Design And Analysis Of Formula Sae Vehicle Rear Upright Andcomparision Of Analytical And Software Analysis Solutions Ofprincipalstressesat Differentpoints

Sasanapuri Sandeep1 ,M. Lakshmi Sramika2

1M.Tech(CAD/CAM) Scholar Dept of Mechanical Engineering Avanthi Institute Of Engineering & Technology
2Associate Professor Dept of Mechanical Engineering Avanthi Institute Of Engineering & Technology

Abstract: Upright is used in vehicles which connects frame or chassis and tires. Chassis is connected to upright with A-arms at the top and bottom using different types of fasteners through which the load is transmitted to it. I choose to design a FSAE(formula society of automobile engineers) car upright based on the data collected from real time hybrid formulavehicle. Upright is modeled in Solidworks 2015 and all principle stresses and strains are found through simulation analysis software ANSYS 19.2. Now these solutions are compared with the analytical solutions which is solved using MATLAB software, So that we can be able to predict which one is more accurate and how these simulation software are being able to solve problems based on these concepts.

1. Introduction

A major component of the vehicles suspension system which allows the steering arm to turn the front wheels and supports the vertical load of the vehicle. It is also known as knuckle. Upright is one which connects steering arms, control arms, springs, brake calipers, tires and incase rear upright then it also connects axles as shown in fig.1. It provides adjustment of different suspension parameters like steering Ackerman geometry, caster, camber and scrub radius [1]. The forces encountered by the car due to road and tire interactions go through upright, so the upright should be stiff and strong to withstand high forces. Also be able to withstand failure at the time of crashing or other emergencies because the failure of upright makes the car undrivable. Car upright is subjected to fatigue load, braking force, cornering force, impact load during its service life [2]. The upright knuckle design determines the geometry of the suspension’s “outboard” side. (The mounting points on the chassis and wishbones / links form the “inboard” side of the suspension and make their own contribution to the overall geometry of the suspension.)

The fig.1 illustrates an example of a non-driven independent wishbone suspension. The upright (Yellow) is attached to the car using the upper and lower wishbones which have fasteners (ball joints or rod ends). This allows the upright to rotate about the kingpin axis and move vertically.

Part attached to the upright is the spindle (green). Bearings (Orange) are inserted into the hub (Red) and it is slide over the spindle and held in place by a retaining nut. The brake disc (Blue) placed over the threaded bolts extending from the hub. The brake caliper (Light blue) is attached using a bracket to upright.

The steering angle of the upright can be set using the steering/toe link which has a rod end that fastens using ball joint to an arm (Purple) on the upright.
Before designing a upright we need to know the following suspension parameters which are based on the alignment of a vehicle such as:

2. **Camber Angle**

   It is the angle measured between the wheel vertical alignment normal to its surface. If the wheel is perpendicular to its surface then its camber angle is 0 degrees. It is described as negative when top of the wheel begins to tilt inward that means towards the vehicle whereas the tire tilts
outwards it is positive camber angle [4]. Most vehicles have neutral camber angles and most racecars have negative camber angle. Negative camber have more grip advantage during cornering thus having good handling whereas in neutral camber it results in tire wear.

3. **Caster angle**
4. It is the measure of angle between the steering axis and vertical axis from the side view as shown in fig.3. When both the axis are in the same angle then it is neutral caster. When the top of steering angle moves forward it is negative angle and vice versa. Most vehicles have positive caster as it makes stable at high speeds and increases steering stability [4].

5. **Other parameters**
The kingpin inclination is the line formed when joining the lower ball joint (LBJ) and upper ball joint (UPJ). It is used to determine the camber and caster angle based on the kingpin inclination [5]. The distance kingpin inclination is offset from the tire center line is called scrub radius as shown in fig.4.
6. LITERATURE REVIEW

The service life of an upright is based on its dynamic conditions like fatigue loads always apply on the upright during jounce and bounce. Longitudinal loads are applied when it is in static and lateral loads are being applied when during braking and centrifugal forces act during cornering of vehicle[1].

