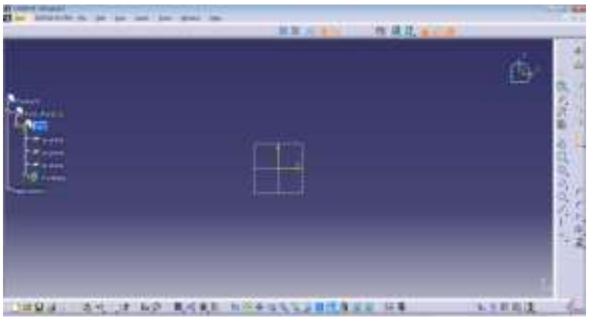


Fig:3.3.1CATIA software display

Fig: 3.3.2 Bottom rod design

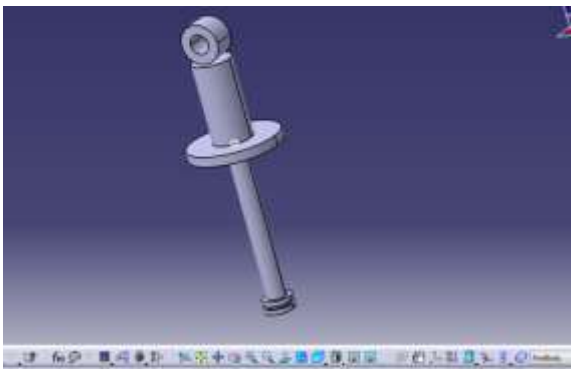


Fig:3.3.3 Top rod design

Fig:3.3.4 Spring design

Fig: 3.3.5 Assembly of shock absorber

Fig: 3.3.6 Exploded view

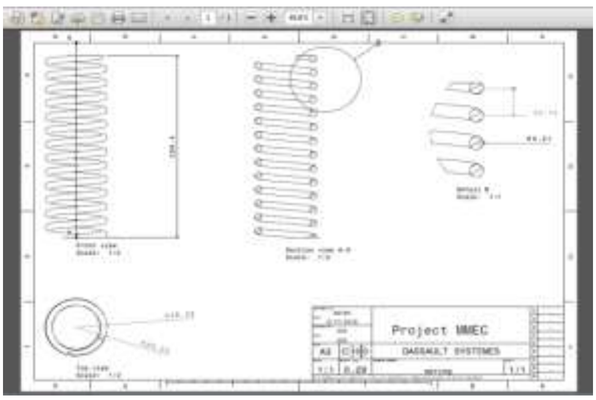


Fig: 3.3.7 2D drawing of spring

International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 06 Issue: 06 | June 2019 www.irjet.net p-ISSN: 2395-0072

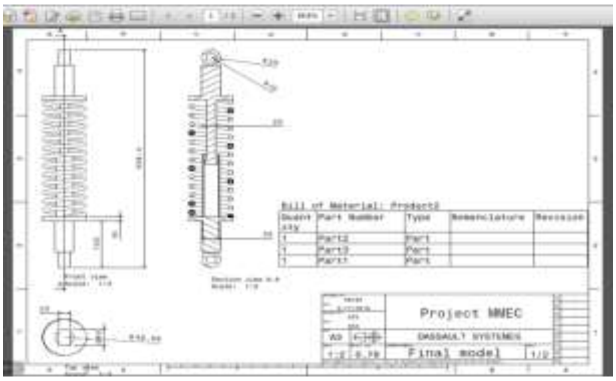


Fig: 3.3.8 2D assembly drawing

4. FINITE ELEMENT ANALYSIS

4.1. INTRODUCTION

Finite Element Analysis (FEA) is a computer-based numerical technique for calculating the strength and behavior of engineering structures. It can be used to calculate deflection, stress, vibration, buckling behavior and many other phenomena. It can be used to analyze either small or large-scale deflection under loading or applied displacement. It can analyze elastic deformation, or "permanently bent out of shape" plastic deformation. The computer is required because of the astronomical number of calculations needed to analyze a large structure.

