STATIC STRUCTURAL ANALYSIS OF SUSPENSION ARM USING
FINITE ELEMENT METHOD

Jagwinder Singh¹, Siddhartha Saha²

¹Student, Mechanical Engineering, BBSBEC, Punjab, India
²Assistant Professor, Mechanical Engineering, BBSBEC, Punjab, India

Abstract
The function of suspension system is to absorb vibrations due to rough terrains or road disturbances and to provide stability under circumstances like accelerating, cornering, uneven road, braking, loading and unloading etc. Control arm is one of the most important part of the suspension system, as it joins the steering knuckle to the vehicle frame. Also suspension arm is responsible for up and down movement of wheels when hitting bumps. It is also designed to maximize the friction between tire contacts, patch the road surface to provide vehicle stability under any circumstances. It can be seen in many types of the suspensions like wishbone or double wishbone suspensions. Many times it is also called as A-type control arm. In this study control arm was reverse engineered. Reverse engineering refers to the process of obtaining a CAD model from an existing physical part. CAD model was prepared using CATIA v5 software and finite element analysis was done using ANSYS 14.5 software by importing the parasolid file to ANSYS. The model is subjected to loading and boundary conditions and then analyzed using the FEA techniques. The static structural analysis was done to find out the stress, deformation and safety factor of component. The model was meshed using 10-noded tetrahedral elements. Result obtained from the analysis were studied to check whether the design is safe or not. In some cases the stresses becomes more than safe limit. In that case optimization approach is carried out to increase the structural strength of the component. In this case maximum von-misses stress is 211 MPa which is below the yield strength of the material.

Keywords – Suspension System, Control Arm, FEA analysis, Reverse Engineering

1. INTRODUCTION
The lower suspension arm are connected to the vehicle frame with bushing and permits the wheel to go up and down in response to the road surface. Control arm is the most crucial part of the suspensions system. It is made from materials like steel, iron or aluminum. Suspension arm is very important for all vehicles on the road, if there is no suspension arm in suspension system, then it is expected that it can result in annoying vibrations and unwanted driving irregularities that could sometimes lead to road accidents like collisions with another car or obstruction on the road. Suspension arm is one of the most important component in the suspension system. It is fitted in various types of the suspensions like Macpherson, wishbone or double wishbone suspensions. During actual working conditions the maximum load is transferred from tire to the ball joint in Macpherson strut system and in double wishbone maximum load is transferred from upper arm to the lower arm which is responsible for the failure and twisting of lower suspension arm at the ball joint locations as well as control arm because of more impact load. Hence it essential to focus on the stress and deformation study of lower suspension arm to develop and the changes in existing design. The FEA approach is used for analysis of a suspension link for static and Von-Mises stress analysis of lower suspension arm. Analysis is to be done considering the Gross vehicle weight. As stress is higher than safe limit or yield strength some geometric changes adopted in design to make it safe.

2. OBJECTIVES
The aim of the project is to FEM analysis of lower arm. Following are the main objectives of project-
- Manual reverse engineering of the suspension arm
- Making a 3D solid parametric model of suspension link in CATIA software
- Meshing the model by 10 noded tetrahedral elements in ANSYS
- Static analysis to find Von-Mises stresses
- Static analysis deformation plot in ANSYS
- If fails corrective actions for design improvement of suspension link

3. PROBLEM DEFINITION
Due to the many functions control arm is main part in components of suspension system. As vehicle goes through speed breaker, bumps etc. and many other kinds of forces transmitted from the car wheels which are transferred to control arm via ball joint assembly to the wheel. Force values are mentioned in the load case. So in this study the key concern is to discover the maximum stress section and stress value in control arm and relate these values with tensile yield strength of material.

4. METHODOLOGY
To solve the problem mentioned above we have used the methodology. We have first manually reverse engineered...
the lower suspension arm with the help of special tools like Vernier gauge, radius gauge and divider etc. All the
dimensions which are measured have been recorded on the
drawing. Now by taking the dimensions from the drawing
modelling of lower suspension arm is done in CATIA
software. After modelling the lower suspension arm, we
have import the IGES file to the ANSYS which is FEA
software by using this software we have to solve the
problem.

Further we have to mesh the model by using the 10 noded
tetrahedral elements. Tetra elements give improved results
as compared to other type of elements, therefore tetrahedral
elements were used. In further step we have to apply the
loads i.e. forces and toques which are mentioned in the load
cases. The material used was mild steel. After the material
had applied we have to apply the boundary conditions
according the actual working conditions i.e. where it fix
d and where it is free. After that problem have to be solved by
using ANSYS software. In the next step we have to solve
the model in the analysis phase. The results are compared if
the results are satisfied then problem is solved and if not
then editing of model is done and again the steps should be
followed.

In following table particular conditions are shown in Table 4
for the typical component.

