Design and Analysis of Steering Column By Vibration / Structural Mode

M.Soundar Rajan [1], N.Ramesh [2], M. Naveen Kumar [3], R.Murali Manokar[4]
PG Scholar, M.E-Engineering Design[1],
Assistant Professor[2, 3] H.O.D[4], Department of Mechanical Engineering[2, 3, 4]
Maharaja College of Engineering, Avinashi, Tamilnadu, India

Abstract-Finite Element Analysis and parametric study of steering column for new generation vehicles to reduce or nullify the steering unit. The analysis is carried out with respect to vibration. Stresses developed in an object design requirements at the joints , deformation in body due to vibrations, continuos twisting and loading these are related to steering rod. Harmonic analysis will be giving us natural frequency of body that compared with harmonic frequency. Aim of project is to perform design optimization of steering column to nullify its functions-ability issues related with stressess, deformation, vibrations also minimize cost by saving material to compare original model. The software Ansys is used for FE Analysis and method of harmonic is used structrual is used for design.

I.INTRODUCTION

Recent trends in automobile development activities for reduction of lead-time and cost have led to a current situation where CAE(computer aided engineering) techniques are fully used to skip conventional development steps for making and checking costly prototypes.

Many automakers now use a computer simulation instead of preparing costly prototypes to analyze the strength and the collision resistance of a vehicle body.

Recent use of computer simulation has been further expanded for a dummy model or vehicle interior accessories which are used for analyzing what and how much impact may occur to passengers.

Some automakers are trying to use a so-called digital prototyping, where all design steps for a prototype are performed through computing operation.

With such a trend for digital developments by automakers, vehicle component makers including KOYO, who are responsible for the development and mass-production of steering column products.(e.g.,a safety steering wheel and an electric power steerings), must keep up with the trend by further improving their CAE analysis techniques for pre-production steps to reduce the number of redundant steps from prototyping to experiment evaluation and to provide drawings with higher accuracy.

The current CAE analysis by Koyo includes four major functions of a vehicle (i.e., strength, noise/Vibration, vehicle motion, and collision),among with collision of a steering column assembly (hereinafter referred to as assembly) will be focused in this paper. Specially, this paper will use a collision model of steering column assembly to examine the consistency between the result of the CAE analysis model and the result of actual collision test of an actual assembly.

1.1 NEED FOR THE STUDY

Recent trends in automobile development activities for reduction of lead time and cost have lead to a current situation where CAE (Computer Aided Engineering) techniques are fully used to skip conventional development steps for making and checking costly prototypes . The Steering System used predominantly in passenger cars today is the Rack and pinion type . A virtual prototyping approach by using a one degree haptic system, makes it possible for the customer to test the virtual prototype of the steering unit in a direct and natural way, in early design phase . An comparison of CAE analysis results and Testing results for the Steering Column Assembly and characteristics of the steering system can be evaluated properly using HIL

A number of Analysis has been performed on virtual prototype of Steering column Assembly. But Static Rack Bending Analysis of Steering column Assembly has not been studied yet. Steering Rack is designed to sustain bending loads during vehicle running. The loads come from tire side and produce the bending loads on Steering Rack. Steering Rack Static Bending Analysis will be focused in this paper.

This project is an attempt to design a Rack and Pinion with specifications minimizing swing torque, in ADAMS (Automatic Dynamic Analysis of mechanical Systems).

This model helped to identify critical parameter which affects steering column. A number of Analysis has been performed on virtual prototype of Steering case Assembly.

The loads come from tire side and produce the bending loads on Steering Rack. Steering Rack Static Bending Analysis will be focused in this paper.

Specifically, this paper will use a CAD / CATIA 3D model of Steering Column /Case Assembly to examine the consistency between the results of the CAE Analysis model and the theoretical calculation of Steering Case Bending and Deflection. The objective of this work is to carry out Computer Aided design and Analysis of Steering Rack. The CAD modeling is done in CATIA V21 and Finite Element Analysis is done in ANSYSR 15.0 Animation.

1.3. Objective of Study

The loads come from tire side and produce the bending loads on Steering Case through Steering Rack Static Bending Analysis will be focused in this project.

- Identify and study using software tools (for simulation/ analysis), the nature and characteristics of stresses acting on the component.
- Evaluate the influence of the loads/ mass/geometry/ boundary conditions over the nature and extend of stresses.
- Review the existing design and consider improvement for negating the harmful influences of undue stresses (Torsion or Shear).

Study and analysis of a modified steering system according to the constraints provided by team.

2. LITERARATURE REVIEW:

IJIRST –International Journal for Innovative Research in Science &

Technology| Volume 2 | Issue 05 | October 2015. " A Literature Review on Collapsible Steering Column". Imran J. Shaikh. Energy absorbing steering column (Collapsible steering column) is a kind of steering column which minimizes the injury of thedriver during a car accident by collapse or breaking particular part of system. Up to now, Collapsible Steering Column for lowbudget passenger car had no way to describe these „Collapse‟ or „Slip‟ by the Axial and Lateral Forces from driver. In this paper,I have created a collapsible steering column from rigid steering column using a Detailed FE model which can describe suchcollapse behavior .

FIGURE-2.1-Steering column collapse

International Journal of Advances in Engineering Sciences Vol.4, Issue 3, April, 2014 12 Print-ISSN: 2231-2013 e-ISSN: 2231-0347 in " Design and Stress Analysis of Steering Rack Using CAE Tool"

Nitalikar et al., International Journal of Advanced Engineering Research and Studies E-ISSN2249–8974Int. J. Adv. Engg. Res. Studies/III/I/Oct.-Dec.,2013/112-114 Review Article "STRUCTURAL ANALYSIS FOR A CARDON JOINT IN STEERING COLUMN ASSEMBLY THROUGH FEA TECHNIQUES" by Ashish Bharatrao Nitalikar, 2R.D.Kulkarni, 3Swapnil S. Kulkarni.