The power and low cost of modern computers has made Finite Element Analysis available to many disciplines and companies. In the finite element method, a structure is broken down into many small simple blocks or elements. The behavior of an individual element can be described with a relatively simple set of equations. Just as the set of elements would be joined together to build the whole structure, the equations describing the behaviors of the individual elements are joined into an extremely large set of equations that describe the behavior of the whole structure. The computer can solve this large set of simultaneous equations. From the solution, the computer extracts the behavior of the individual elements. From this, it can get the stress and deflection of all the parts of the structure. The stresses will be compared to allowed values of stress for the materials to be used, to see if the structure is strong enough

The term "finite element" distinguishes the technique from the use of infinitesimal "differential elements" used in calculus, differential equations, and partial differential equations. The method is also distinguished from finite differential equations, for which although the steps into which space is divided are finite in size, there is little freedom in the shapes that the discreet steps can take. Finite element analysis is a way to deal with structures that are more complex than can be dealt with analytically using partial differential equations. FEA deals with complex boundaries better than finite difference equations will, and gives answers to "real world" structural problems. It has been substantially extended in scope during the roughly 40 years of its use

4.2 General description of the FEA

In the finite element analysis, the actual continuum or body of matter like solid, Liquid or gas is represented as an assemblage of subdivisions called finite elements. These elements are considered to be interconnected at specified joints called nodes or Nodal points. The nodes usually lie on the element boundaries where adjacent elements are considered to be connected since actual variation of the field variable inside a finite variable inside a finite element can be approximated by a simple function. These approximate functions are defined in terms of the value of the field's variables at the nodes. By solving the field equations, which are generally in the form of matrix equations, the nodal values of the field variable will be known.

4.3 STEPS IN FINITE ELEMENT ANALYSIS:

The steps in the finite element method when it is applied to structural mechanics are as follows.

1. Divide the continuum into a finite number of sub regions (or elements) of simple geometry such as line segments, triangles, quadrilaterals (Square and rectangular elements are subset of quadrilateral), tetrahedrons and hexahedrons (cubes) etc.

2. Select key point on the elements to serve as nodes where conditions of equilibrium and compatibility are to be enforced.

3. Assume displacement functions within each element so that the displacements at each generic point are depending upon nodal values.

4. Satisfy strain displacement and stress– strain relationship within a typical element. Determine stiffness and equivalent nodal loads for a typical element using work or energy principles.

5. Develop equilibrium equations for the nodes of the discredited.

6. Continuum in terms of the element contributions.

7. Solve the equilibrium for the nodal displacements
The basic premise of the FEM is that a solution region can be analytically modeled or approximated by replacing with an assemblage of discrete elements. Since these elements can be put together in a variety of ways,

they can be used to represent exceedingly complex shapes. The important feature of the FEM, which sets it apart from the other approximate numerical methods, is the ability to format solutions for the individual element before putting them together to represent the entire problem. Another advantage of FEM is the variety of ways in which one can formulate the properties of individual elements.

FEM can be broadly classified in to:
1. Pre-processing
2. Processing (solution)
3. Post-processing

1) Pre-processing:

It consists of solid model generation and discretization that in to finite elements. Definition of properties of modal such as element type, material properties, various constants such as young's modulus, Poisson ratio etc., dimension of each element i.e., thickness, moment of inertia, area etc. Generation of element: two different methods are used in generating the elements.

1. Direct generation
2. From solid model

In direct generation method, the node is defined first and the elements are interconnected to obtain the final model.

In solid generation method solid model is generated and then, model is divided into finite elements. This conversion of solid model to finite elements is done through mesh generation. This method is more useful for complex models. In the present work solid generation method is used for making FEM models. Elements from solid model method can be subdivided into two categories.

• Free meshing.
• Mapped meshing.

