Abstract: The vehicle suspension system is always responsible for driving safety and comfort. The suspension unit carries the whole vehicle body and transmits all forces between body and road. Mostly Structure optimization techniques in static load conditions have been used in automotive industry for light weight and for performance improvement of modern new cars. However, these static loading conditions could not represent all the severe situations of various automobile parts which subjected to complex loads those varying with time, especially for lower control arm of front suspension of automobile. This paper deals with static analysis of the upper arm suspension of double wishbone suspension. This paper shows the study of practical example for static analysis and optimization of upper control arm of Mahindra Scorpion. In this optimization of upper control arm 6.29% weight reduction is achieved.

1. Introduction

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 the 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 [1]. Suspension arm is one of the most important components 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.

It is essential to focus on the stress and deformation study of upper 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 upper suspension arm.

2. Project Objectives

a) Study of existing upper control arm & analyze it by using ANSYS software.
b) To optimize the existing upper control arm to reduce weight by topology optimization.

3. Methodology of Project

- CAD Model Generation
- Input parameters: - Input parameters will be getting through reverse engineering. Reverse engineering will follow a method of hand calculation
- Model creation in CATIA V5 from input parameters.
- Determination of loading.
- FEA analysis of control arm discretized model by applying the required boundary conditions and for the various loadings and carrying out static analysis
- Material is removed from low stressed region to optimize the model
- Static analysis is done the model
- Testing will be carried to validate analysis results.
- Validation & the costing of project is approximately Rs. 40,000.

4. Analysis of Existing Upper Control Arm

Force Calculation

- Condition I: static condition:

Following fig shows the forces on a stationary car. The gravitational force (mg) acts through the center of gravity and the reaction (remember: to every action there is always an equal and opposite reaction) acts always through the contact patches between the tires and the road [11]. The vectors shows the combined reactions at both front wheels (R₁) and both rear wheels (R₂).
Total weight of the car = 2650 kg = 25996.5 N = 25.997 KN
Capacity of Scorpio is 7 persons, and take weight of each person is 70 kg.
So total weight of person = 7 * 70 = 490 kg.

Vehicle mass = Weight of vehicle (car) – No. of persons weight

= 2650 – 490
= 2160 kg = 21189.6 N = 21.19 KN

This total weight must be divided into front axle weight and rear axle weight. 52% of total weight is taken by front axle and 48% of total weight is taken by rear axle.

Front axle weight = 0.52 * 2650
= 1378 kg = 13518.18 N
= 13.52 KN

Reaction at one wheel = 13518.18/2 = 6759.09 N

Rear axle weight = 1272 kg = 12478.32 N = 12.48 KN

We have to find the term bcg, Consider a simply supported beam, where force F = 25.996 KN which acts at a distance X(S1) from point A,

\[ \text{Front Axle Breaking Force (FB) per Wheel} \]
\[ \text{FB} = \frac{\mu}{2} \left[ \text{Static} + \text{dynamic load} \right] \]
\[ = \frac{\mu}{2} \left[ W * \text{bcg}/S + m * \ddot{a} * \text{hcg}/S \right] \]
\[ = \frac{\mu}{2} W \left[ \text{bcg}/S + \ddot{a}/g \text{hcg}/S \right] \]

We have to find the term bcg, Consider a simply supported beam, where force F = 25.996 KN which acts at a distance X(S1) from point A,

\[ \text{bcg} = 3040 - X = 1580.8 \text{ mm} \]

Breaking force FB can be calculated as FB = 12.1 KN

b. Vertical Force (FV):

\[ \text{FV} = 3/2 \left[ \text{Static} + \text{dynamic load} \right] \]
\[ \text{FV} = 3/2 \left[ W * \text{bcg}/l + m * \ddot{a} * \text{hcg}/l \right] \]
\[ = 3/2 W \left[ \text{bcg}/l + \ddot{a}/g \text{hcg}/l \right] = 24.3 \text{ KN} \]
c. Lateral Force (FL):

\[ FL = W[\text{Static} + \text{dynamic load}] \]
\[ FL = W \left( \frac{bcg}{l} + \alpha \cdot \frac{hcg}{gl} \right) = 16.2\text{KN} \]

Figure 5. CAD model view of Upper Control Arm

After the load and boundary conditions applied on the model the solver deck is exported from hypermesh to Ansys in .cdb format. The model is then imported in Ansys and solved. The results obtained are as follows:

Figure 6. Load & boundary conditions applied in Hypermesh

Figure 7. Deformation result for existing upper control arm

Figure 8. Von mises stress for existing upper control arm

The maximum deformation is found to be 0.293mm which is very less. The max stress obtained is 181MPa which means the design is safe. As the design is safe, means the stresses are well within the ultimate stress 390MPa and deformation is much less there is a scope for optimization.

5. Optimization

Optimization is a technique or process or methodology of making something (as a design, system or decision) as fully perfect or effective as possible. The basic principle of optimization is to find the best possible solution under given circumstances. Structural optimization is one application of optimization. Here the purpose is to find the optimal material distribution according to some given demands of a structure. Sizing optimization is the simplest form of structural optimization [12]. The shape of the structure is known and the objective is to optimize the structure by adjusting sizes of components. Here the design variables are the sizes of the structural elements, for example the diameter of a rod or the thickness of a beam or a sheet metal. See Figure 9(a) for an example of size optimization where the diameters of the rods are the design variables.