Friction due to rubbing between the spider and the yoke bores is minimized by incorporating needle-roller bearings between the hardened spider journals and hardened bearing caps pressed into the yoke bores.

3. METHODOLOGY OF DESIGN:

3.1. STEPS FOR THE PROPOSED WORK

- Creation of Geometry for Steering Column.

- Importing the geometry for meshing.
- Assigning the nature of loads and the values for loading.
- Solving for the meshed model to identify stressed areas.
- Viewing the results.
- Modifying the geometry/ mass/ boundary conditions
- Solving the meshed model again (iteration/s)
- Comparison / Interpretation of the results
- Recommendations.

4. MODEL ANALYSIS OF STEERING COLUMN

The analytical/ computational approach offers results through simulation/ analyses for the case study predefined for the solver. The technique would deploy any of the following software tools: Patran/Hyper Mesh/ Nast ran, ANSYS, Abaqus, RadioSS orany compatible CAE software Benefits of using CAE software - The CAE software usually has an intuitive graphical user interface with direct access to **CAD geometry**, advanced tools for meshing and integration with other compatible software for solving. It is optimized for large scale systems, assemblies, dynamics and NVH simulations. Typically, the CAE interface design to handle structural problems as the case study concerned here Is adept to linear static analysis with a post processing interface to view results. The Geomentric Dimensions should be carried out by CAD 2016 versions of software. For modeling of the component, CATIA V5 R21 Software is used. Preprocessing work like meshing and analysis work is carried out in ANSYS R15.0 software. Using FEA analysis,

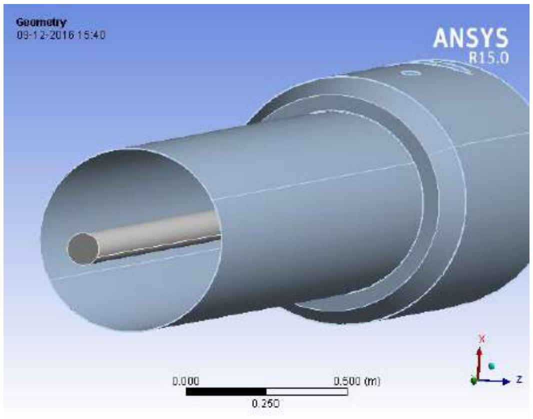


FIGURE4.1-Model (A4) > Static Structural (A5) > Remote Displacement > Image

we can identify the nature and characteristics of stress acting on the steering case and rod also evaluate the influence of the loads/mass/geomentry/boundary conditions over the yoke.

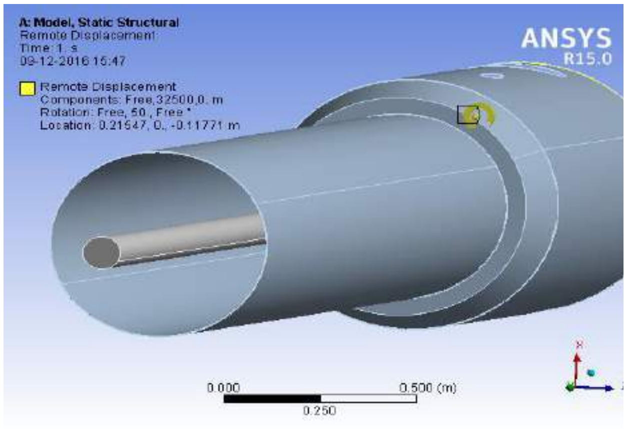


FIGURE.4.2Model (A4) Static Structural (A5)

Figure shows the 3D model geomentry of Steering Case, rod with assembly.

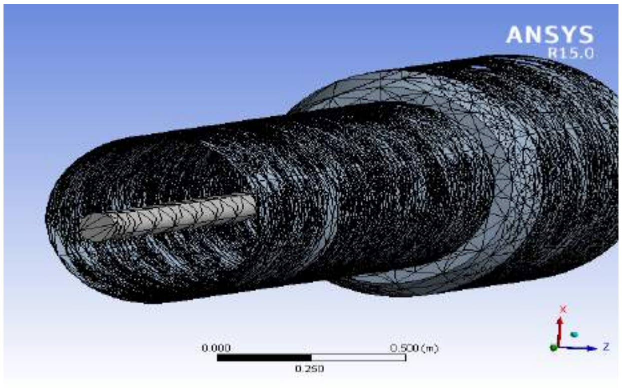
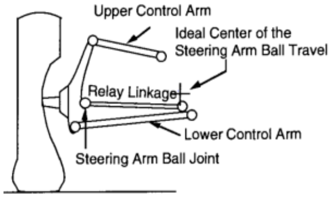


FIGURE-4.3-Modal-Meshing of steering

5 DESIGN OF STEER COLUMN

5.1. Basic description of Steering Components:

Friction materials used are Cork and Copper Powder Metal. Material used for inner disc is steel and outer disc is bronze.



FIGUR5.1 Steering Arm knucle joint

Due to caster angle.

- $F_{zr} \sin \gamma$ = Force component in the direction parallel to caster angle seen in side view.
- $d \cos \delta$ = moment arm forward to force.

- Moment due to both wheel is opposite in direction. This force balances the left right wheel load. This may result into wheel toe-in and asymmetry of tie rod resulting in its push or pull.
- Axle rolls with steered.
- Sensitive to left right load imbalance.
- Torque gradient depends upon wheel offset at the ground castor angle, left right load difference in cornering, front and rear suspension roll stiffness, Suspension roll centre height, centre of gravity height, lateral acceleration level.

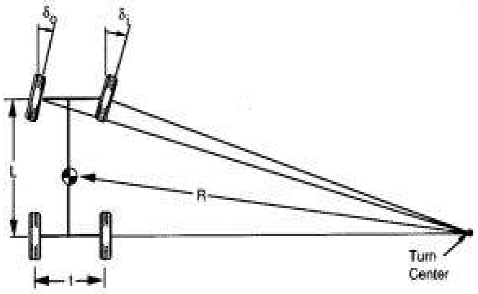


FIGURE-5.2 Steering Arm turn

5.1.1 Design Theory of Steering Column (Hub/Shaft):

Steering system forces and moment:

Three types of forces are normally seen in vehicle tire:

1. **force (aligning torque) z-direction.**
2. **Tractive force (Rolling resistance moment) y-direction.**
3. **Lateral force (overturning moment) x-direction.**

The reaction in the steering system is due to the moment about steering axis , which must be reduce to control the wheel steer angle.

1. **Vertical force**
2. **It has inclusion of two forces.**
3. **Due to lateral inclination angle (left side of equation).**

Caster angle (right side of equation):

$$MV = -(FZ1+FZr)\,d\sin$$

$$My = -(Fzl + Fzr)\, d \sin \lambda \sin \delta + (Fzl - Fzr)\, d \sin \gamma \cos \delta$$

My = Total moment from left and right wheels.

F_{zl}, F_{zr} = Vertical load on left and right wheel

d = lateral offset on ground or scrub radius.

λ = lateral inclination angle or king pin angle.

δ = Steer angle

γ = caster angle

Due to lateral inclination angle:

- $F_{zr} \sin\lambda$ = Sine angle of force component acting laterally and parallel to king pin axis.
- d sin δ = moment arm of above force
- The moment is zero when no steering. When steering, because of this force vehicle tends to lift, Increasing the steering effort and also self-centring force.
- Axles lift when steered.
- Unaffected by right left load differences.
- Torque gradient depends upon wheel offset at ground, Inclination angle, and axle load.

5.1.2. Calculation of steering shaft:

Steering Hub & Steering Rod

Elliptical Section:

a = Major Axis

b = Minor Axis

l = Length of Shaft

T = Applied Torque

C = Rigidity of Modulus

Maximum Shear Stress (t):-

$T = 16T/pi*a*b^2$

Maximum shear stress occurs at the ends of the minor axis:

Angle of Twist (q):

Theta = 16*l*T*/pi*a*b*c [1/a2+1/b2]

Torsional Stiffness (k):-

$K = C*pi*a^3b^3 / 16(a^2+b^2)$

Equilateral Triangles:-

A = side of triangle

L = Length of shaft

T = Applied torque

C = Rigidity modulus

Maximum shear stress occurs at the centre of each side while the shear stress of each corner is equal to zero.

Angle of Twist (Q):

$$Q = 80/a^4 v3 * T*l/C$$

Torsional stiffness (K):

$$K = v3/80*a^4 C$$

Calculation:

Applied Torque,

$$\mathbf{T = 2.5KN/m.}$$

Maximum Permissible shear stress:-

$$\mathbf{T = 80MN/m^{2.}}$$

Major Axis (a) and Minor Axis (b):-

$$W.K.T.\ T = 16*T/pi*a*b^2$$

$$80*10^6 = 16*2.5*10^3 / pi*1.5b*b^2 \Rightarrow b^3 = 1.061*10^{-4}$$

b = 0.0473m or 47.3 mm.

a=1.5b =1.5*47.3 mm

a=70.95 mm.

Angular twist per metre length, q/l:

$$\text{Angular Twist} = Q = 16T/(pi*a*b*C[1/a^2 + 1/b^2]$$

$$\text{Angular Twist} = Q = 16*2.5*10^3 /pi*70.95*10^{-3}*47.3*10^{-3}*80*10^9[1/(70.95*10^{-3}m) + 1/(47.3*10^{-3})^2].$$

$$\mathbf{= 40.0306\ rad\ (1.75\ deg).}$$

THEORITICAL BENDING STRESS AND DEFLECTION:

The vertical Load causes the bending stress and if the Load is higher than critical load then it will lead to breakage.

Considering the Vehicle Front Axle Weight of 6 kN.

The assembly is considered as Cantilever beam.

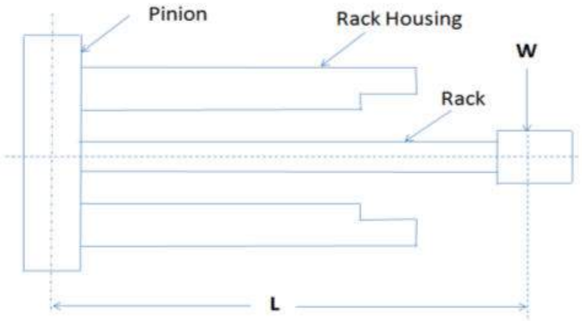


FIGURE-5.3-Rack Housing – Vertical load

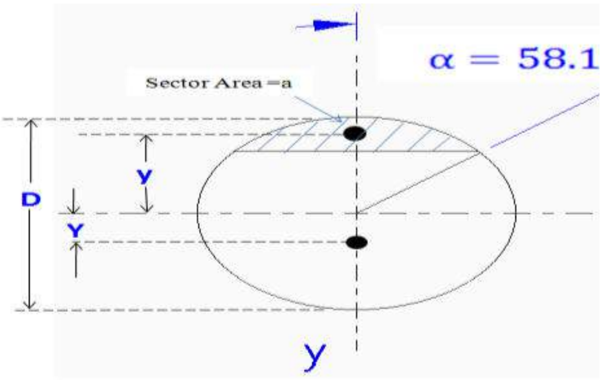


FIGURE-5.4-Minimum cross section steering column

Deflection Equation at Point Load: (1)

$$\delta = \frac{W.L^3}{3.E.I}$$

Steering Case Breakage Stress Equation: (2)

$$\sigma = \frac{W.L}{Z}$$

Using Equations 1 and 2 above and putting Values from Table No. 1, the results are as below:
Rack Deflection, **δ= 5.8 mm**
Rack Bending Stress**, σ = 420 Mpa**

Maximum Principal Stress & Equivalemt Stresses are Analysed by Analytical Method - After the construction of the geometry (3D model) and preprocessing (meshing), a static stress analysis is planned by using the mechanical properties of the material (Elasticity modulus = 205 GPa, Poisson's ratio = 0.29 of the typical Carbon steel material variant) as input data for preparing the model for analysis. The solid model followed by finite element mesh followed by static analysis for assessing the distribution of von Misses stress values should offer good inputs, in turn, to review the design in the light of these results.

6. STRUCTURAL ANALYSIS OF STEERING COLUMN

The analytical/ computational approach offers results through simulation/ analyses for the case study predefined for the solver. The technique would deploy any of the following software tools: Patran/HyperMesh/ Nastran, ANSYS, Abaqus, RadioSS orany compatible CAE softwareBenefits of using CAE software - The CAE software usually has an intuitive graphical user interface with direct access to **CAD geometry**, advanced tools for meshing and integration with other compatible software for solving. It is optimized for large scale systems, assemblies, dynamics and NVH simulations. Typically, the CAE interface design to handle structural problems as the case study concerned here Is adept to linear static analysis with a postprocessinginterface to view results.

6.1. Analysis Project Report :

Project

First Saved	Friday, October 21, 2016
Last Saved	Thursday, December 08, 2016
Product Version	15.0.7 Release
Save Project Before Solution	No
Save Project After Solution	yes

Contents

- Units
- Model (A4)
    - Geometry
        - Parts
    - Coordinate Systems
    - Connections
    - Mesh
    - Static Structural (A5)
        - Analysis Settings
        - Loads
        - Solution (A6)
            - Solution Information
            - Results
    - Chart
    - Chart 2
- Material Data
    - Structural Steel

Units

TABLE 6.1

Unit System	Metric (m, kg, N, s, V, A) Degrees rad/s Celsius
Angle	Degrees
Rotational Velocity	rad/s
Temperature	Celsius

Model (A4)

Geometry

TABLE 6.2
Model (A4) > Geometry

Object Name	*Geometry*
State	Fully Defined
Definition	
Source	C:\Users\student\Desktop\Thu oct\Product B.igs
Type	Iges
Length Unit	Meters
Element Control	Program Controlled
Display Style	Body Color
Bounding Box	
Length X	0.7765 m
Length Y	6.4901 m
Length Z	0.7765 m
Properties	
Volume	7.6169e-002 m^3
Mass	597.92 kg
Scale Factor Value	1.
Statistics	
Bodies	2
Active Bodies	2
Nodes	72928
Elements	36411
Mesh Metric	None
Basic Geometry Options	
Solid Bodies	Yes
Surface Bodies	Yes
Line Bodies	No
Parameters	Yes
Parameter Key	DS
Attributes	No
Named Selections	No
Material Properties	No
Advanced Geometry Options	
Use Associati	Yes

vity	
Coordinate Systems	No
Reader Mode Saves Updated File	No
Use Instances	Yes
Smart CAD Update	No
Compare Parts On Update	No
Attach File Via Temp File	Yes
Temporary Directory	C:\Users\student\AppData\Local\Temp
Analysis Type	3-D
Mixed Import Resolution	None
Decompo	Yes

se Disjoint Geometry		
Enclosure and Symmetry Processing	Yes	
Object Name	*Part 1*	*Part 2*
State	Meshed	
Graphics Properties		
Visible	Yes	
Transparency	1	
Definition		
Suppressed	No	
Stiffness Behavior	Flexible	
Coordinate System	Default Coordinate System	
Reference Temperature	By Environment	
Material		

Assignment	Structural Steel	
Nonlinear Effects	Yes	
Thermal Strain Effects	Yes	
Bounding Box		
Length X	0.1 m	0.7765 m
Length Y	6.4901 m	5.3678 m
Length Z	0.1 m	0.7765 m
Properties		
Volume	4.9043e-002 m³	2.7126e-002 m³
Mass	384.99 kg	212.94 kg
Centroid X	-1.7879e-018 m	-1.7824e-003 m
Centroid Y	3.0857 m	2.499 m
Centroid Z	-9.7929e-019 m	-1.4888e-005 m
Moment of Inertia Ip1	1242.1 kg·m²	529.82 kg·m²
Moment of Inertia Ip2	0.47066 kg·m²	22.705 kg·m²
Moment of Inertia Ip3	1242.1 kg·m²	529.62 kg·m²
Statistics		
Nodes	982	71946
Elements	359	36052
Mesh Metric	None	

Coordinate Systems

TABLE 6.4 Model (A4) > Coordinate Systems > Coordinate System

Object Name	*Global Coordinate System*
State	Fully Defined
Definition	
Type	Cartesian
Coordinate System ID	0.
Origin	
Origin X	0. m
Origin Y	0. m
Origin Z	0. m
Directional Vectors	

X Axis Data	[1. 0. 0.]
Y Axis Data	[0. 1. 0.]
Z Axis Data	[0. 0. 1.]

Connections

TABLE 6.5
Model (A4) > Connections

Object Name	*Connections*
State	Fully Defined
Auto Detection	
Generate Automatic Connection On Refresh	Yes
Transparency	
Enabled	Yes

Mesh

TABLE 6.6
Model (A4) > Mesh

Object Name	*Mesh*
State	Solved
Defaults	
Physics Preference	Mechanical
Relevance	0
Sizing	
Use Advanced Size Function	Off
Relevance Center	Coarse
Element Size	Default
Initial Size Seed	Active Assembly
Smoothing	Medium
Transition	Fast
Span Angle Center	Coarse
Minimum Edge Length	2.5e-003 m
Inflation	
Use Automatic Inflation	None
Inflation Option	Smooth Transition
Transition Ratio	0.272
Maximum Layers	5
Growth Rate	1.2
Inflation Algorithm	Pre
View Advanced Options	No
Patch Conforming Options	
Triangle Surface Mesher	Program Controlled
Patch Independent Options	

Topology Checking	Yes
Advanced	
Number of CPUs for Parallel Part Meshing	Program Controlled
Shape Checking	Standard Mechanical
Element Midside Nodes	Program Controlled
Straight Sided Elements	No
Number of Retries	Default (4)
Extra Retries For Assembly	Yes
Rigid Body Behavior	Dimensionally Reduced
Mesh Morphing	Disabled
Defeaturing	
Pinch Tolerance	Please Define
Generate Pinch on Refresh	No
Automatic Mesh Based Defeaturing	On
Defeaturing Tolerance	Default
Statistics	
Nodes	72928
Elements	36411
Mesh Metric	None

Static Structural (A5)

TABLE 6.7
Model (A4) > Analysis

Object Name	*Static Structural (A5)*
State	Solved
Definition	
Physics Type	Structural
Analysis Type	Static Structural
Solver Target	Mechanical APDL
Options	
Environment Temperature	22. °C
Generate Input Only	No

TABLE 6.8
Model (A4) > Static Structural (A5) > Analysis Settings

Object Name	*Analysis Settings*
State	Fully Defined
Step Controls	
Number Of	1.

Steps	
Current Step Number	1.
Step End Time	1. s
Auto Time Stepping	On
Define By	Substeps
Initial Substeps	400.
Minimum Substeps	20.
Maximum Substeps	2000.
Solver Controls	
Solver Type	Program Controlled
Weak Springs	Program Controlled
Large Deflection	Off
Inertia Relief	Off
Restart Controls	
Generate Restart Points	Program Controlled
Retain Files After Full Solve	No
Nonlinear Controls	
Newton-Raphson Option	Program Controlled
Force Convergence	Program Controlled
Moment Convergence	Program Controlled
Displacement Convergence	Program Controlled
Rotation Convergence	Program Controlled
Line Search	Program Controlled
Stabilization	Off
Output Controls	
Stress	Yes
Strain	Yes
Nodal	No

Forces	
Contact Miscellaneous	No
General Miscellaneous	No
Store Results At	All Time Points
Analysis Data Management	
Solver Files Directory	C:\Users\student\Desktop\stering hub_files\dp0\SYS\MECH\
Future Analysis	None
Scratch Solver Files Directory	
Save MAPDL db	No
Delete Unneeded Files	Yes
Nonlinear Solution	Yes
Solver Units	Active System
Solver Unit System	mks

TABLE 6.9
Model (A4) > Static Structural (A5) > Loads

Object Name	*Remote Displacement*	*Remote Displacement 2*	*Compression Only Support*	*Compression Only Support 2*	*Fixed Support*	*Fixed Support 2*	*Fixed Support 3*
State	Fully Defined						
Scope							
Scoping Method	Geometry Selection						
Geometry	1 Edge		1 Face			1 Edge	1 Face
Coordinate System	Global Coordinate System						
X Coordinate	0.21547 m	-0.21687 m					

Y Coordinate	0. m			
Z Coordinate	-0.11771 m	0.11847 m		
Location	Defined			
Definition				
Type	Remote Displacement		Compression Only Support	Fixed Support
X Component	Free			
Y Component	32500 m (ramped)			
Z Component	0. m (ramped)	Free		
Rotation X	Free			
Rotation Y	50. ° (ramped)			
Rotation Z	Free			
Suppressed	No			
Behavior	Deformable			
Rotation X		Free		
Rotation Y		50. ° (ramped)		
Rotation Z		Free		
Advanced				
Pinball Region	All			
Normal Stiffness			Program Controlled	

Update Stiffness		Never	

FIGURE 1
Model (A4) > Static Structural (A5) > Remote Displacement

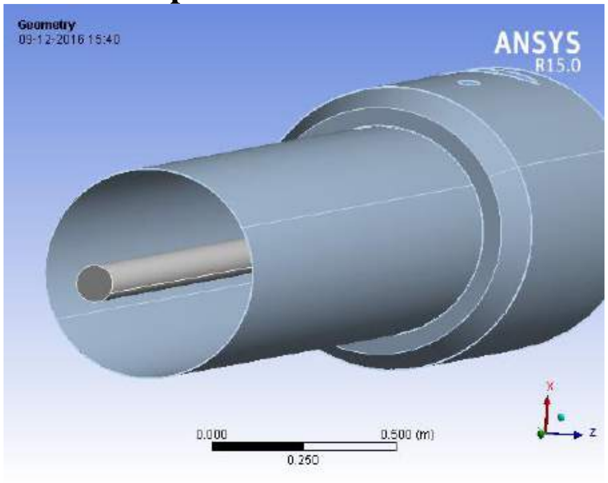


FIGURE 6.2
Model (A4) > Static Structural (A5) > Remote Displacement > Image

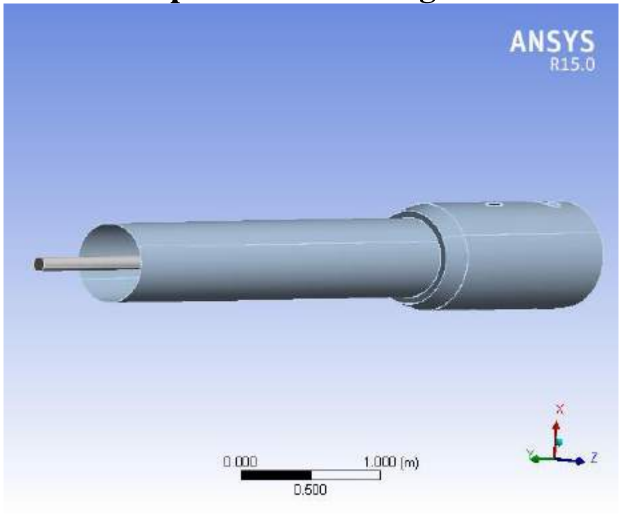


FIGURE-6.3-Model (A4) > Static Structural (A5) > Remote Displacement 2

Solution (A6)

