Design of Upright

5. 2.1 Design parameters

Before designing of any components there are various parameter that are to be included in it. Irrespective of other details the main design parameters determine mostly the performance, adaptability with the environment, mates with the sub-component in an assembly, space occupancy etc. They are Special consideration and often are the constrains which are to be met.

The parameters that molded the design of the upright were:

1. Include castor angle of 6^o along the vertical axes of upright.
2. Project the brake caliper mounts at one side of upright.
3. Provide sufficient thickness to brake caliper mounts to endure sudden torque from the disk rotors.
4. Check alignment of the brake caliper mount on both of the uprights i.e. Left and Right uprights. Since the brake caliper doesn't have plane of symmetry along its center, brake mounts will have different spacial arrangements along the side of both uprights.
5. Steering arm mount be on the opposite side that of the brake caliper mount of the respective uprights.
6. Twin steering arm mount be provided to facilitate double shear for steering arm bolts.
7. Dual bolt holes will be provided to counteract the moment in the steering arm.
8. Have sufficient fillet radius throughout the design to minimize notch sensitivity.
9. Length of upright will be taken as per the suspension design that gave the optimum results.
10. Upper and lower wishbone mounting will be dependent on the kingpin inclination obtained from the suspension geometry
11. Bore be provided to accommodate the stub axle.
12. Press fitting tolerance to be provided in the central bore diameter to press fit the stub axle.
13. Sufficient wall thickness to make the component rigid, unsusceptible to external moment.
14. Design optimization will be done after the component is analyzed for various loading cases to relief weight.

5. 2 .2 Material Selection

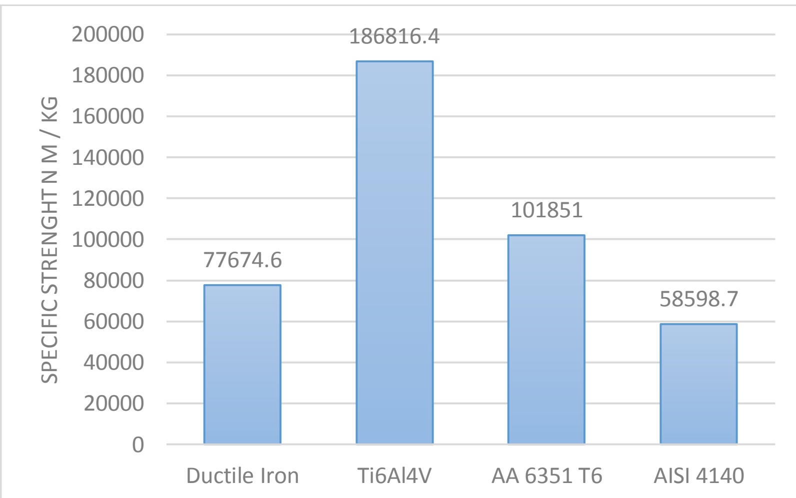
As it was mentioned above in design parameters, weight consideration was the main objective that dictated the choice of material for the suspension components. Also as mentioned earlier the preference of low unsprung mass in the vehicular system, it was necessary to opt the material which could bear the forces induced during the motion as well be light.

Market survey revealed the local availability of the following suitable material candidate for the component.

1. Cast Ductile Iron
2. Titanium Alloy Ti 6Al-4V
3. Aluminum Alloy AA 6351 T-6
4. Alloy Steel AISI 4140

Specific Strength comparison

It is the strength to weight ratio of the material. Strength here can be tensile or yield strength. A higher ratio dictates the material has appreciable strength compared to its weight.



The above graphs shows the usage of Titanium alloy to be most apt choice for the component. Nevertheless other factors also have be considered before material selection is to be made.

Cost per unit yield strength

Comparison also has to be made with the cost of the material. The cost comparison (C) is made by following equation;

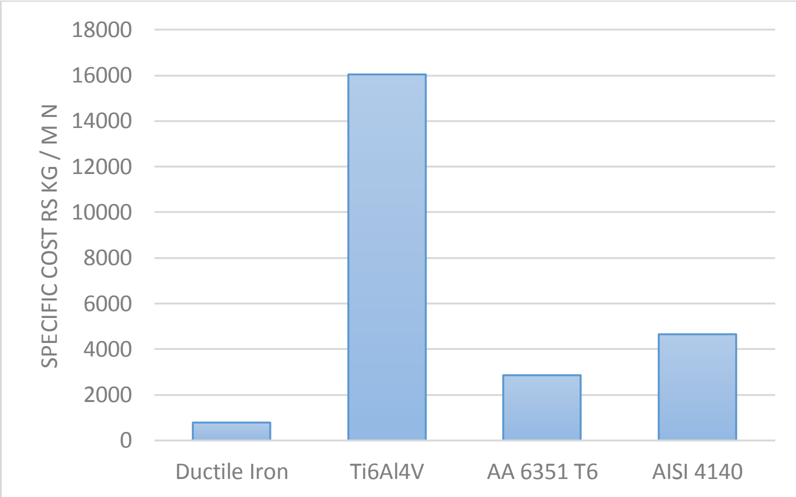
$$C = c_m * \frac{\rho}{\sigma}$$

Whereas,

C_m = Cost of the material per unit mass (Rs/Kg)

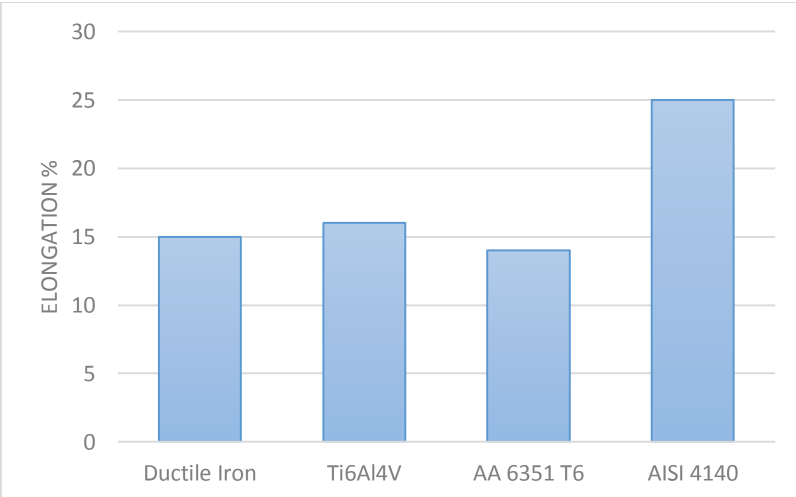
ρ = Density of the material (Kg / m$^{3)}$

σ = Yield strength of the material (N/m^2)



Ductility

Defined as the percentage elongation under stress before the material ruptures. This property is important as it gives a clear warning before the metal breaks down.



The above graph shows the cost value for unit strength of the material. It can be seen that Titanium alloy have the highest value for its strength whereas ductile alloy have the least.