While doing analysis we need to take consideration of camber and caster angle which is considered as 6° and then the loads are applied on it. Also the forces acted by wheel bearing is not taken into consideration while doing the analysis. The upright for doing static analysis is considered the rear one as it is easier to consider by eliminating the forces acted by the shaft [8].

Weight is one of the important considerations for a race car component mainly upright as it comes under unsprung mass. So optimization of design is important and also the selection of material which gives us the actual weight of the component whereas it should be rigid to withstand 1/4th of car’s sprung mass and three times gravity acting on it during longitudinal loading[9].

7. DESIGN OF UPRIGHT

Design considerations

We consider various parameters while designing a component. Irrespective of other details the major design parameters determine the performance of the upright.

The parameters that are considered while designing the uprights:

1. Castor angle along the vertical axis of the upright is 6°.
2. It should have a brake caliper mounted on one side of the upright and the steering rack mount should be on the opposite side.
3. Bores should be provided on the top and bottom of the upright to accommodate the ball joints.
4. Sufficient wall thickness to make the component strong and stiff to withstand the weight of the vehicle.
5. Length of the upright should be considered as it should fit in the wheel hub.
6. Weight is an important parameter as it helps for fuel economy and good handling performance as well as more acceleration.

In order to design an upright you have to consider all the suspension parameters such as wheel dimensions, the estimated weight of the whole vehicle, track width, wheel base. This following data is taken from a Formula hybrid vehicle team.

| Wheelbase       | 68 in |
|-----------------|-------|
| Overall length of vehicle | 116 in |
Table 1: Suspension parameters

| Trackwidth | Front: 48 in |
|-------------|-------------|
| Tires       | R13 155 65  |
| Mass of vehicle | 470 kg   |
| Ground clearance | 2 in     |
| Suspension  | Double wishbone damper to lower wishbone |

8. Material selection

After studying the comparison made in the table below an easy discussion can be made that AISI1018 is the suitable material as it is having better weight to strength ratio at a reasonable cost in comparison to other materials.

| Material  | Ultimate strength (MPa) | Yield strength (MPa) | Density (g/cc) | Strength to weight ratio | Cost/mealter |
|-----------|-------------------------|----------------------|----------------|--------------------------|--------------|
| AISI10    | 350                     | 340                  | 7.87           | 55-60                    | 4.12         |
| AISI10    | 380                     | 370                  | 7.87           | 60-62                    | 6.2          |
| AISI41    | 410                     | 400                  | 7.85           | 70-75                    | 8            |

Table 2: Material properties comparison

9. Forces acting on an upright

- Longitudinal force during braking.
  While braking the weight on rear side tends to come to front side of the vehicle. So there is load transfer takes place from rear to front.
  Force at the front side = mass at the rear side * acceleration
  Let the mass at the rear side of the vehicle be 0.6 times the total weight = 0.6 * 470 = 282 kg
  Force is considered for worst-case conditions and 4 g loads are applied on the upright. Force = 282 * 4 * 9.8 = 110544.4 N
  Force on 1 wheel = 110544.4 / 2 = 5527.2 N

- Lateral forces
  Lateral forces are because of two reasons centrifugal forces and lateral load transfer from outside to inside while turning.
  Turning radius = 3 m
  V = 30 km/h = 8.33 m/s
  Centrifugal force = \( \frac{mv^2}{r} \) = \( \frac{0.4 * 470 * 8.33^2}{3} \) = 4348.37
  Lateral load transfer = 0.4 * 470 = 188 kg
  Force = 188 * 3 * 9.8 = 5527.2 N
  Lateral force for one wheel = 2763.6 N
  Force acting on caliper mounting = torque / radius = 58000 / 110 = 527.27 N

10. Modeling

Upright is modeled in Solid Works 2015 software, which is used for designing, drafting and as well as analysis of different components. It is developed by Dassault Systems. Using all the design considerations and based on the hybrid formula, rules this upright is being modeled.
As shown in figure.6 we can see the cross section area where the lateral and longitudinal forces are applied. They are applied along the axis so while applying the forces the axis is being rotated in Ansys and then the forces are applied.
11. THEORITICALCALCULATIONS