TABLE 6. 10
Model (A4) > Static Structural (A5) > Solution

Object Name	*Solution (A6)*
State	Solved
Adaptive Mesh Refinement	
Max Refinement Loops	1.
Refinement Depth	2.
Information	
Status	Done

TABLE 6.11
Model (A4) > Static Structural (A5) > Solution (A6) > Solution Information

Object Name	*Solution Information*
State	Solved
Solution Information	
Solution Output	Solver Output
Newton-Raphson Residuals	0
Update Interval	2.5 s
Display Points	All
FE Connection Visibility	
Activate Visibility	Yes

Display	All FE Connectors
Draw Connections Attached To	All Nodes
Line Color	Connection Type
Visible on Results	No
Line Thickness	Single
Display Type	Lines

TABLE 6.12
Model (A4) > Static Structural (A5) > Solution (A6) > Results

Object Name	*Equivalent Stress*	*Maximum Shear Stress*
State	Solved	
Scope		
Scoping Method	Geometry Selection	
Geometry	All Bodies	
Definition		
Type	Equivalent (von-Mises) Stress	Maximum Shear Stress
By	Time	
Display Time	Last	0.32564 s
Calculate Time History	Yes	
Identifier		
Suppressed	No	
Integration Point Results		
Display Option	Averaged	
Average Across Bodies	No	
Results		
Minimum	2.3314e-009 Pa	6.6861e-010 Pa
Maximum	5.7639e+015 Pa	1.076e+015 Pa
Minimum Occurs On	Part 1	
Maximum Occurs On	Part 2	
Minimum Value Over Time		
Minimum	1.9191e-017 Pa	1.108e-017 Pa
Maximum	7.495e-009 Pa	4.214e-009 Pa
Maximum Value Over Time		

Minimum	1.441e+013 Pa	8.2603e+012 Pa
Maximum	5.7639e+015 Pa	3.3041e+015 Pa
Information		
Time	1. s	0.32564 s
Load Step	1	
Substep	27	13
Iteration Number	29	14

FIGURE-6.4-Model (A4) > Static Structural (A5) > Solution (A6) > Equivalent Stress

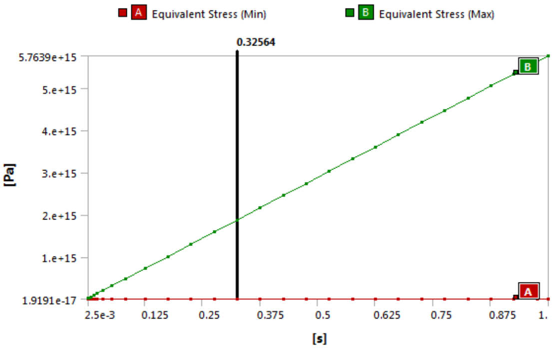


TABLE 6. 13
Model (A4) > Static Structural (A5) > Solution (A6) > Equivalent Stress

Time [s]	Minimum [Pa]	Maximum [Pa]
2.5e-003	1.2817e-009	1.441e+013
5.e-003	7.7388e-010	2.8819e+013
8.75e-003	5.0816e-010	5.0434e+013
1.4375e-002	1.9191e-017	8.2856e+013
2.2812e-002	1.8969e-009	1.3149e+014
3.5469e-002	4.124e-009	2.0444e+014
5.4453e-002	9.9251e-010	3.1386e+014
8.293e-002	4.838e-009	4.78e+014
0.12564	7.0217e-009	7.242e+014
0.17564	4.9334e-009	1.0124e+015
0.22564	4.2322e-009	1.3006e+015
0.27564	2.1545e-009	1.5888e+015
0.32564	1.1794e-009	1.877e+015
0.37564	3.4324e-009	2.1652e+015
0.42564	3.2208e-009	2.4534e+015
0.47564	3.2519e-009	2.7416e+015
0.52564	4.256e-010	3.0298e+015
0.57564	2.3927e-009	3.3179e+015
0.62564	4.0819e-009	3.6061e+015

0.67564	6.9676e-009	3.8943e+015
0.72564	4.7056e-009	4.1825e+015
0.77564	1.8444e-009	4.4707e+015
0.82564	3.7097e-009	4.7589e+015
0.87564	7.495e-009	5.0471e+015
0.92564	2.4282e-009	5.3353e+015
0.97564	3.4423e-009	5.6235e+015
1.	2.3314e-009	5.7639e+015

FIGURE-6.5-Model (A4) > Static Structural (A5) > Solution (A6) > Maximum Shear Stress

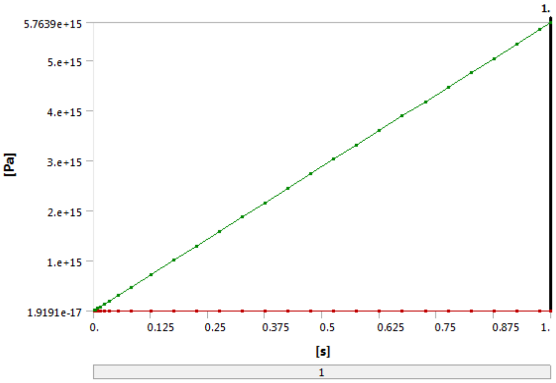


TABLE 6. 14
Model (A4) > Static Structural (A5) > Solution (A6) > Maximum Shear Stress