Interpretations from above three detrimental property graphs;

1. Titanium and aluminum seems to be viable candidates where the strength to weigh ratios are to be considered.
2. Ductile cast iron is the cheapest material which is to be considered, also fact that most of the commercial automobile are fit with ductile cast iron uprights.
3. Titanium alloys have the highest specific cost among all

Considering all the above, it has been decided AA 6351 T-6 seems to be the most viable material for the component. The strength on weight ratio is sufficient to meet our standards. Choosing aluminum alloy also seems to be a most economical selection.

Since Aluminum alloy have been chosen, fatigue characteristics of haven to be taken into account.

Components subjected to fluctuating loading fail at much lower loads than their service loads due to fatigue. When not considered in the design stage it can lead to catastrophic failures in their service life. Factors causing a fatigue loading may be many which are:

1. Large fluctuation of Loads i.e. Stress amplitude of high magnitude
2. Sufficient large number of cycles of stress applied

Additional factors which may exaggerate fatigue failure are:

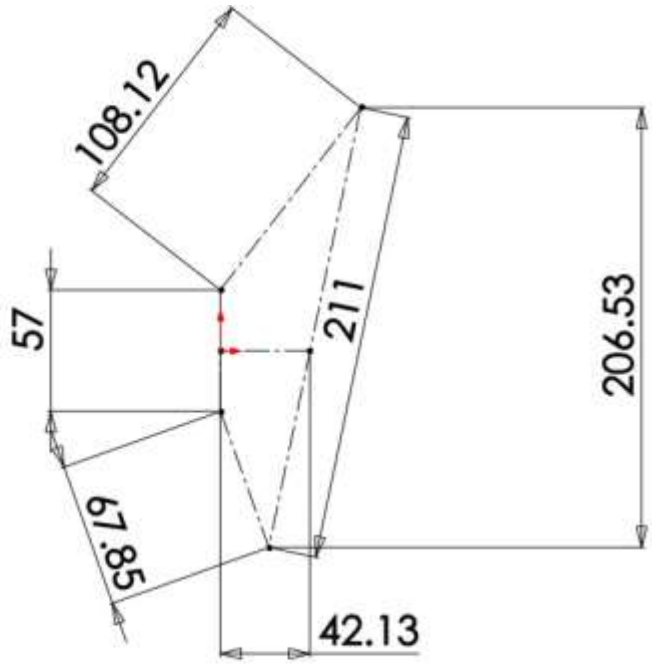
1. Corrosion
2. Residual stress
3. Stress concentration
4. Temperature
5. Surface finish
6. Stress range
7. Use of welding

The mechanism of fatigue failure can be simple put into two stages i.e. crack initiation and crack propagation to the point of static failure. This crack once formed begins to grow in each cycle of loading. Growth is also accelerated by higher amplitude of loading often characterized by multiple cracks initiated. Final catastrophic failure occur when a crack has grown to a significant length such that the next application of load results in static failure due to reduced area in that region. Fatigue cracks may start at various places such as at concentration of plastic strains, extrusion or intrusions of the surface, grain boundaries, internal voids and surface scratches.

Design methodology for any aluminum component used in car was to:

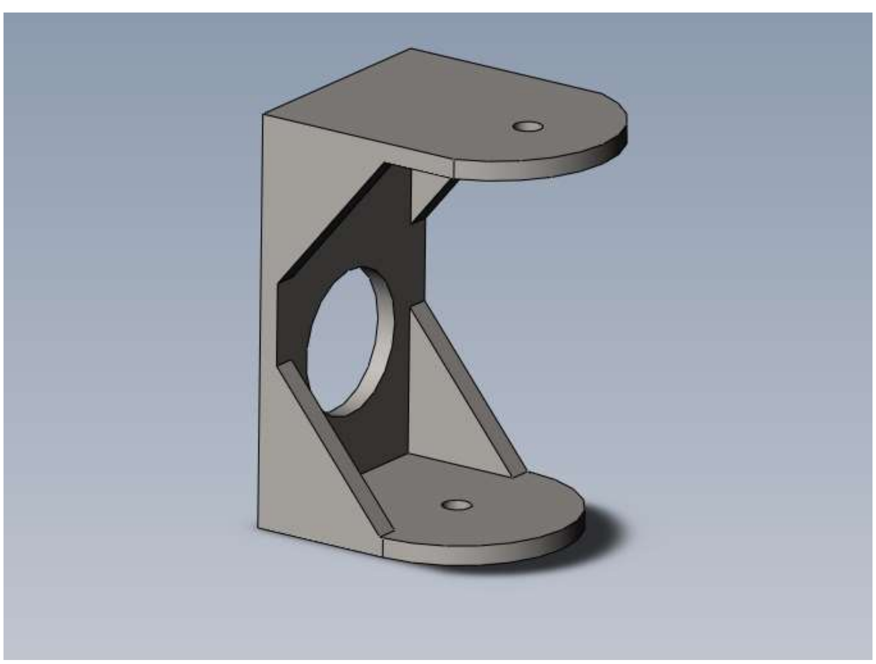
1) Load be highly approximated to actual condition which is being acted upon the component
2) Optimize the component so that the life of the component be under 10^5 cycles
3) Check for maximum stress under the stated endurance range instead of checking for yield criteria.

5. 2.3 Modeling



The uprights geometry determines the dynamic characteristics of the vehicle. Irrespective of the model, geometry is kept same in all the models. The models prepared were first based on manufacture feasibility, economic consideration and analytical sustainability. Various models were initially prepared for the same.

The following are the various models prepared and the reasons for disapproval of the designs.



Design-1 : Saruman

The first of the designs was initially desired to be made up of Steel plates. The plates were given the thickness of 10mm. The plates were countered cut with the required measurements and profile. Joining process was chiefly welding i.e. TIG (Tungsten Inert Gas) welding.

Adaptability of the Design :

- Cheap material cost
- Easy Manufacturing
- Less event completion time
- Provision of steering stopper

Disapproval of Design

- Weight of the component- 2.5 Kgs.

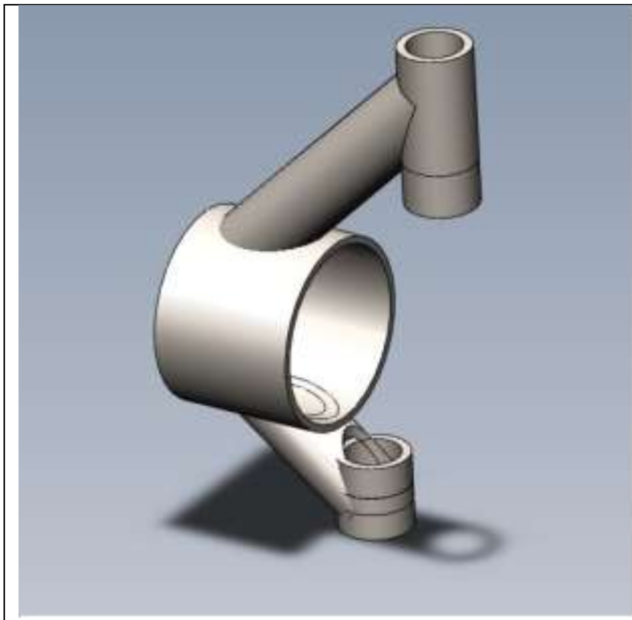
  Since high thickness of plates were being used the weight of the component was considerably high. Also use of 10mm thick sheet was indispensable as any reduction of thickness was detrimental.

- Low fatigue strength

  Since the plate were joined by welding process the fatigue strength of the component was considerably effected. This is because uncontrolled cooling and heating rates produced around the weld zone, also called as heat effected zones. This microstructural variation produces areas of high hardness whereby reducing strength.

- Steering upright geometry not accurate

  Since manual fabrication processes were being employed, the dimensional accuracies were uncertain.



Design-2 : Gollum

Mild Steel pipe of outer diameter 65mm and thickness of 2.5 mm was found to be adequate to press fit wheel bearing. The profiled Mild steel rods following the geometry was welded to the mother rim.

Adaptability of design

- Simple design
- Easy manufacturing
- Easy procurement of the materials required

Disapproval of design

- Intricate couping requirement.
- Welding reduces the fatigue strength and improper hardness in the sample.
- Strength not uniform due to alternate heating and cooling rates.
- Brake mounting arms are weak.

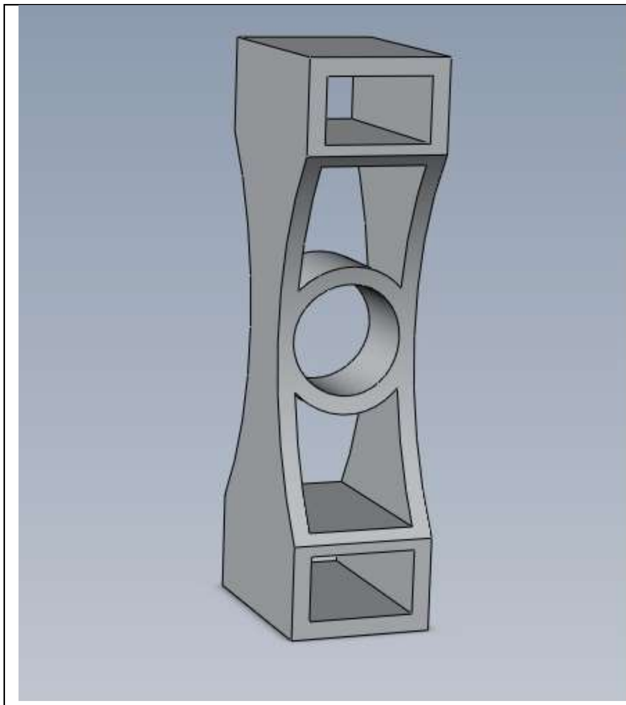
Brake mounting arms due to relative length from the parent pipe, became long and prone to failure

- Weight of the component

As mild steel tubes were used the weight of the component was measured up to 1.2 Kgs.

- Poor Strength

Finite Element analysis found it to be weak at weld zones. Hence prone to failure



Design-3 : Gandalf

Stock Aluminum alloy 6351 T-6 would have been used. Manufacturing process being CNC milling.

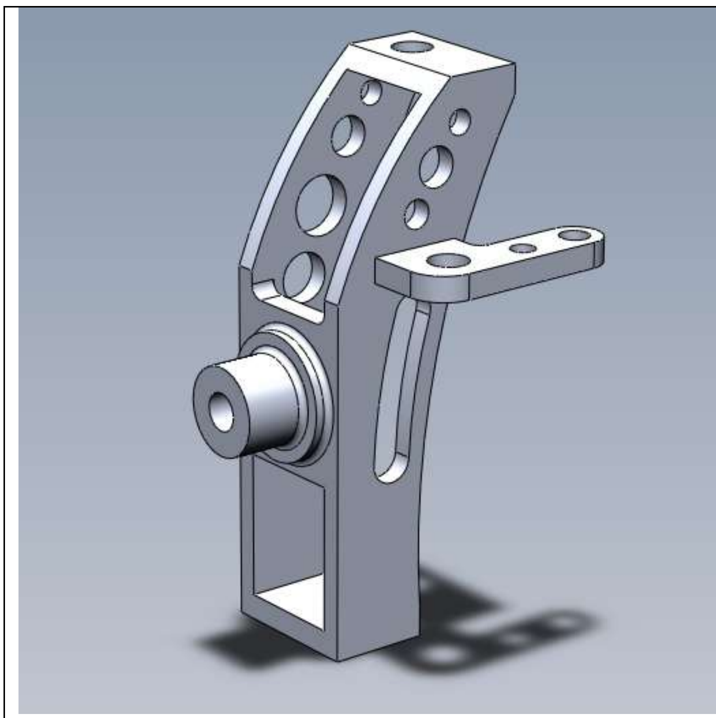
Adaptability of design

- Geometry of the upright was captured better
- CNC milling process was fast and accurate
- High torsional rigidity.

Disapproval of design

- Mounting points for upper and lower wishbone was too small whereby increasing stress in those areas.
- Increasing the upper and lower wishbone mount areas increased the weight of the component considerably.
- Weight of the component

  It weighs 2.5 Kgs comparable to its steel counterpart.



Design-4 : Bilbo

It was then decided to provide stub-axle on the upright itself. The aluminum Upright was then modeled to be CNC milled.

Adaptability of design

- Fast and accurate manufacturing
- Holes and slots provided to relief weight. It weighs only 760 gm. The least weight until now.
- Geometry captured accurately and effectively
- Bearing size is considerably reduced, whereby reducing weight.
- Bearing lock could be provided on built in stub axle

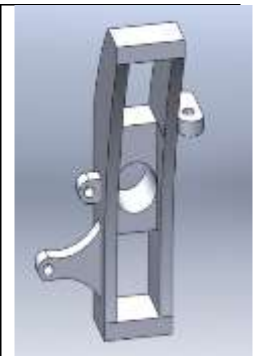
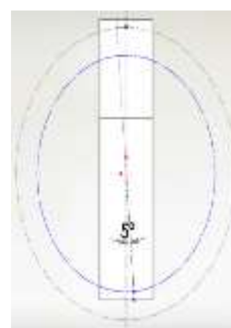
Disapproval of design

- Many relief holes increased notch sensitivity of the component
- Built in stub axle cannot have wheel bolt as threading on aluminum shears off easily.
- Required hub offset cannot be changed.
- In case of failure of stub axle whole upright have to be changed
- Longer stub axle created immense bending strength on upright
- Fillets and notches are requirement on the stub axle to be able to hold other components well. These fillets and notch sensitivity added up stress on bending stress, creating more stress it can hold
- Built in stub axle was failing in large loading conditions.
- Longer steering arm also made it more susceptible to shear from the upright
- Also built in steering arm dictates changing the upright when steering arm fails, the failure which is very common in the testing.

Rejection of all the other designs was valid as even though they all met with the requirement of designing parameters but failed at various other important suppositions. New design was required to have the modeling advantages of all other designs at the same time to rectify all the drawbacks of the previous designs.

The new and improved design was modeled in the following steps.

Step 1: The basic mold was created around the upright geometry. This captures almost all the features which were important to suspension design.	
Step 2 : The next step was to differential the upper and lower wishbone mounting positions. An arbitrary thickness was given which will be verified once the model is added to testing module. Care was also taken care to be able to provide sufficient room for accommodation of castle nuts.	
Step 3 : The brake mount position was carefully positioned so as when brake caliper in engagement would reach the end of the brake disk, for it to hold it firmly when under braking effect. The longer brake mount was given a parabolic profile to have a uniform strength also to reduce material. After calculated width of material was left for the center hole on the upright which would hold the stub axle. The extended length of the steering arm was reduced to decrease the added moment at the end of the upright.	

Step 4: After bail calculation considering the material, diameter and step dimensions for the stub axle. the central bore for the same was provided on upright	
Step 5: It was seen as an added strength to steering arm by placing its bolt under double shear. Also to counteract the effect of couple number of holes for its bolts were increased from 1 to 2.	
Step 6: The castor was given 5 degrees, according to which the position of upper and lower mounting points were determined. The hole diameter was kept equal to the rod-end diameter.	

The model was thus prepared with rectifying all the drawbacks obtained from the previous failures of various models, also every model was an improvement over the other until the final design (Hobbit) was considered to be selected.

The various dimension were kept variable in view of further analysis. Finite element analysis will dictates all the dimension of the component.

5. 2.4 Loading scenarios

Contemplation of actual force distribution on a single suspension component during motion of car can be quite complicated. To ease this complication and various complicated vector forces experienced by the component, it is usually split down into various individual scenarios. These scenarios take up the maximum force derived from calculation and constrains the model appropriately to represent the actual event.

The following scenarios of loading will be considered for the analysis

1. Steering effort
2. Brake mounting force
3. Remote loading
4. Cornering Force

It is to be noted that weight of the component was considered during analysis.

1. Steering Effort Calculation,

As steering effort was calculated in *section-12,* Steering effort (F_{st}) is taken as 1.5g

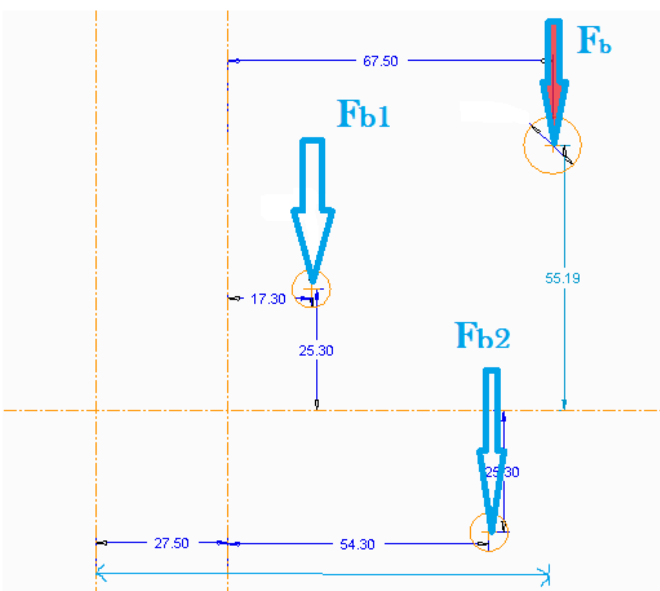
$$F_{st} = 4400$$

The tie rod being designed to take purely axial load, whereas such is not the case with steering arm. As the wheel travels the direction of forces changes. To accommodate this effect the direction of steering force wasn't taken parallel to ground but at an angle 45^{o} with the horizontal.

Then components along X and Y comes out to be equal i.e.

$$F_{st}\,x = F_{st}\,y = 3100$$

2. Braking Mount force calculation,



Radius of disk

The following free body diagram shows the position of brake caliper mounting pints with respect to center of the disk. This is because the brake mounting position is not symmetrical with the center of the caliper. And hence the force experienced by the each mount will be different.

Braking Mount Force (F_{bm})

$$F_{bm} = \frac{T_b}{r_b}$$

By sum of forces and applying moment at a point we get,

Force on larger brake mount = 3200 N

Force on smaller brake mount = 2300 N

3. Remote loading

Due to rim and hub offset, the bumps force is not a direct loading but an eccentric loading. The values of the mentioned dimension have already been calculated.

F_{shock} have already been calculated (6. c.vi) = 22,000 N

4. Cornering force

As per calculation the cornering force was taken to be 1.5g

$$F_{corner} = 5000$$

5. 2.5 Analysis

An upright is the crucial component as every force experienced by the car is passed on through it. Also analyzing every effect of this complex ever changing moment and forces becomes complicated. A worst case scenario's is therefore recommended where forces are scaled and restrains are applied in view of real time. The team performed various finite element computational analysis to match approximately with the actual conditions that will be experienced by the component so as to avoid failure in real-time.

There are many different types of analysis that must be completed to ensure that the part in question is able to withstand the applied loads. In addition, there are other factors that must be included in each analysis to ensure that the analysis itself is correct. In order to analyze the part correctly, the restraints must be an accurate representation of the real world scenario and the loads must be calculated for different loading scenarios.

Finally, the mesh must be as homogenous as possible. This would include minimizing the difference in aspect ratio between elements, as well as maximize element mapping quality. We must ensure that all of the meshes we use for the different components in our assembly are set up to be compatible with one another.

The main objective behind analysis was to check the maximum stress induced, predict the life of component and establish a suitable factor of safety in design.

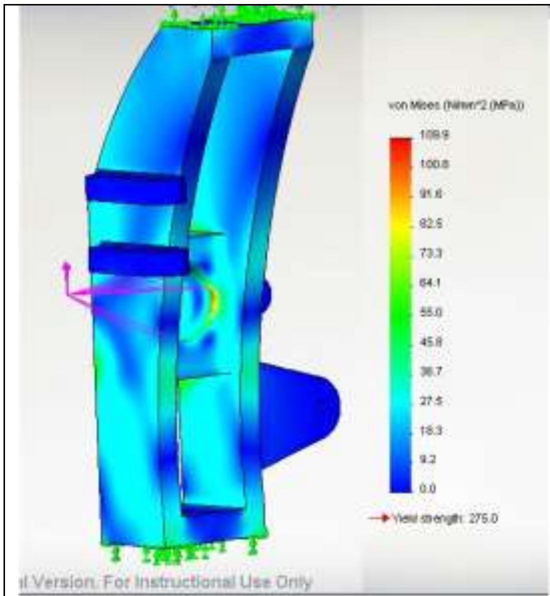
Since most of suspension component used were aluminum it must be noted forth that it doesn't exhibit a fixed fatigue limit unlike major steel categories.