Free meshing:

Free meshing allows more flexibility in defining mesh areas. Free mesh boundaries can be much more complicated than mapped mesh without subdividing in to multiple regions. The mesh will automatically be created by an algorithm that tries to minimize element distortion (deviation from a perfect square). Free mesh surfaces can easily have internal holes, where mapped mesh surfaces can't. Free meshing is controlled by two parameters assigned to each mesh surface or volume that affect the size of the elements generated. The first is the element length, which is the normal size of elements the program will attempt to generate. The second parameter controls mesh refinement at curves in the model by controlling how much deviation is allowed between straight element sides and curved boundaries. This parameter is expresses either as a percentage deviation or an absolute number.

Mapped meshing:

Mapped meshing requires the same number of elements on opposite sides of the mesh area, and requires that mesh area be bounded by three or four "edges". If you define a mapped mesh area with more than four curves, you must define which vertices are topological corners of the mesh. Mapped mesh boundaries with three corners will generate triangular elements in on "degenerate" corner. The number of elements per edge and biasing of elements of elements of element size towards the end or the center of edges control the mesh density. Another advantage of mapped meshing is we can produce dense mesh where we are interested and can produce coarse mesh where we are not interested even though it is of curved structure

2) Processing Solution:

The FE solver can be logically divided into three main parts, the pre-solver, the mathematical-engine & the post-solver. The pre-solver reads in the model created by the pre-processor and formulates the mathematical representation of the model. All parameters defined in the pre-processing stage are used to do this, so if one left something out, chances are the pre-solver will complain and cancel the call to the mathematical-engine. If the model is correct the solver proceeds to form the element-stiffness matrix for the problem & calls the mathematical-engine, which calculates the result (displacement, temperatures, pressures, etc.). The results are returned to the solver & the post-solver is used to calculate strains, stresses, heat fluxes, velocities, etc. for each node within the component or continuum.

3) Post-Processor:

Here the results of the analysis are read & interpreted. They can be presented in the form of a table, a contour plot, deformed shape of the component or the mode shapes and natural frequencies if frequency analysis is involved. Other results are available for fluids, thermal and electrical analysis types. Most post-processors provide an animation service, which produces an animation & brings your model to life. Contour plots are usually the most effective way of viewing results for structural type problems. Slices can be made through 3D models to facility the viewing of internal stress patterns. All post-processors now include the calculation of stress & strains in any of the x, y or z directions, or indeed in a direction at an angle to the coordinate axes. The principal stresses and strains may also be plotted, or if required the yield stresses and strains according to the main theories of failure (Von-Misses, St. Venant, Tresca etc.). Other information such as the strain energy, plastic strain and creep strain may be obtained for certain types of analyses. Many user- oriented general-purpose finite element packages such as MSC-NASTRAN, NISA ANSYS etc., are available in the market today. In the present context, the finite element analysis for the base frame was carried out by using the package, MSC NASTRAN (NASA TRANSLATION)

4.4Advantages and Disadvantages of FEA

Advantages:

1. This method can be efficiently applied to cater irregular geometry. It can take care of any type of boundary.
2. Material anisotropy and non-homogeneity can be catered without much difficulty.
3. Any type of loading can be handled. Although we have many approximate methods such as wide residual methods, Rayleigh-Ritz method, Gale kin's method, FEM is superior to all of them.

Disadvantages:

1. There are many types of problems where some other methods of analysis may prove efficient than the finite element method.
2. The cost involved in the solution of the problem for vibration and stability problem in many cases, the cost of analysis by the finite element method may be prohibitive.
3. In FEM, the admissible functions are valid over the simple domain and have to do with the boundary, however simple or complex it may be.

4.5 Limitations of FEA

1. Due to the requirement of large computer memory and time, computer programs based on the FEM can be run only in high-speed computers.

2. There are also certain categories of problems where other methods are more effective. E.g.: fluid problems having boundaries at infinity are better treated by the boundary element method.

3. In the FEM the size of the problem is relatively large. Many problems lead to round-off errors. Since the computer works with a limited number of digits, a desired degree of accuracy may not be expected or in some cases it gives totally erroneous results. This results in errors in the analysis results

5. THEORETICAL CALCULATION

FORMULA SYMBOLS:

- d=Wire diameter
- Do=Outer diameter
- D=Mean diameter
- E=Young's Modulus of Material
- G=Shear Modulus of Material
- L=Free Length
- K=Spring Rate
- Kc=Stress Correction factor

CALCULATION:

- **SPRING RATE:-**

K=$\frac{GD^4}{8D^3N}$ G=80,000MPA

K=$\frac{80000*(10.42)}{8*(87.95)^3*12}$ G=80,000N/mm^2

K=14.44

- **DEFLECTION**

δ=$\frac{8WD^3N}{GD^4}$

K=$\frac{W}{\delta}$

14.44=$\frac{1617}{\delta}$

$\delta = 111.98$mm

- **SHEAR STRESS**

$\tau = Ks\frac{8FD}{\pi d^3}$

Ks=$\frac{(2C+1)}{2C}$

Ks=$\frac{(2*8.44+1)}{(2*8.44)}$

Ks=10.59N/mm^2

- $\tau = 1.059\frac{8*1617*87.95}{\pi*10.423}$

$\tau = 338.93N/mm^2$

5.1 DESIGN CALCULATIONS FOR HELICAL SPRINGS FOR SHOCK ABSORBERS

Material: Steel Modulus of rigidity G = 80000 Pa,

Average shear stress =338.93 MPa

C = spring index = 8.44, Wahl‟s stress factor= 1.184

Mean diameter of a coil D=87.95mm

Diameter of wire d = 10.42

Total no of coils n1= 14

Outer diameter of spring coil D0 = D +d =93.16mm

No of active turns n=12

Weight of bike = 125kgs

Let weight of 1 person = 75Kgs

Weight of 2 persons = 75×2=150Kgs

Weight of bike + persons = 275Kgs

Rear suspension = 60%[1] Hence 60% of 275 = 165Kgs

Considering dynamic loads it will be double W = 330Kgs = 3234N

For single shock absorber weight = w/2= 1617N = W

Compression of spring (δ) = 111.98mm

Solid length, Ls=n‟×d=16×10.3=165.6mm

Free length of spring, Lf = solid length+maximum compression + clearance between adjustable coils = 165.6 + 111.98 + 0.15 × 111.98 = 295.3mm

Spring rate, K =14.44

Pitch of coil, P = 22.15

6. ANALYSIS USING ANSYS

6.1 INTRODUCTION TO ANSYS

ANSYS is an engineering simulation software provider founded by software engineer John Swanson. It develops general-purpose finite element analysis and computational fluid dynamics software. While ANSYS has developed a range of computer-aided engineering (CAE) products, it is perhaps best known for its ANSYS Mechanical and ANSYS Multi-physics products.

ANSYS Mechanical and ANSYS Multi--physics software are non exportable analysis tools incorporating pre-processing (geometry creation, meshing), solver and post-processing modules in a graphical user interface. These are general-purpose finite element modeling packages for numerically solving mechanical problems, including static/dynamic structural analysis (both linear and non-linear), heat transfer and fluid problems, as well as acoustic and electro-magnetic problems.

ANSYS Mechanical technology incorporates both structural and material non-linearity's. ANSYS Multi-physics software includes solvers for thermal, structural, CFD, electro-magnetic, and acoustics and can sometimes couple these separate physics together in order to address multidisciplinary applications. ANSYS software can also be used in civil engineering, electrical engineering, physics and chemistry.