Stress is a measure of external force acting on the cross sectional area of a component or body. Stress has a unit of N/m². There are two types of stress: 1. Normal stress - when force acts perpendicular to surface and other one is 2. Shear force - when force acts parallel to surface of an object.

\[ \sigma = \frac{p}{A} \]

When we consider this equation, there are lots of assumptions. They are - we assume all materials are homogeneous, isotropic and elastic as well as object as prismatic meaning the cross-section will be same along its length. Because of all these assumptions, the object deforms uniformly at every point along its cross-section. Normal stress at a point on a cross section is defined by (with similar equations in the y and z directions) [10].

\[ \sigma = \lim \]
Every small area is subjected to similar forces, and the sum of all the forces should be equal to the resultant forces. We integrate both sides of the equation and arrive at a relationship for normal stress.

\[ \int dF = \int \sigma dA \]

\[ \therefore \sigma = \sigma_A \]

So we used the above equation to find \( \sigma_x \) and \( \sigma_y \) using the area on which the stress acts upon.

\( \sigma_x = 3.83 \text{MPa} \quad \sigma_y = 2.87 \text{MPa} \)

12. Finding principle stresses

As the caster angle is applied so now the upright is rotated with 6° and then we find the principle stresses and also the shear stress formed on upright using equation (1) and (2). These equations are being solved using MATLAB.

\[ \sigma = \frac{\sigma_x + \sigma_y}{2} \]

\[ \sigma_x - \sigma_y \]

\[ + \tau x \] (1)

\[ \pm \frac{2 \tau_{xy}}{\sigma_x - \sigma_y} \]

\[ \tan 2\theta_p = \frac{2 \tau_{xy}}{\sigma_x - \sigma_y} \] (2)
13. STATIC STRUCTURAL ANALYSIS

ANSYS

It is a simulation software package where it solves different governing equations to do the static structural analysis as well as other simulations. They solve these equations by dividing the component to a number of small parts. This process is called meshing, where it is a finite element analysis. The results can be obtained in various formats. As we are not able to do structural analysis for complex structures with different kinds of loads applied on the model, so we use ANSYS to do simulations over the complex structures.

Steps to do structural analysis in ANSYS:

1. Select a chosen material from the engineering data.
2. Create a geometry or import IGES geometry file from SolidWorks.
3. We should do meshing to get accurate results after simulation.
4. After meshing, different types of loads are being applied on the upright, and the results are being obtained as seen below.

The upright is rotated 6° around the Z-axis as we need to find principal stresses when the caster angle is in 6° angle. This is being done in ANSYS software setup.

Figure 8: Rotation of axis in ANSYS
Meshing

**Figure 9** Meshing with element size 6mm

| Elementsize | 6.0mm |
|-------------|-------|
| No.ofnodes  | 23261 |
| Refinement  | Fine  |

*Table 3: Mesh information*

Forces applied in longitudinal and lateral directions

**Figure 10** Forces applied during zero caster angle
Forces applied during positive caster angle (After rotation)

Total deformation at zero caster

Figure 11: Forces applied during positive caster angle (After rotation)

Figure 12: Total deformation on upright
After the rotation of axis through the z-axis with 6°, the loads are being applied as seen in figure 11. We can see the decrease of deformation value when compared to upright at zero degree angle in figure 13.

**Total deformation at positive caster (After rotation)**

![Total deformation on upright](image1.png)

**Stress distribution at zero caster**

![Stress distribution on upright](image2.png)

**Figure 13 Total deformation on upright**

**Figure 14 Von-Mises stresses**

2805
Stress distribution at positive caster (After rotation)

Finding factor of safety

Here we are using maximum distortion energy theory to find factor of safety for chassis. According to this theory when the material is subjected to biaxial or triaxial stress it will fail only if maximum shape distortion energy is greater than shape distortion energy of specimen. (This theory is generally used for ductile materials).