<table>
<thead>
<tr>
<th>Table4: Vehicle Specification</th>
</tr>
</thead>
<tbody>
<tr>
<td>Wheel Base</td>
</tr>
<tr>
<td>Overall Length</td>
</tr>
<tr>
<td>Overall Width</td>
</tr>
<tr>
<td>Overall Height</td>
</tr>
<tr>
<td>Ground Clearance</td>
</tr>
<tr>
<td>Turning Radius</td>
</tr>
<tr>
<td>Unladed Weight</td>
</tr>
<tr>
<td>Laden Weight</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Table4: Assumptions made for calculations</th>
</tr>
</thead>
<tbody>
<tr>
<td>Force</td>
</tr>
<tr>
<td>Material</td>
</tr>
<tr>
<td>Density</td>
</tr>
</tbody>
</table>

5. STATIC ANALYSIS

5.1. Introduction

The control arm bushing consist of two types of bushings, a
durable rubber bushing and inner metal sleeve. The bushing
acts as hinged Damper to support the suspension and
provide a more controllable and comfortable ride.

The bushings are attached to the chassis or the frame of the
vehicle and the ball joint is attached to the wheel of the
vehicle.

5.2 3D Model of Control Arm

Control arm is first modeled in CATIA software which is
outstanding CAD software. CATIA is very user friendly
software which makes modeling so easy.
5.3. Mesh Generation

In this analysis mesh generation is auto generation by the ANSYS software, after that sizing of 5mm was given to the suspension arm. Tetrahedral 10-noded elements were used to mesh the lower suspension arm.

- No of Nodes – 9634
- No of Elements- 34175
- Element Type – Tetrahedral (10 node)

5.4. Loading and Boundary Conditions

5.4.1 Static Loads on the Vehicle

Weight of vehicle acting at its center of gravity

\[ \text{G} = \text{m} \cdot \text{g} \]

As we know approximately 0.39% of weight of the whole body or car is carried by front wheels and 0.61% weight carried by rear wheels.

So reaction equations are:
- Reaction on front wheels \((R_1) = 0.39 \times \text{G}\)
- Reaction on rear wheels \((R_2) = 0.61 \times \text{G}\)

So equation becomes:

\[ R_1 \cdot S_1 = R_2 \cdot S_2 \]

As we know

\[ S = S_1 + S_2 \]

Loads on front axle and rear axle are found by using the equilibrium equation:

\[ \text{G}_{FA} = \text{G} \times \frac{S_2}{S} \]
\[ \text{G}_{RA} = \text{G} \times \frac{S_1}{S} \]

Static load on one wheel of the front wheel:

\[ \text{G}_{FAW} = \frac{\text{G}_{FA}}{2} \]

Table 5.4

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>G</td>
<td>Weight of vehicle acting at CG</td>
</tr>
<tr>
<td>m</td>
<td>Mass</td>
</tr>
<tr>
<td>G</td>
<td>Acceleration due to gravity</td>
</tr>
<tr>
<td>R_1</td>
<td>Reaction on front wheels</td>
</tr>
<tr>
<td>R_2</td>
<td>Reaction on rear wheels</td>
</tr>
<tr>
<td>S_1</td>
<td>Distance between CG and front axle</td>
</tr>
<tr>
<td>S_2</td>
<td>Distance between CG and rear axle</td>
</tr>
<tr>
<td>S</td>
<td>Wheel base</td>
</tr>
<tr>
<td>G_{FA}</td>
<td>Loads on front axle</td>
</tr>
<tr>
<td>G_{RA}</td>
<td>Loads on rear axle</td>
</tr>
<tr>
<td>G_{FAW}</td>
<td>Loads on one wheel of front axle</td>
</tr>
</tbody>
</table>

Loads are in form of force values and moments. The boundary conditions applied on the component.
5.4.2. Boundary Conditions

6. RESULTS

Many features were analyzed like stress and displacement from FEM analysis and results that were found are given below.

6.1. Stress Analysis

In stress analysis of lower suspension arm subjected to high stress shown in figure 10. According to the methodology to safer the design stress induced in arm should be less than the yield strength of the material. The stress found from the analysis is 211.06MPa.

6.2. Total Deformation

Total deformation shows that at what stage the component will deflect from its original position. Total deformation should be within the limits and Total deformation found in the analysis was 0.65515mm.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Results</th>
</tr>
</thead>
<tbody>
<tr>
<td>Equivalent stress</td>
<td>211 MPa</td>
</tr>
<tr>
<td>Deformation</td>
<td>0.65515 mm</td>
</tr>
<tr>
<td>Safety Factor</td>
<td>1.1845</td>
</tr>
</tbody>
</table>

7. CONCLUSION

1. In stress analysis, the stresses of material for the given loading condition fall well in within the yield stress i.e. 211.06 MPa as shown in figure 6.1.
2. The total deformation due to the force applied on the suspension arm was 0.65515 mm which is maximum was at the ball bearing of the suspension arm shown in figure 6.2
3. The minimum safety factor was 1.1845 which shows the component is safe.
REFERENCES