Every analysis involves four main steps:

A. Preliminary Decisions

1. What type of analysis: Static, modal, etc.?

2. What to model: Part or Assembly?

3. Which elements: Surface or Solid Bodies?

B. Preprocessing

1. Attach the model geometry

2. Define and assign material properties to parts

3. Mesh the geometry

4. Apply loads and supports

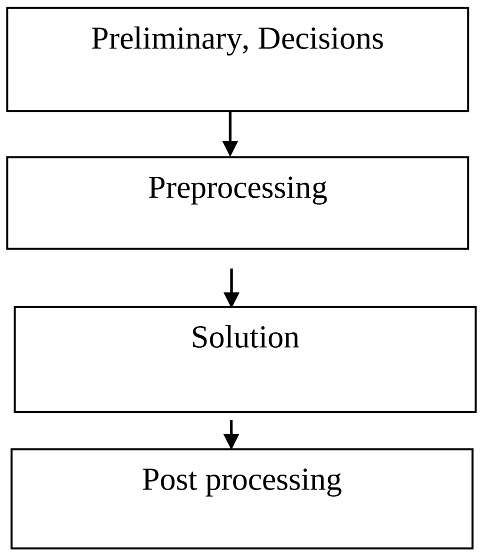
5. Request results

C. Solve the Model

D.Post processing

1. Review results

2. Check the validity of the solution



6.2 ANSYS WORK BENCH ANALYSIS

The ANSYS Workbench platform is the framework upon which the industry's broadest and deepest suite of advanced engineering simulation technology is built. An innovative project schematic view ties together the entire simulation process, guiding the user through even complex metaphysics analyses with drag-and-drop simplicity. With bi-directional CAD connectivity, powerful highly-automated meshing, a project-level update mechanism, pervasive parameter management and integrated optimization tools, the ANSYS Workbench platform delivers unprecedented productivity, enabling Simulation Driven Product Development. The Workbench environment allows you to solve much more complex analyses, including:

1. Multi-part assemblies

2.3-D solid elements, shell elements, and shell-solid assemblies

3. Nonlinear contact with or without friction

4. Small-displacement and large-displacement static analyses

5. Modal, harmonic, and Eigen value buckling analyses

6. Steady-state thermal analysis, including temperature-dependent material properties and thermal contact.

6.3 ANALYSIS SYSTEMS

One way to start an analysis in ANSYS Workbench is to select an analysis system from the Toolbox. When you select one of these analysis types, the corresponding system will appear in the Project Schematic, with all the necessary components of that type of analysis. Some analysis types offer different solvers, noted in parentheses. The features available can differ from one solver to another.

A static structural analysis determines the displacements, stresses, strains, and forces in structures or components caused by loads that do not induce significant inertia and damping effects. Steady loading and response conditions are assumed; that is, the loads and the structure's response are assumed to vary slowly with respect to time. You will configure your static structural analysis in the Mechanical application, which uses the ANSYS or SAMCEF solver, depending on which system you selected, to compute the solution.

1. Add a static structural analysis template by dragging the template from the Toolbox into the Project Schematic or by double-clicking the template in the Toolbox.

2. Load the geometry by right-clicking on the Geometry cell and choosing Import Geometry.
3. View the geometry by right-clicking on the Model cell and choosing Edit or double-clicking the Model cell. Alternatively, you can right click the Setup cell and select Edit. This step will launch the Mechanical application.
4. In the Mechanical application window, complete your static structural analysis using the Mechanical application's tools and features. See Static Structural Analysis in the Mechanical application help for more information on conducting a structural analysis in the Mechanical application.

Imported model form CATIA

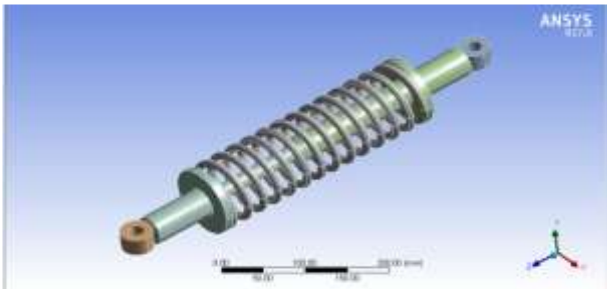


Fig:6.1.1 Imported model

Shock absorber after meshing

Fig: 6.1.2Meshed model

Model (A4) > Geometry > Parts

Object Name	PartBody	PartBody	PartBody	PartBody	PartBody
State	Meshed				
Graphics Properties					
Visible	Yes				
Transparency	1				
Definition					
Suppressed	No				
Stiffness Behavior	Flexible				
Coordinate System	Default Coordinate System				
Reference Temperature	By Environment				
Behavior	None				