Figure 9. Types of Optimization
In shape optimization the design variables can for example be thickness distribution along structural members, diameter of holes, radii of fillets or any other measure. See Figure 9(b) for an example of shape optimization.

The most general form of structural optimization is topology optimization [12]. As with shape and size optimization the purpose is to find the optimum distribution of material. With topology optimization the resulting shape or topology is not known, the numbers of holes, bodies, etc., are not decided upon. See Figure 9(c).

Optimization methods were developed to have lighter, less cost and may have better strength also. Many optimization types, methods, software technique and tools are available due to the revolution of high speed computing and software development. Topology optimization: It is an optimization process which gives the good optimum material layout according to the design space and loading case.

From the plots of von-mises stress plot the blue colored region show stresses which are very low. So there is a scope for optimization by removing the material. The material is removed by iterations and the model is checked for it withstands the load and is safe.

A. Iteration 1: Two elongated holes measuring 40xR15 mm from the arm of the upper control arm is removed as show below

Figure 10. Deformation result for upper control arm: Iteration 1

Figure 11. Von mises stress for upper control arm: Iteration 1

The maximum deformation is found to be 0.256mm which is very less. The max stress obtained is 187MPa which means the design is safe.

B. Iteration 2: One elongated and one circle is removed after iteration 1 from other side of arm measuring 60xR15 mm and D-30 mm respectively.

Figure 12. Deformation result for upper control arm: Iteration 2

Figure 13. Von mises stress for upper control arm: Iteration 2
The maximum deformation is found to be 0.261mm which is very less. The max stress obtained is 199MPa, which means the design is safe.

C. Iteration 3: Two section of irregular shape is removed after iteration_2 from the upper part of the upper control arm

The maximum deformation is found to be 0.262mm which is very less. The max stress obtained is 202MPa, which means the design is safe.

D. Iteration 4: Another two elongated holes from the UCA at different cross section is removed after iteration_3 measuring as 60xR15 mm & 40xR15 mm.

The maximum deformation is found to be 0.260mm which is very less. The max stress obtained is 209MPa, which means the design is safe.

E. Iteration 5: Welding is added for connecting more stressed part to main body as shown below
The maximum deformation is found to be 0.272mm which is very less. The max stress obtained is 213.8MPa, which means the design is safe.

6. Experimental Validation

The experimental investigation performed on fabricated prototype on universal testing machine. The input conditions are recreated in the lab while the component is tested. The loading and the boundary conditions are matching the practical working conditions under which the specimen is expected to perform. The loading conditions are applied on the prototype for testing purpose.

Deformation from of optimized upper control arm from Ansys result = 0.272mm.
Deformation of optimized upper control arm from testing = 0.2630mm.
Percentage Error = (FEA - Experimental) / FEA.
= (0.272 -0.263) /0.272
= 0.03308
= 3.308 %.

Hence the result is validated with a % error of approximately 3.3.

7. Result and Discussion

The existing model is analyzed in Ansys and find out deformation and stresses. In existing model analysis we got max deformation is 0.293mm and maximum stress is 181MPa.

Then existing model is optimized by iterations so as to converge towards the optimum results. The maximum stress value for optimized upper control arm is 213.8 Mpa. The critical yield value of the material is 390Mpa. From the results, the stresses are well below the critical value. The final optimized model is capable of handling the loading conditions under the safety limits.

Deformation of existing model under loading condition is 0.293mm while deformation got in optimized model is 0.272mm. From above values of deformation optimized model have less deformation than existing which safe.
Table 2. Weight Reduction.

<table>
<thead>
<tr>
<th>Component</th>
<th>Weight (Kg)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Existing</td>
<td>8.58</td>
</tr>
<tr>
<td>iteration_1</td>
<td>8.47</td>
</tr>
<tr>
<td>iteration_2</td>
<td>8.37</td>
</tr>
<tr>
<td>iteration_3</td>
<td>8.18</td>
</tr>
<tr>
<td>iteration_4</td>
<td>8.04</td>
</tr>
<tr>
<td>iteration_4</td>
<td>8.04</td>
</tr>
</tbody>
</table>

Percentage Reduction = \[
\frac{\text{Existing} - \text{Optimized}}{\text{Existing}}
\]

= \[
\frac{(8.58- 8.04)}{8.58}
\]

= 6.29 \%

A comparative study of FEA and Experimental results is made. From the results it can be concluded that the validation of results show close resemblance with a % error of 3.3. Hence the objective is achieved. This study reduced the final weight without compromising the strength of the UCA at obtained boundary condition.

8. Conclusion

FEA & Topology optimization techniques can be effectively used for effective performance and weight reduction of upper control arm. Final optimized model has been showed positive results it means stress and deformation values are in safe limit.

After topology optimization of upper control arm, achieved around 6.29 % weight reduction.

9. References


