Knuckle Design for SAE Baja Vehicles using Real-life Force Data
Yash Punde¹, Sumedh Bagade², Amit Chopde³, Yugandhar Dhande⁴
Student¹, ², ³, ⁴
Department of Mechanical
Vishwakarma Institute of Technology, Maharashtra, India

Abstract:
The steering knuckle is often described as the most important dynamic component of the vehicle. It acts as a coupling between the unsprung mass and the sprung mass and helps the driver to obtain required feedback from the ground. Not only this but it is also one of the most heavily stressed components of not only wheel assembly but the entire vehicle. It is one of the major load-bearing members and is prone to sudden breakdown. Yet in the era of lightweight vehicles, performance and fuel economy have gained importance but yet safety is of utmost importance. Thus, finding a perfect balance between safety and optimization is an arduous task. Therefore, this project was undertaken where we found out the maximum stress by simulating some real-life conditions where the external force is deduced to be maximum and measured the corresponding values of strain generated in knuckle for the same by using a strain gauge sensor which had been previously calibrated in standard conditions. The corresponding value of stress and thus the maximum value of force in terms of ‘g’ force was then gained. The final model was then analyzed using Ansys.

Key Words: Steering, knuckle, vehicle, off-road vehicle, stress analysis, optimization.

1. INTRODUCTION:

The steering knuckle is often described as the most important dynamic component of the vehicle. It holds the wheel assembly together and provides housing for the wheel bearings. It also contains mounting points for the suspension wishbones and brake caliper mounting (in most cases). While in some suspension types, it also serves the purpose of shock absorber mounting. Being an intermediate member steering knuckle needs to carry the load transferred during impact form the tire to the shock absorber. So, it can be easily depicted from the above discussion that steering knuckle needs to be sturdy and robust in design so as to enable it to sustain such high loads. But in the era of performance cars, components cannot be heavily overdesigned just by justifying its importance. So, the optimization of steering knuckles is a trade-off between the factor of safety and performance. So, whilst designing, one needs to be concise about the magnitudes of input forces. The standard actual force values used by the automotive industry are highly confidential and are not available to a general consensus. Thus, by this experiment, we wish to make this data public to the students. To find the nearly truthful values of the input forces, we simulated some of the most extreme real-life conditions in a monitored manner and using the data acquisition system, were able to acquire the magnitudes of forces. The most severe condition occurs during impact from a certain height, that is, a fall from a certain height. Under our simulated condition, this height was assumed to a value of around six feet. A spot on the knuckle was so chosen that it was part that gets maximum deformation and would be able to accommodate two strain gauges each for compressive stress and bending stress. These strain gauges were calibrated beforehand in ideal testing conditions and were handled with appropriate precautions. The result of the simulations was then matched with the results obtained after the calibration of strain gauges and the values of stresses were obtained. These stress values were then converted to force values in the number of "g" force for the sake of generalization. The FEA of the same was done using Ansys.
3. CAD MODEL PREPARATION:

CAD model of the steering knuckle is prepared using SOLIDWORKS V16 and designed dimensions.

![Figure 1: Knuckle CAD Model](image1)

4. KNUCKLE DESIGN:

Kinematics hardpoint: Selection of hardpoint primarily depends on characteristics like caster, camber, toe change, halftrack change.

<table>
<thead>
<tr>
<th>Table 1. Knuckle design parameters</th>
</tr>
</thead>
<tbody>
<tr>
<td>Parameter</td>
</tr>
<tr>
<td>Material</td>
</tr>
<tr>
<td>Caster</td>
</tr>
<tr>
<td>Steering Axis Inclination</td>
</tr>
<tr>
<td>Stub diameter</td>
</tr>
<tr>
<td>Ball joint bore diameter</td>
</tr>
<tr>
<td>Stub offset horizontal</td>
</tr>
<tr>
<td>Stub offset vertical</td>
</tr>
</tbody>
</table>

5. MATERIAL SPECIFICATION:

<table>
<thead>
<tr>
<th>Table 2. Knuckle material specifications</th>
</tr>
</thead>
<tbody>
<tr>
<td>Parameter</td>
</tr>
<tr>
<td>Yield strength</td>
</tr>
<tr>
<td>Ultimate-tensile strength</td>
</tr>
<tr>
<td>Young’s modulus</td>
</tr>
<tr>
<td>Poison’s Ratio</td>
</tr>
</tbody>
</table>

6. FE MODELLING:

- FE Modelling is done as per general guideline, tetra collapse is kept above 0.1.