International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 06 Issue: 06 | June 2019 www.irjet.net p-ISSN: 2395-0072

Material					
Assignment	Structural Steel				
Nonlinear Effects	Yes				
Thermal Strain Effects	Yes				
Bounding Box					
Length X	93.16 mm	40. mm	93.16 mm	40. mm	93.16 mm
Length Y	93.16 mm	25. mm	93.16 mm	25. mm	93.16 mm
Length Z	298.4 mm	40. mm	250. mm	40. mm	253. mm
Properties					
Volume	2.8735e+005 mm³	23562 mm³	2.1844e+005 mm³	23562 mm³	2.2423e+005 mm³
Mass	2.2557 kg	0.18497 kg	1.7147 kg	0.18497 kg	1.7602 kg
Centroid X	8.20989 mm	-1.8098e-006 mm	3.0613e-017 mm	-1.8121e-006 mm	-2.575e-017 mm
Centroid Y	8.13031 mm	-2.1973e-003 mm	-1.752e-015 mm	2.1972e-003 mm	6.6863e-016 mm
Centroid Z	-154.2 mm	-413.4 mm	-309.67 mm	104.93 mm	-2.6845 mm
Moment of Inertia Ip1	18275 kg·mm²	32.456 kg·mm²	6064.1 kg·mm²	32.452 kg·mm²	7115.1 kg·mm²
Moment of Inertia Ip2	18301 kg·mm²	45.751 kg·mm²	6064.5 kg·mm²	45.747 kg·mm²	7115.5 kg·mm²
Moment of Inertia Ip3	3845.3 kg·mm²	32.463 kg·mm²	833.52 kg·mm²	32.463 kg·mm²	752.76 kg·mm²
Statistics					
Nodes	29627	1870	44452	1870	7928
Elements	11770	308	8952	308	4208
Mesh Metric	None				

6.4 Analysis Structural Analysis using Spring Steel as spring material

Material: Spring Steel

Material Properties:

Young"s Modulus (EX): 202000N/mm2

Poisson"s Ratio (PRXY): 0.292

Density: 0.000007820kg/mm3

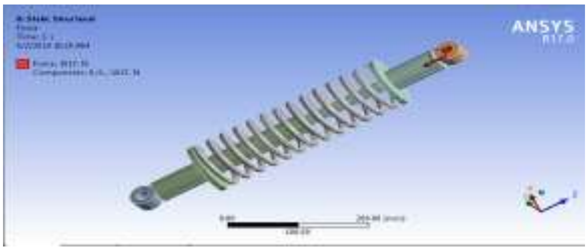


Fig: 6.4.1 Appling load

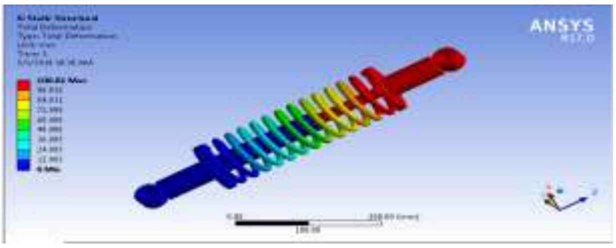


Fig: 6.4.2 Total deformation

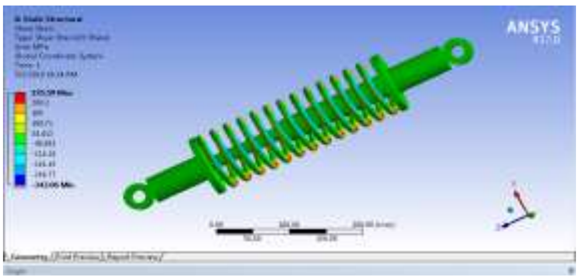


Fig: 6.4.3 Shear stress

6.5 Modal Analysis using Spring Steel as spring material

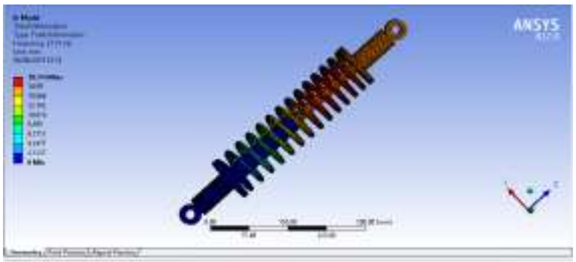


Fig: 6.5.1 first mode shape

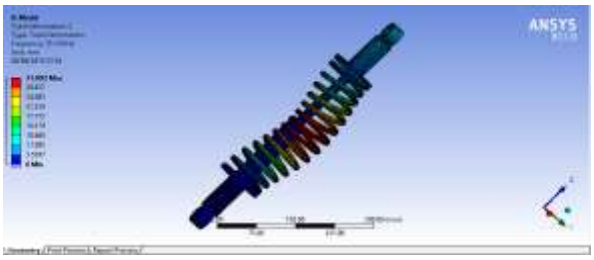


Fig:6.5.2 second mode shape

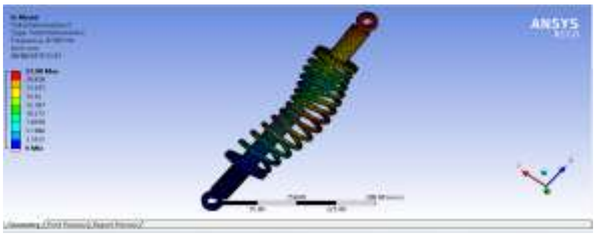


Fig: 6.5.3 Third mode shape

6.6 Structural Analysis using Phosphor Bronze as spring material

Material: Phosphor Bronze

Material Properties:

Young's Modulus (EX): 103000N/mm2

Poisson's Ratio (PRXY): 0.34

Density: 0.000008160kg/mm3

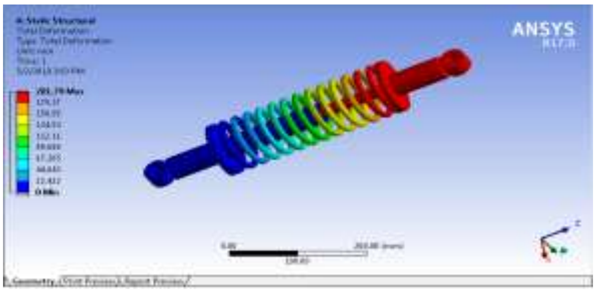


Fig:6.5.1 Total deformation

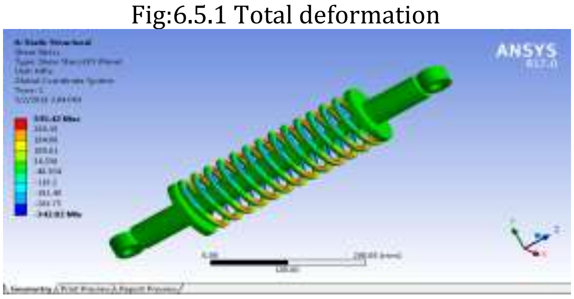


Fig: 6.5.2 Shear stress

6.7 Modal Analysis using Phosphor Bronze as spring material

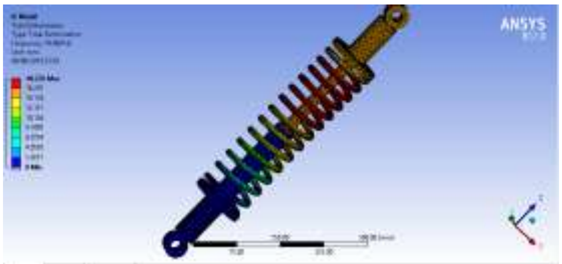


Fig: 6.7.1 first mode shape

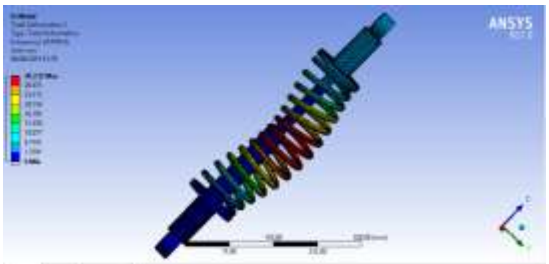


Fig:6.7.2 second mode shape

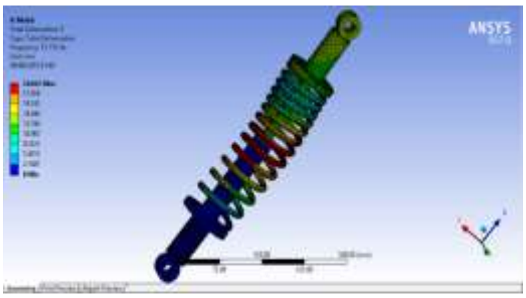


Fig: 6.7.3 Third mode shape

Results

	Analytical	Spring Steel	Phosphor Bronze
Total Deformation (mm)	**111.9**	**108.01**	**201.79**
Shear Stress (N\mm2)	**338.95**	**335.59**	**335.42**
Natural Frequency		**27.31**	**19.08**
		33.39	**23.33**
		35.15	**24.47**
		43.55	**30.50**
		47.65	**33.11**
		48.48	**33.84**

7. CONCLUSIONS

1. In this project we have designed a shock absorber. We have modeled the shock absorber by using 3D parametric software CATIA
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 varying spring material Spring Steel and Phosphor Bronze.
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 deformation value is less for Spring Steel than Phosphor Bronze. So stiffness is more for Spring Steel.
5. By comparing the results for both materials, the Natural Frequency is more for Spring Steel than Phosphor Bronze.
6. So we can conclude that as per our analysis using material spring steel for spring is best.

REFERENCES

1. International Journal of Engineering Research and Development e-ISSN: 2278-067X, p-ISSN: 2278-800X, www.ijerd.com Volume 10, Issue 2 (February 2014), PP.22-2

2. "Strength of Materials", S.S. Rattan, Tata McGraw Hill, 2009

3. "Strength of Materials", S.S. Bhavikatti, Vikas publications House -1 Pvt. Ltd., 2nd Ed., 2006.

4. Design Data Hand Book, K. Lingaiah, McGraw Hill, 2nd Ed.

5. Design Data Hand Book, H.G. Patil, I. K. International Publisher, 2010.

6. Design of Machine Elements, V.B. Bhandari, Tata McGraw Hill Publishing Company Ltd., New Delhi, 2nd Edition 2007.

7. Finite Element Method, J.N. Reddy, McGraw -Hill International Edition.

8. "Finite Element Methods for Engineers" U.S. Dixit, Cengage Learning, 2009

9. VarunBrijpuria, K. K. Jain, "Analysis of Closed Coil Helical Spring Subjected to Heavy Duty", IJEA, Volume 1, Issue 3, pp. 52-53

BIOGRAPHIES

	**KARAN CHOUGULE** Pursuing the bachelor of engineering degree and presently in the final year of B.E.
	**ADITYA R. PARTE** Pursuing the bachelor of engineering degree and presently in the final year of B.E.
	**MILIND DEEPAK HAVAL** Pursuing the bachelor of engineering degree and presently in the final year of B.E.