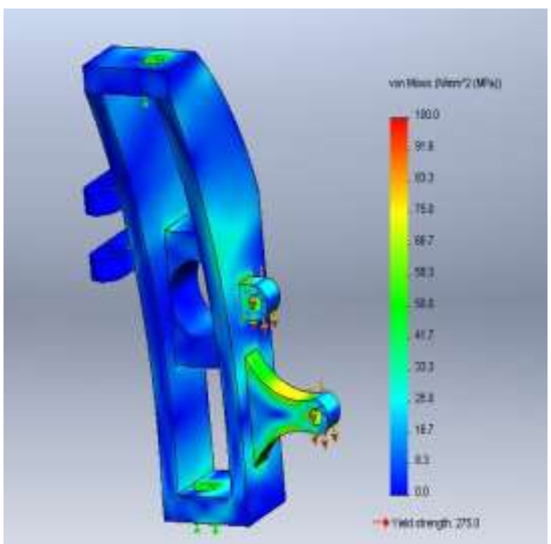
All of the analysis was done in student package of Solid works 2013 Simulation module.

The meshing package utilizes the tetrahedral mesh on the component. To give better accuracy the mesh size was made finer until the results became stagnant. Any more decrease in mesh size would just waste computer resource without marginally increasing any solution accuracy.

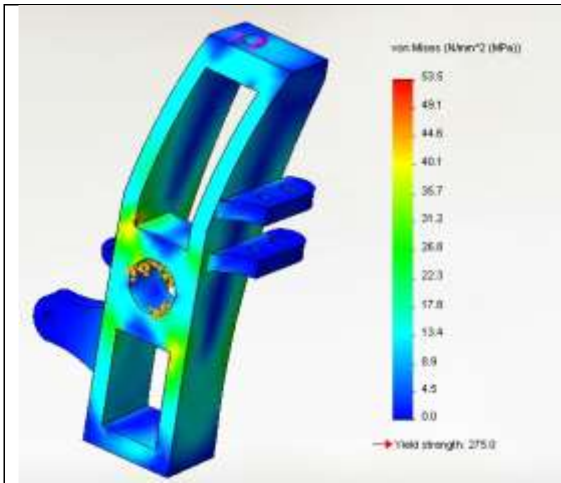
Modal analysis was also done on the component to determine how the system behaves in its displacement dynamic response. It was done to check how different frequencies naturally excite the system to a degree where resonance fluctuates through the component resulting in dropping of life expectancy of the system. For this study no external load is applied to the component, while it is known that external loads do not affect the natural frequency. The component was however restrained.

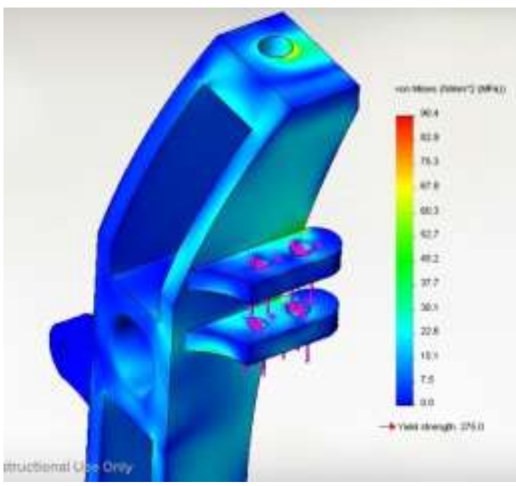
In all the analysis self-weight of the component was considered.

	Scenario	Bump loading
	Loading	Stub-axle bore at an offset of 50mm
	Constrains	Upper and lower wishbone mounting
	Force	22,000 N
	Maximum Stress	109 MPa
	Maximum Deflection	0.053 mm
	Factor of safety	2.42

	Scenario	Brake caliper mount loading
	Loading	Caliper holes
	Constrains	Upper and lower wishbone mounting
	Force	3200, 2300 N respectively
	Maximum Stress	100 MPa
	Maximum Deflection	0.3 mm
	Factor of safety	1.88

	Scenario	Cornering of vehicle
	Loading	Upper and lower wishbone mounting
	Constrains	Stub-axle bore
	Force	5000 N

	Maximum Stress	53 MPa
	Maximum Deflection	0.16 mm
	Factor of safety	4.91

	Scenario	Steering of vehicle
	Loading	Steering arm holes
	Constrains	Upper and lower wishbone mounting points
	Force	3000 N
	Maximum Stress	90 MPa
	Maximum Deflection	0.13 mm
	Factor of safety	2.73

The above plots of the results for various cases that might be experienced the component during its operation life cycle

Being the suspension component of an automobile, it is under constant non uniform loading condition. Also the loading condition vary drastically from terrain. In order to simplify the loading scenarios fully reversed loading i.e. R (Stress Ratio) = -1 was preferred over other stress ratios ranging from -∞ to ∞. The tabular values of S_a (Stress amplitude) with the corresponding life cycles [1] was fed into Solid works Fatigue Analyzer. The corresponding S-N diagram was obtained from the data.

Solid works doesn't have built in S-N graphs for all the materials that are present in its directory, hence it was needed to plug them manually into the software.

Following is the obtained fatigue data that has to be fed into the system to perform fatigue analysis.

Table 1: Fatigue data distribution for Aluminum 6061 T6

No of Cycles	Stress Amplitude (Zero Mean Stress) N/m^2
10.000000	482000000.000000
70.000000	482000000.000000
100.000000	420000000.000000
200.000000	325000000.000000
500.000000	241000000.000000
1000.000000	198000000.000000
2000.000000	168000000.000000
5000.000000	142000000.000000
7000.000000	135000000.000000
10000.000000	120000000.000000
20000.000000	99000000.000000
50000.000000	80000000.000000
100000.000000	71000000.000000
200000.000000	64000000.000000
500000.000000	58000000.000000

1000000.000000	55000000.000000
2000000.000000	53000000.000000
5000000.000000	51000000.000000
10000000.000000	50000000.000000
20000000.000000	49880000.000000
50000000.000000	49280000.000000
100000000.000000	48990000.000000
200000000.000000	48700000.000000
500000000.000000	48500000.000000
1000000000.000000	48400000.000000

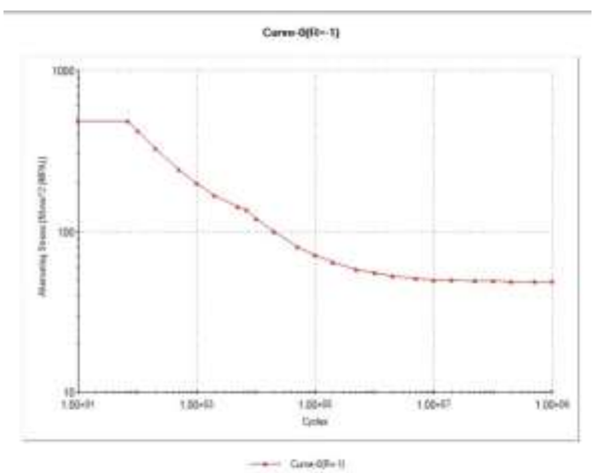
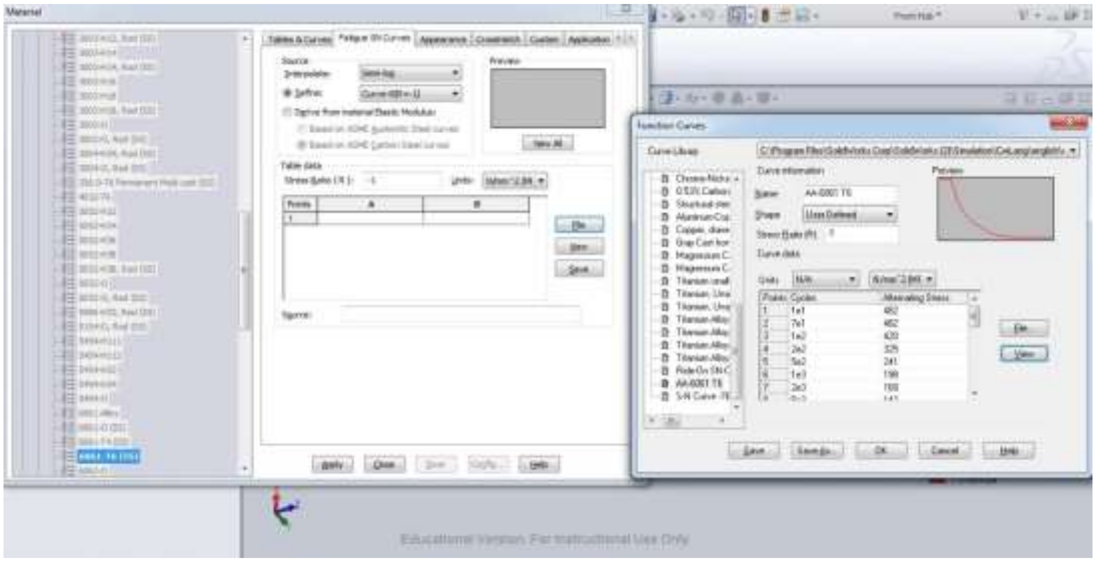
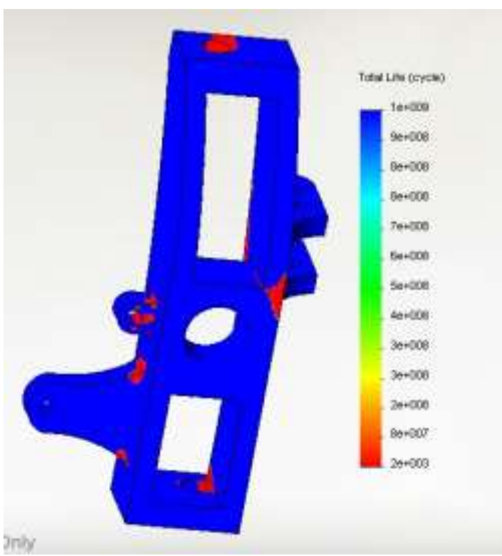


Figure 1: S-N curve for zero mean stress of Aluminum 6061 T6



The fatigue analysis could be done for individual scenarios however for a combined force situation it seems logical to optimize. For this bump force, heavy braking, steering pull were applied to component.



It can be seen the life of the component is well defined for more than 95% area having 1E9 cycles. Some of the areas shows a reduced life but that can be ignored potentially. All attempts were made to restrict the maximum von misses stress under 100 MPa, after studying the fatigue properties of aluminum. Also not compromising on weight gave the best combination of strength, weight and fatigue characteristics for the component.

5. 3 Design of Front Hub

5. 3.1 Introduction

A wheel hub is a mounting position for wheel of the vehicle, it houses the wheel bearing as well as supports the lugs and brake disk. It can either transmit power or be just rolling. Its function is basically to keep the wheel spinning freely on the bearing while keeping it attached to the vehicle. Designing a hub is very crucial as it alones is the interface between the wheels and rest of the vehicle.

Lug bolts are usually integral to hub, hence these are also called as locking bolts.

Spacers are also used to fit between the hub and the brake disks. This is done to accommodate different brake calipers as to avoid the scraping between them and the calipers.

Usually in hubs in commercial vehicle are made up of alloy steels or cast iron. But for a mini Baja vehicle it is advantageous to look at alternative material to make it light weight.

5. 3.2 Bearing selection

Selection of suitable bearing for a particular purpose is immensely important in view of the load it is meant to take at given rotational speed giving a certain life.

Max gear ratio of the transmission,

$$7.6$$

Max rpm of the engine crank,

$$3800\ rpm$$

The maximum rotational speed attained by the tire is,

$$500\ rpm$$

Assuming the life of the bearing to be designed for is,

$$1000\ hours$$

Loading ratio (From data book)

$$\frac{C}{P} = 3.11$$

Axial Force is assumed to be during cornering, which is taken as 1.5g (Pr)

$$2500\ N$$

Radial Force is the drop weight of the car (Pa)

$$4000\ N$$

the ratio of axial to radial,

$$\frac{P_a}{P_r} = 0.45$$

for the corresponding ratio, the value of equivalent load

$$P = S(XP_r + YP_a)$$

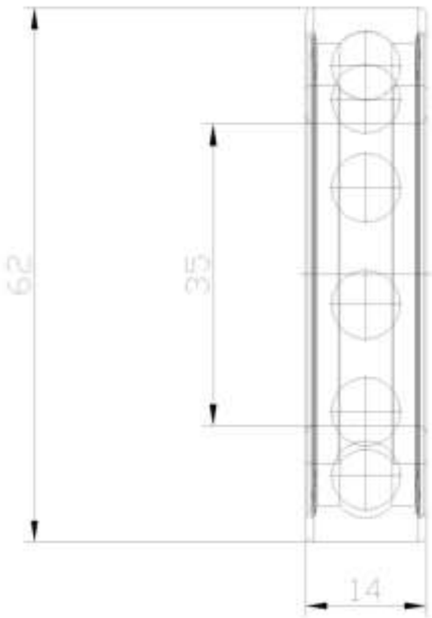
$$P = 4000\ N$$

The dynamic load carrying capacity was found out to be,

$$12995\ N$$

For the given life and load rating the bearing number SKR 6007 was chosen for the front wheels.

The given are dimensions of the bearing



5. 3.3 Designing Parameters & Considerations

Following are the Parameters which guided the design of Front Hubs,

1. Pitch Circle Diameter of Lug Bolts on rim

Since stock Maruti rims were chosen to be on the vehicle on all four wheels. In order that the hub to sit on rim, the pitch circle diameter of the rim had to match with the designed hubs.

Pitch Circle Diameter of Lug bolts on hub = 144 mm

2. No of Lug bolts and their size

Since the rim had 4 equi-spaced lug bolts holes. With the hole size of 12.5 mm diameter. The holes to match with rim had to be provided on hub.