Time [s]	Minimum [Pa]	Maximum [Pa]
2.5e-003	6.7236e-010	8.2603e+012
5.e-003	4.2253e-010	1.6521e+013
8.75e-003	2.8222e-010	2.8911e+013
1.4375e-002	1.108e-017	4.7496e+013
2.2812e-002	1.0145e-009	7.5375e+013
3.5469e-002	2.1826e-009	1.1719e+014
5.4453e-002	5.7221e-010	1.7992e+014
8.293e-002	2.7765e-009	2.7401e+014
0.12564	4.0532e-009	4.1514e+014
0.17564	2.8421e-009	5.8035e+014
0.22564	2.4434e-009	7.4555e+014
0.27564	1.216e-009	9.1076e+014
0.32564	6.6861e-010	1.076e+015
0.37564	1.9783e-009	1.2412e+015
0.42564	1.655e-009	1.4064e+015
0.47564	1.8667e-009	1.5716e+015
0.52564	2.4262e-010	1.7368e+015
0.57564	1.3035e-009	1.902e+015
0.62564	2.2804e-009	2.0672e+015
0.67564	3.931e-009	2.2324e+015
0.72564	2.6341e-009	2.3976e+015
0.77564	9.8937e-010	2.5628e+015

0.82564	2.0937e-009	2.728e+015
0.87564	4.214e-009	2.8932e+015
0.92564	1.3908e-009	3.0584e+015
0.97564	1.9868e-009	3.2236e+015
1.	1.3357e-009	3.3041e+015

Chart

FIGURE 6. 6
Model (A4) > Chart

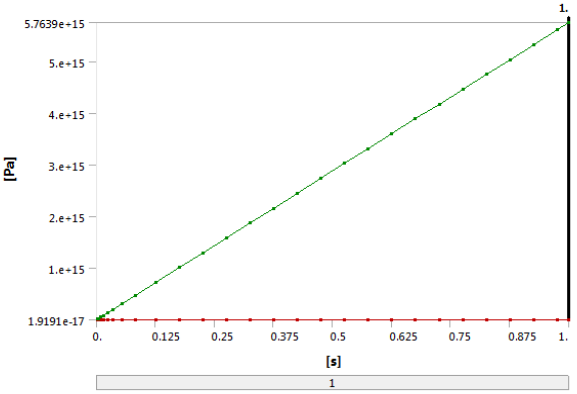


TABLE 6. 15
Model (A4) > Chart

Steps	Time [s]	[A] Equivalent Stress (Min) [Pa]	[B] Equivalent Stress (Max) [Pa]
1	2.5e-003	1.2817e-009	1.441e+013
	5.e-003	7.7388e-010	2.8819e+013
	8.75e-003	5.0816e-010	5.0434e+013
	1.4375e-002	1.9191e-017	8.2856e+013
	2.2812e-002	1.8969e-009	1.3149e+014
	3.5469e-002	4.124e-009	2.0444e+014
	5.4453e-002	9.9251e-010	3.1386e+014
	8.293e-002	4.838e-009	4.78e+014
	0.12564	7.0217e-009	7.242e+014
	0.17564	4.9334e-009	1.0124e+015
	0.22564	4.2322e-009	1.3006e+015
	0.27564	2.1545e-009	1.5888e+015
	0.32564	1.1794e-009	1.877e+015
	0.37564	3.4324e-009	2.1652e+015
	0.42564	3.2208e-009	2.4534e+015
	0.47564	3.2519e-009	2.7416e+015
	0.52564	4.256e-010	3.0298e+015

	0.57564	2.3927e-009	3.3179e+015
	0.62564	4.0819e-009	3.6061e+015
	0.67564	6.9676e-009	3.8943e+015
	0.72564	4.7056e-009	4.1825e+015
	0.77564	1.8444e-009	4.4707e+015
	0.82564	3.7097e-009	4.7589e+015
	0.87564	7.495e-009	5.0471e+015
	0.92564	2.4282e-009	5.3353e+015
	0.97564	3.4423e-009	5.6235e+015
	1.	2.3314e-009	5.7639e+015

Chart 2

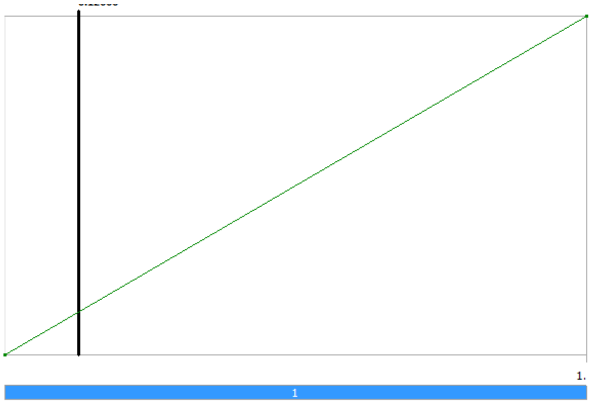


FIGURE 6.7
Model (A4) > Chart 2

TABLE 6. 16
Model (A4) > Chart 2

Steps	Time [s]	[A] Equivalent Stress (Min) [Pa]	[B] Equivalent Stress (Max) [Pa]
1	2.5e-003	1.2817e-009	1.441e+013
	5.e-003	7.7388e-010	2.8819e+013
	8.75e-003	5.0816e-010	5.0434e+013
	1.4375e-002	1.9191e-017	8.2856e+013
	2.2812e-002	1.8969e-009	1.3149e+014
	3.5469e-002	4.124e-009	2.0444e+014
	5.4453e-002	9.9251e-010	3.1386e+014
	8.293e-002	4.838e-009	4.78e+014

	0.12564	7.0217e-009	7.242e+014
	0.17564	4.9334e-009	1.0124e+015
	0.22564	4.2322e-009	1.3006e+015
	0.27564	2.1545e-009	1.5888e+015
	0.32564	1.1794e-009	1.877e+015
	0.37564	3.4324e-009	2.1652e+015
	0.42564	3.2208e-009	2.4534e+015
	0.47564	3.2519e-009	2.7416e+015
	0.52564	4.256e-010	3.0298e+015
	0.57564	2.3927e-009	3.3179e+015
	0.62564	4.0819e-009	3.6061e+015
	0.67564	6.9676e-009	3.8943e+015
	0.72564	4.7056e-009	4.1825e+015
	0.77564	1.8444e-009	4.4707e+015
	0.82564	3.7097e-009	4.7589e+015
	0.87564	7.495e-009	5.0471e+015
	0.92564	2.4282e-009	5.3353e+015
	0.97564	3.4423e-009	5.6235e+015
	1.	2.3314e-009	5.7639e+015

Material Data

Structural Steel

TABLE 6. 17
Structural Steel > Constants

Density	7850 kg m^-3
Coefficient of Thermal Expansion	1.2e-005 C^-1
Specific Heat	434 J kg^-1 C^-1
Thermal Conductivity	60.5 W m^-1 C^-1
Resistivity	1.7e-007 ohm m

TABLE6.18

Structural Steel > Compressive Ultimate Strength

Compressive Ultimate Strength Pa
0

TABLE 6.19

Structural Steel > Compressive Yield Strength

Compressive Yield Strength Pa
2.5e+008

TABLE 6.20

Structural Steel > Tensile Yield Strength

Tensile Yield Strength Pa
2.5e+008

TABLE 6.21

Structural Steel > Tensile Ultimate Strength

Tensile Ultimate Strength Pa
4.6e+008

TABLE 6. 22

Structural Steel > Isotropic Secant Coefficient of Thermal Expansion

Reference Temperature C
22

TABLE 6. 23

Structural Steel > Alternating Stress Mean Stress

Alternating Stress Pa	Cycles	Mean Stress Pa
3.999e+009	10	0
2.827e+009	20	0
1.896e+009	50	0
1.413e+009	100	0
1.069e+009	200	0
4.41e+008	2000	0
2.62e+008	10000	0
2.14e+008	20000	0
1.38e+008	1.e+005	0
1.14e+008	2.e+005	0
8.62e+007	1.e+006	0

TABLE 6.24

Structural Steel > Strain-Life Parameters

Strength Coefficient Pa	Strength Exponent	Ductility Coefficient	Ductility Exponent	Cyclic Strength Coefficient Pa	Cyclic Strain Hardening Exponent
9.2e+008	-0.106	0.213	-0.47	1.e+009	0.2

TABLE 6.25
Structural Steel > Isotropic Elasticity

Temperature C	Young's Modulus Pa	Poisson's Ratio	Bulk Modulus Pa	Shear Modulus Pa
	2.e+011	0.3	1.6667e+011	7.6923e+010

TABLE 6.26
Structural Steel > Isotropic Relative Permeability

Relative Permeability
10000

Experimental Method-Upon creating a physical prototype identical in geometry and mechanical properties to the intended component during production, the same is set-up for testing under identical service conditions for the component on field. A comparison of the results obtained through physical experimentation and the analytical (using simulation/ software) could offer a basis forvalidation.To simulate the working conditions, the force considered to be applied at the spider mounting location as a **torsional moment could be about 25Nm**and above (based on the application and the size of the vehicle). However the value takes a minimum and a maximum limit depending on the **driving conditions** and the **auxiliary mechanisms** to assist the maneuverability of the vehicle.

CONCLUSION

There is a much scope in design of steering rod to minimize its defect due to twisting, Vibrations, etc., optimization of design [existing/optimized] will provide better stability and less vibration defects in steering rod as well as column for

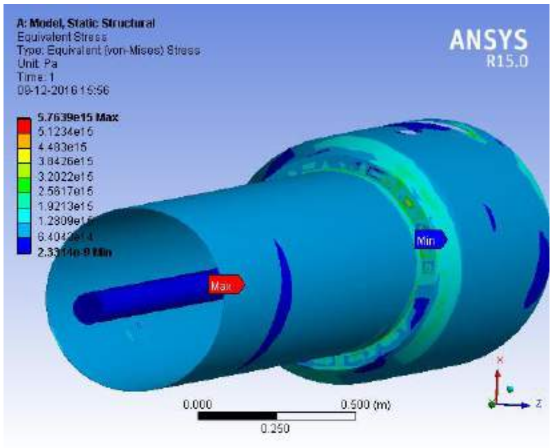


FIGURE7.1Equivalent Stress

making the rod better the rod ends should be made thicker where the coupling is to be used at the end were the universal joint used at the end.

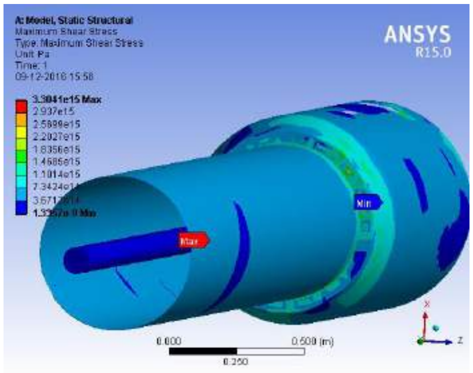


FIGURE 7.2Maximum Shear Stress

The material properties at both the ends should be made, different and instead of circular cut at the ends if any other shapes should be tried for better results.

Scope of the Project:

There is a much scope in design of steering rod to minimize its defect due to twisting, Vibrations, etc., optimization of design [existing/optimized] will provide better stability and less vibration defects in steering rod as well as column for making the rod better the rod ends should be made thicker where the coupling is to be used at the end were the universal joint used at the end. The material properties at both the ends should be made , different and instead of circular cut at the ends if any other shapes should be tried for better results.

References:

1. International Journal for numerical methods in engineering , Volume 2, issue 3 , July /sep/1970 pf “ Finite elwment analysis of torsional and torsional flexural stability problems: by ROSHDY S.BARSOUM , RICHARD H : GALLAGHER .
2. 2. Steering collapse analysis using detailed Fe model in ISSN : 2278-0181 , Vol.3 Issue6,June-2014,IJERTV#ISO617322138.
3. Dashboard support with vibration – damping feature in US 20050279909 A1 on Dec 22,2005 published by jochem Fischer, Bentler of Abstract.