\[
\frac{\sigma^2}{\sigma_yt^2} = \frac{\sigma_1^2}{\sigma_y^2} + \frac{\sigma_2^2}{\sigma_y^2} - \frac{\sigma_1\sigma_2}{\sigma_y^2}
\]

\( \sigma_y \) = Yield strength of material
\( \sigma_1 \) = Maximum stress
\( \sigma_2 \) = Minimum stress
\( Fos \) = Factor of safety

|                      | \( \sigma_1 \) (Mpa) | \( \sigma_2 \) (Mpa) | Fos |
|----------------------|-----------------------|-----------------------|-----|
| Upright (At neutral caster) | 31.1                  | 0.036                 | 8.36|
| Upright (At positive caster of 6°) | 27.71                 | 0.036                 | 9.02|

Table 4: Factor of safety
14. RESULTS AND DISCUSSION

![Max stress acting point](image1.png)

We can see the max stress acting at the corner where the area of longitudinal and lateral forces applied are met, as shown in figure 16 and also we can see the maximum stress decreases when the transformation of angle takes place as the shear stress is developed in the component. Whereas rest of the area of component has a feasible stress which is near to the minimum stress obtained in the stress vector plot.

![Stress at different points](image2.png)
| Stresses at probes (MPa) | LocationX | LocationY | LocationZ |
|-------------------------|-----------|-----------|-----------|
| 3.8169                  | 11.370683 | 82.519946 | 9.599767  |
| 4.3803                  | 9.629474  | 82.002066 | 13.350872 |
| 6.2682                  | 10.525925 | 82.763561 | 3.099029  |
| 4.4129                  | 7.047330  | 81.974359 | 6.932136  |
| 2.7135                  | -8.104269 | 82.389498 | 19.559983 |
| 3.3396                  | -8.104269 | 84.288237 | 2.133707  |

Table 4 Coordinates of probe points

As the objective of the project is to compare analytical solution with analysis software solutions we can obtain approximate results at the surface where the forces is applied because of the reason we have obtained $\sigma_x = 3.83$ and $\sigma_y = 2.87$. Whereas the rest of the body they have different stresses at different points as the lateral and longitudinal forces are applied at certain area of the body and deforms non-uniformly.

Actually the comparison has few objections because the analytical calculations used to find the principle stresses are solved in 2D plane and are solved for infinitesimal element considering the same deformation takes place over the whole body. Ansys gives result based on the governing equation solved using finite element method.

REFERENCES

1. 2. "eCOURSES," [Online]. Available: http://www.ecourses.ou.edu/cgi-bin/ebook.cgi?topic=me&chap_sec=07.2&page=theory.

3. A. garg, "Fatigue Analysis and Optimization of Upright of a FSAE Vehicle," International Journal of Science and Research (IJSR), vol. 6, no.9, p. 6, 2017.

4. A. S. C. R. J. I. G. Gaurav Saxena, "Simulation and Optimization of wheel Hub and Upright of Vehicle: A Review," IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE), vol. 14, no. 1, p. 6, 2017.

5. "Build your own race car," 2016. [Online]. Available:https://www.buildyourownracecar.com/race-car-suspension-basics-and-design/2/.

6. "YOSPEED," [Online]. Available: http://yospeed.com/wheel-alignment-explained-camber-caster-toe/.

7. "come and drive it," [Online]. Available:https://www.comeanddriveit.com/suspension/camber-caster-toe.

8. Ó. K. Pétursson, "Uprights, wheel hubs and brake system for a new formul student racecar," reykjavik university, 2016.

9. D.M. William.FMilliken,Racecar vehicle dynamics, SAE international, 1998.
10. C.M.A.P. Z.S.J. W. William Kinkead, "Design and Optimization of a Formula SAE Vehicle," Worcester Polytechnic Institute, 2010.

11. M. Azmeer, "Design optimization of rear uprights for UniMAP Automotive Racing Team Formula SAE racing car," Journal of Physics, p. 6, 2015.

12. "Boston university ME," [Online]. Available: http://www.bu.edu/moss/mechanics-of-materials-stress/.