4 Lug Bolt holes with diameter = 12 mm ø

3. Bearing Provision

Since as mentioned earlier the bearing selected was SKF 6007. The bore on the hub with sufficient tolerance is to be provided for the bearing to sit inside the hub.

The bearing bore on hub = $62\ ^{0}_{-0.03}$

4. Bearing Seat

For able to lock the bearing in the hub on one end it should have bearing seat. The thickness of seat must be enough to withstand axial force while cornering.

5. Common Holes for Rim and Brake disk mounting

Common holes for both rim and brake will minimize the number of holes from 8 to 4 on the hub.

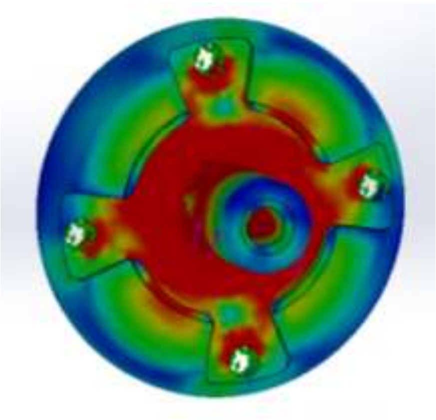
6. Bearing Lock provision

On other end of the hub, the bearing will be locked by an internal circlip. A groove of 2mm is to be provided to facilitate the circlip.

5. 3.4 Modeling

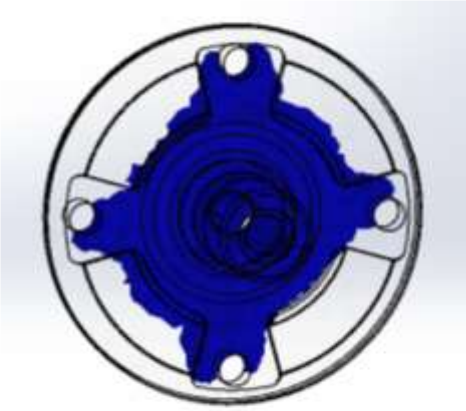
A literature survey was undertaken before modelling of the component began. Initially as a reference the stock Maruti hub was taken. The model was loaded in the Solid-works. Initially it was decided to use it but due to its weight idea was soon dropped. Also component was analyzed to develop a lighter and stronger equivalent in aluminum alloy.

The details in the model were removed to simplify the analysis. The brake mounting holes were also removed to see the effect of analysis. Since the goal of the analysis will be to target potent areas of weight reduction and geometry changes to suit the Baja vehicle. Such method reduces the chances of failure of design as the adopted design is commercially utilized. Also as mentioned in design parameter stub axle will be eliminated in the front hubs to accommodate hub bearing which in turn will hold the axle.



An arbitrary force of any value have been loaded on the stub axle of the hub in context. The constrains were the lug holes. The plot shows the stress distribution close to stub axle extending towards lug holes. It was also evident that rest of the area experienced a relatively less induced stress.

A plot of design insight reveals the observation to a scaled level.



The study of stock hub gave us the insights to new design of the hubs that would be precisely follow its design guidelines.

Following is the timeline of the model of the component.

The blank is initially made and stepped to host the wheel bearing. Provision is also given to lock the bearing in the other direction by a bearing seat.	

	
The Pitch circle diameter equivalent to rim is kept on the hub.	
A single chuck of material is removed to observe the effect.	
The pattern is then repeat around to obtain optimized wheel hub. The grove for the circlip is provided to lock the bearing in opposite direction. Hence modelling is complete.	

5. 3.5 Loading scenarios

Following are the various loading that were applied to hubs for analysis. This of these forces so obtained will help the prediction of actual results.

1. Drop test

This test try to replicate stress produced when the vehicle falls from a certain height. To obtain the impact load on the hubs,

Assuming the vehicle falls from the height (h),

$$h = 1\,m$$

The impact velocity (v) of the vehicle is,

From kinematic relation for rectilinear motion

$$v\ = \sqrt{(2 * g * h) - u^2}$$

whereas,

g = acceleration due to gravity (9.8 m/s^2)

u = Initial velocity of the vehicle before the fall (0 m/s)

Substituting the above value,

$$v\ = 4.42\ m/s$$

And now for calculating the impact force (F)

From Work-Energy Principle,

Change in Kinetic Energy of the object = Work done on the object

$$\frac{1}{2}\,m\,(\,v^2 - u^2) = F * d$$

whereas,

d = the compression of the shock springs (0.15 m)

Thus,

$$F_{shock} = 22{,}800\ N$$

2. Braking Torque Test

This test analysis the effect of panic braking on wheel hub. Since brake disk and hub are directly connected, while under braking, brake disks induces opposite torque on the hub to halt the vehicle.

Assuming the vehicle comes to a complete halt from 55 Km/h in a distance within 6m on an off-road track.

Initial velocity (u) = 15.2 m/s

Braking distance (d) = 6m

Deceleration (a_d),

$$v^2 = u^2 + 2 * a_d * d$$

whereas,

v = final velocity

thus,

$$a_d = 11.7\ m/s^2$$

Assuming weight distribution is 50:50, Force on Front wheels (F_f),

$$F_f = 2047.5\ N$$

Braking torque on front wheels (T_f) is,

$$T_f = F_f * R_t$$

whereas,

R_t = Radius of Tire (0.3048 m)

Thus,

$$T_f = 624\ Nm$$

3. Cornering & Skidding

Slip angle changes at the turn of the vehicle, sometimes amateur driver may fail understand its significance and it results in vehicle skidding in turns.

It has been taken as 1.5g force for skidding whereas 1g for cornering force. As speed of the vehicle is restricted the assumed values holds good.

Cornering Force = 3500 N

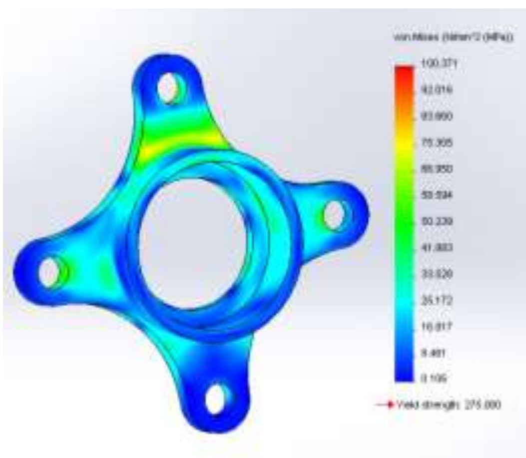
Skidding Force = 5200 N

4. Rim fracture

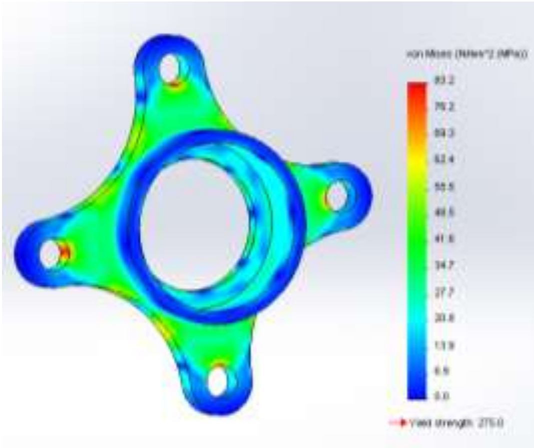
In the event of a rim failure, i.e shear of rim across the lug bolts or failure from flange, it produces an eccentric loading at the hub due to its rim offset.

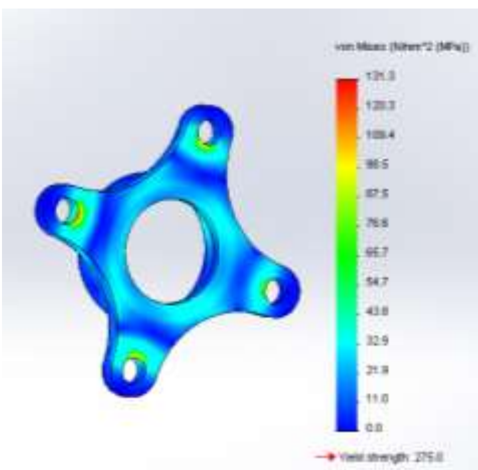
To take this into account the self-weight of the vehicle will be taken in consideration.

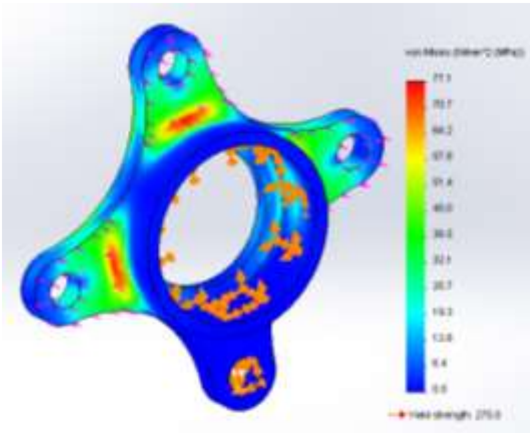
5. 3.6 Analysis

	Scenario	Drop Test
	Loading	Central bore
	Constrains	4 Lug holes
	Force	11,000 N
	Maximum Stress	100 MPa
	Maximum Deflection	0.041 mm
	Factor of safety	2.5

	Scenario	Braking Test
	Loading	Central bore
	Constrains	4 lug holes
	Torque	700 Nm

	Maximum Stress	83 MPa
	Maximum Deflection	0.022 mm
	Factor of safety	

	Scenario	Cornering & skidding
	Loading	Skidding-Bearing seat Cornering-Central bore
	Constrains	4 Lug holes
	Force	Skidding-5200 N Cornering-3500 N
	Maximum Stress	131 MPa
	Maximum Deflection	0.049
	Factor of safety	1.96

<table>
<tr><td rowspan="7"></td><td>Scenario</td><td>Rim fracture</td></tr>
<tr><td>Loading</td><td>3 spokes</td></tr>
<tr><td>Constrains</td><td>Central bore</td></tr>
<tr><td>Force</td><td>4000 N</td></tr>
<tr><td>Maximum Stress</td><td>77 MPa</td></tr>
<tr><td>Maximum Deflection</td><td>0.11 mm</td></tr>
<tr><td>Factor of safety</td><td>3.2</td></tr>
</table>

To optimize the component even further it was then added into Solid works Optimization.

The variable that was kept in the study was the thickness of the blank of the hub. Following results were found,

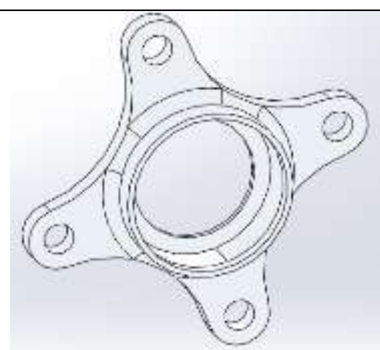
Component name	**Units**	**Current**	**Initial**	**Optimal**	**Scenario1**	**Scenario2**
thickness	mm	8	8	4	4	8
Stress1	N/mm^2 (MPa)	85.493	85.493	172.74	172.74	85.493
Mass1	Kg	0.151107	0.151107	0.115572	0.115572	0.151107

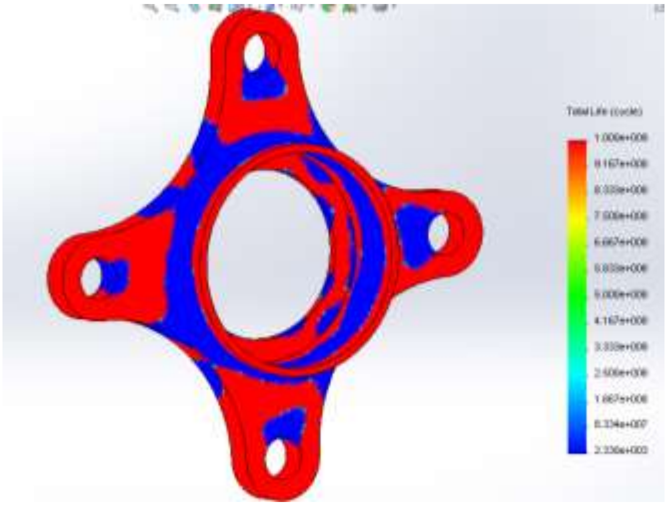
It can be seen that reducing the thickness from 8 mm to 4 mm would increase the max stress in the component to 172.74 MPa which is fairly under the limit. The weight of the component could also be reduced by 25%. But the optimization wasn't carried out in the fabrication due to reduced fatigue effects.

Component name	Units	Current	Initial	Scenario1	Scenario2	Scenario3	Scenario5	Scenario6
bearing seat thickness	mm	4.5	4.5	2.25	4.5	2.25	2.25	4.5
seat thick	mm	5	5	2.5	2.5	5	7.5	7.5
Stress	N/mm^2 (MPa)	131.28	131.28	150.3	128.02	155.78	160.57	135.51
Mass	kg	0.151107	0.151107	0.119011	0.137345	0.129541	0.14007	0.16487

The optimization was however carried to determine the optimum thickness of bearing seat and thickness center bearing tube. Based on above optimization it was seen to reduce the weight by 10%, the thickness of the bearing seat thickness as reduced to 2.5 mm from 5mm.

And hence after optimization the results were sent for fatigue optimizations. To check the life of the component, a combined study was prepared to sum up all the forces in different scenarios. This being done gives us the combined fatigue analysis also saves the time for each individual iterations.

Optimizes component Weight = 132 grams Seat thickness reduced by = 50%	



Since it was assumed the vehicle to be running 1000 hours at constant speed of 60km/h. The predicted cycles each component has to run is calculated to be 2.8 E8 cycles.

The above fatigue plot shows various portion of the component actually close to required value of cycle it was prescribed for. Hence the fatigue analysis were in good agreement with the fatigue design parameters.