

Design and Weight optimization of wheel assembly components using FEA for BAJA

Rahul Sharma

B.Tech Automotive Design Engineering
University of Petroleum and Energy Studies
i.rahulsharma10@gmail.com

Raman Nain

B.Tech Automotive Design Engineering
University of Petroleum and Energy Studies
nainraman@yahoo.com

ABSTRACT

The BAJA SAE is an international series of events which attracts passionate engineers from all over the world. In India, the event is known as Mini Baja SAEINDIA which is held annually at NATRAX, Indore under the supreme organization of Mahindra. The event challenges young enthusiasts to design, fabricate and race the ATV buggies against the teams from top colleges in India. The general layout of the vehicle is a 4 wheel rear wheel drive system with engine provided by Briggs and Stratton. The vehicle undergoes rigorous tests including hill climb, suspension and traction, maneuverability, acceleration and a 4 hour endurance race.

The vehicle must be able to sustain all the loads that are generally encountered in an off-road scenario both static and dynamic. These loads are generally Impact loads which occur either due to a crash or jump. The designed vehicle is analyzed to ensure its durability under these circumstances.

General Terms

SolidWorks, Ansys WB etc.

Keywords

Hub, Upright, Disc, BAJA, ATV, FEA, Workbench, Modeling and Meshing.

1. INTRODUCTION

The wheel assembly of an ATV generally comprises of a Wheel rim attached to a hub and fastened using 4 bolts. Further the brake rotor is also mounted on this hub and through the hollow axis of this hub the spindle from the upright passes which supports the hub. The hub is mounted on the spindle using 2 roller bearings designed as per the requirement. The components procured for the BAJA vehicle are Douglas A5 rims with PCD holes at 156mm of 10mm dia. The hub and upright are designed custom to reduce the weight of the wheel assembly and provide appropriate strength to the wheel assembly components. The components are designed to ensure the safety of the driver and also look aesthetically pleasing.

2. MATERIAL SELECTION

To optimize the weight of the wheel assembly and ensure its durability, two different metals were considered for machining – EN-24 steel and Aluminum 7075-T6.

Table 1. Mechanical Properties of En24& Al 7075-T6

Material type	En 24	Al 7075-T6
Yield Strength (N/mm ²)	680	505
Tensile Strength (N/mm ²)	870	570
Mass Density (Kg/m ³)	7850	2810

Considering the above mentioned parameters, Aluminum 7075-T6 was selected the appropriate material for the machining of hubs and the upright.

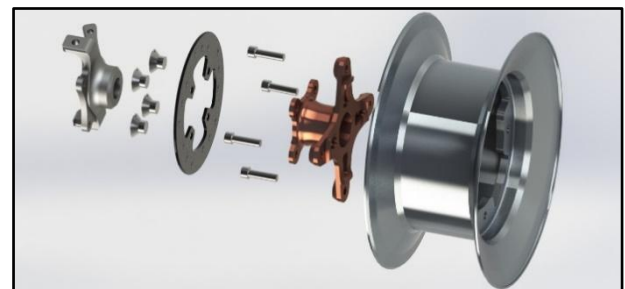


Fig 1: Exploded view of Front wheel assembly in SolidWorks.

3. DESIGN METHODOLOGY

3.1 Force Calculation

The Design of the hub and upright must ensure the durability of the components under crash loads, jumps, bumps and lateral acceleration. The vehicle was assumed to go over a jump of 6ft under a projectile motion and the force acting on the hub and upright during the time of landing was calculated and then later applied with two different cases. If the vehicle lands on only two wheels and other one is if vehicle lands on 4 wheels. The forces acting during a two wheel landing are more vulnerable and thus, they were considered as the base for design of these components.

Jump height = 6 feet = 1.83m

Mass of vehicle along with driver= 250kg.

Velocity at impact= $V^2 = 2XgXh$

$V = 6\text{m/s}$

Force on all 4 wheels during impact=

$$mv/t = 250 \times 6 / 0.10 = 15000\text{N}$$

If vehicle lands on 2 wheels, force on single wheel = 7500N

Force on the Assembly = 7500N $\sim 3g$.

3.2 PART MODELING & FEA

With its easy to use interface and high functionality during assembly, SolidWorks is the best software package available for part modeling of the hubs and uprights. These models were prepared using 3D modeling technique and appropriate material was applied to calculate their respective weight. However for the FEA, ANSYS WB 14.0 was used. The files were imported in .iges format and meshed using solid 3d elements.

4. DESIGN OF HUB

The vehicle is a rear engine rear wheel drive that indicates that the design of front and rear hubs will be different from each other as the rear hub has to support the driving torque from the axle half shafts through splines in addition to all the impact loads. The hubs also undergo the braking torque when the driver hits the brake pedal.

4.1 Front hubs

The front hubs provides mounting holes for both wheel rim and brake rotor. The rim fits with 4 holes PCD 156mm and 10mm bolt diameter whereas the brake rotor is drilled with 4 holes PCD 82mm with 10mm bolt dia. A central shaft connecting these mounting points is designed with grooves cut for inserting the roller bearings at the two ends of the hub.

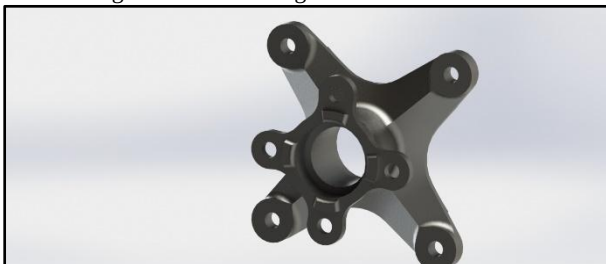


Fig 2: Front Hub modeled in SolidWorks.

After modelling of the hub in SolidWorks, the model was meshed in ANSYS WB. For the impact analysis, the central axial shaft was given a roller support and Force of 3g was applied on the mounting holes of the wheel rim in upward direction. Von-Mises stress gives the appropriate value of amount of stress induced in the component with respect to the force applied. For the simulation of the braking torque, the mounting holes of the brake disc were fixed using rigid support and the braking torque was applied at the mounting holes of the rim along the shaft axis.

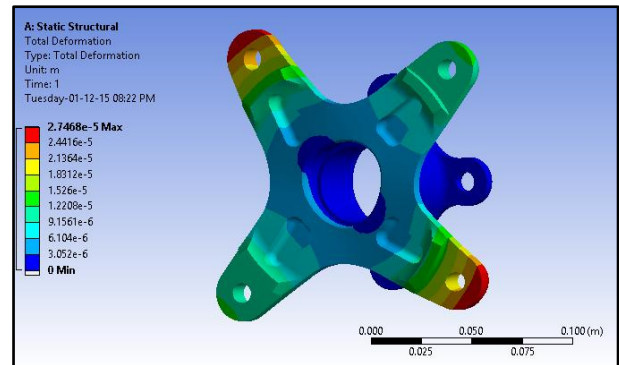


Fig 3: Max deformation in front hub using ANSYS WB.

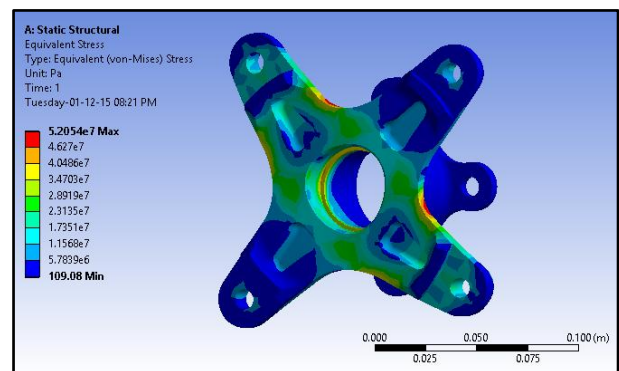


Fig 4: Max Stress in front hub using ANSYS WB.

4.2 Rear hubs

The design of the rear hub is different from the front hub as it is only supported by the splined shaft which is inserted in the rear upright. Although the mountings of disc and wheel rim remain the same.

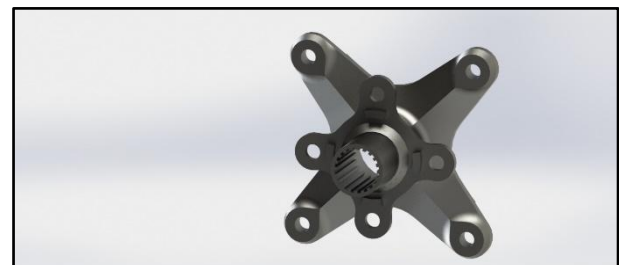


Fig 5: Rear Hub modeled in SolidWorks.

In addition to the impact load during a bump and the braking torque, the rear hub also supports the propulsion torque from the axle half shafts. This was applied to the inner splined shaft and the mounting holes of the wheel rim were fixed using rigid support.

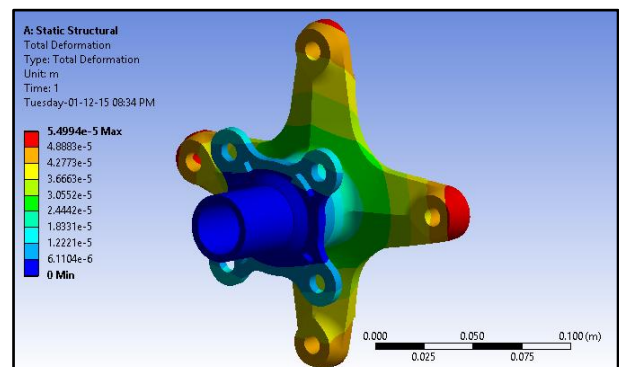


Fig 6: Max deformation in rear hub using ANSYS WB

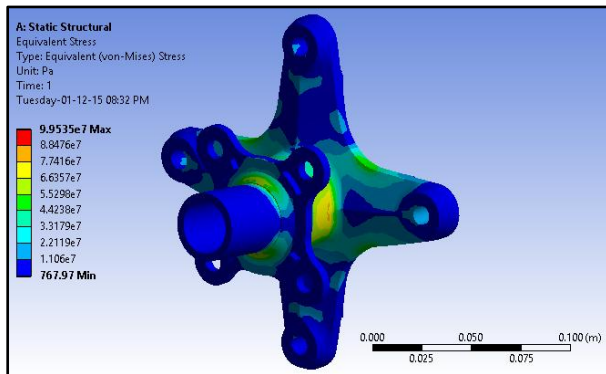


Fig 7: Max stress in rear hub using ANSYS WB

5. DESIGN OF UPRIGHT

The upright in the wheel assembly holds the rest of the components into their places. The front and rear uprights feature very different design approach as the front upright is a male type and rear upright is a female type.

5.1 Front upright

Front upright is connected to the wishbones using 2 ball joints and a spindle supports the hub internally with 2 roller bearings. Mounting points for brake caliper are also provided along with a steering arm as per the design of the steering geometry.



Fig 8: Front upright modeled in SolidWorks.

The part was analyzed in ANSYS by first fixing the spindle surface externally and applying the impact force of 3g on the two mounting points of ball joints in the upward direction.

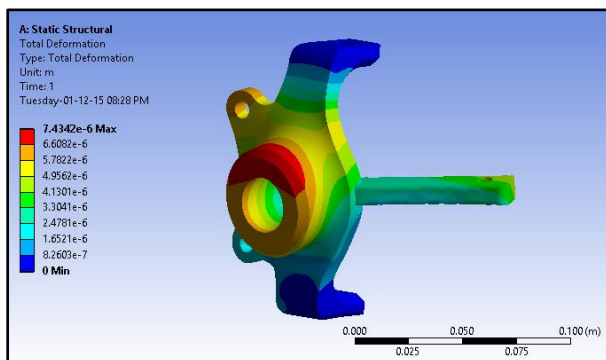


Fig 9: Max deformation in front upright using ANSYS

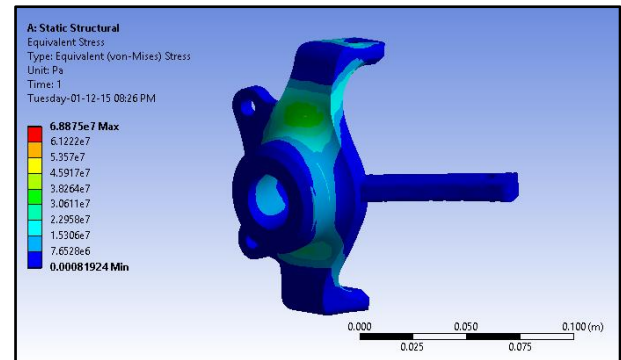


Fig 10: Max stress in front upright using ANSYS WB

5.2 Rear upright

Unlike the double wishbones at the front, the rear suspension system features a unique H-arm type linkage which requires 3 mounting points at the upright rather than 2. Heim joints are used at the rear to increase the durability and vertical travel smoothness. The design of the rear upright is a female type where the central cavity is provided to fit the bearing through which the hub and axle half shafts pass.



Fig 11: Rear upright modeled in SolidWorks.

For analyzing the upright with respect to the impact forces during the bump or jump landing, the central bearing case was fixed and the force of 3g was equally applied to the arms of the heim joints along the axis of the upright.

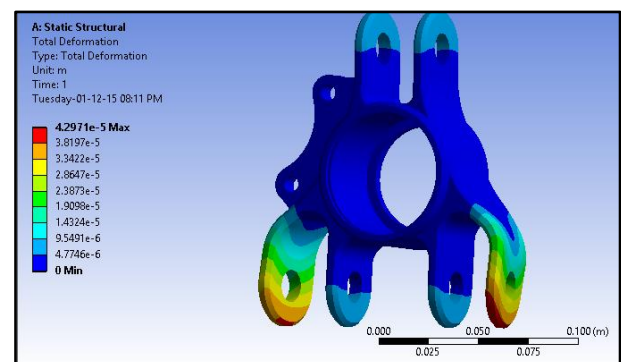


Fig 12: Max deformation in rear upright using ANSYS

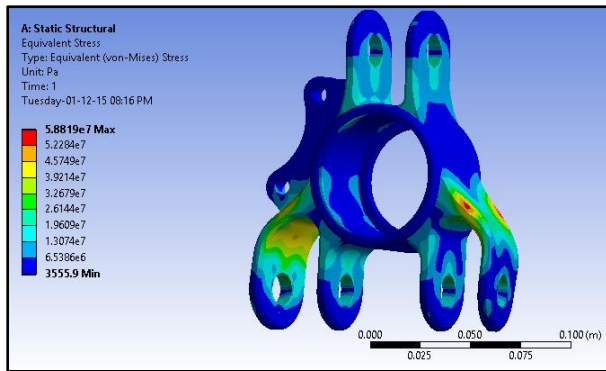


Fig 13: Max stress in rear upright using ANSYS

6. CONCLUSION

Case	Max Stress (MPa)	Max Deformation (mm)	FOS	Mass (gm)
Front Hub	52	0.025	9.7	440

Rear hub	62	0.055	8.14	425
Front Upright	69	0.007	7.3	380
Rear Upright	59	0.043	8.5	490

The Stress results in the above table clearly shows that the stress induced in the component is less than the yield strength with a suitable Factor of Safety. Thus the components can be considered safe for their implementation in BAJA vehicle.

7. REFERENCES

- [1] BAJA SAE INDIA Rule Book 2015.
- [2] R.K.Rajput, (2007). Strength of Materials, 4thEd.S.Chand Inc.
- [3] Nitin S.Gokhale,(2008).Practical Finite Element Analysis, Finite To Infinite.