- Perform tria mesh with finer mesh in critical areas (mesh size is subjective to model dimensions but general guideline is to keep at least 2 rows of tria. elements on the fillets).
- R-tria elements to be used on all the cylindrical surfaces and regular tria. elements to be used on all the other surfaces.
- Convert the mesh to tetra mesh and ensure tetra collapse to be above 0.1.
- Achieve mesh convergence for accurate results and get accurate mesh size.
- Load of 3g is applied in the vertically upward direction at the wheel end of knuckle and upper ball joint bore is constrained in all dof as shown below:

![Figure 2: Meshed Model with loads and boundary conditions](image2)

7. PHYSICAL TESTING:

A strain gauge is used to calculate strain and stress. For all three major load cases same type of setup is used and strain gauges are pasted on the position of max stress (obtained from FEA software) of respective load case. Below are snapshots of test setup:

![Figure 3: Experimental Setup](image3)
Testing has been done for 3 load-cases:
i. Acceleration/Braking load (Longitudinal Force).
ii. Cornering Loading (Lateral Force).
iii. Drop Test (Bump Force).

Procedure:
i. Vehicle driven by a single driver with all the test setup is dropped from 6 feet tabletop.
ii. Readings from strain gauge are saved on a memory storage.
iii. The raw output from strain gauge is converted into useful voltage values using electronic devices and then these values are converted into strain values by using previous calibration data for the gauges.
iv. Maximum value of stress is noted down for the given load case.
Similar testing is carried out for other two load-cases and stresses are calculated, max stress of all three load-cases is used for further evaluation

8. RESULTS AND DISCUSSION:
After converting the raw sensor data into useful strain values, the stresses are then found by using material properties. The vehicle used, weighed a total of 220kg including the driver. The results have been converted into g force notation, so that these values could be used by designers with different goals and systems according to their need. Following are a set of results for the harshest 6 feet blind drop load case

9. STRESS ASSESSMENT FROM TEST DATA
Reading of voltage from digital multi-meter is 0.0332V. For this reading equivalent strain (using calibration data) strain is 0.001906. For 20MnCr5 Young’s Modulus is 200GPa. Stress induced =strain* Young’s Modulus Stress=381.2 N/mm2
For this stress value Force applied is found out by iterations on Ansys. Motive of iterations is to get maximum stress induced in the gauged area of knuckle around 381.2 N/mm2. We get force value =6971 N which is equal to 3.23*g force.

10. G FORCE VALUES FOR ALL THE LOAD CONDITIONS:
Following table depicts the g force values obtained from the testing:

<table>
<thead>
<tr>
<th>Load Case</th>
<th>Value (g force)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bump force</td>
<td>3.23</td>
</tr>
<tr>
<td>Lateral loading</td>
<td>2.1</td>
</tr>
<tr>
<td>Longitudinal loading</td>
<td>1.5</td>
</tr>
</tbody>
</table>

11. FEA RESULTS:

12. CONCLUSION:
Max stress in knuckle occurs during drop test. Cornering test and acceleration/braking test is still however crucial for a successfully optimized design. Max force and critical spot can be determined by FE analysis. Based upon evaluated results knuckle can be designed for the vehicle for standard load-cases and can be optimized based upon the results of baseline FEA results.

Acknowledgements:
The authors would like to acknowledge the support received from VIT Pune. And thank the mechanical engineering department for the support provided to carry out this work.

13. REFERENCES:
