Fatigue Analysis and Optimization of Upright of a FSAE Vehicle

Ayush Garg

Manipal Institute of Technology, Department of Mechanical and Manufacturing Engineering, Manipal University, Udupi-576103, Karnataka

Abstract: ***Formula racing car components experience immense mechanical loads which keep varying all through the life span of the components. It is essential to know the life of a component and replace it in time to ensure performance, reliability and durability of the vehicle. This paper addresses the fatigue life analysis of the Uprights for a Formula SAE vehicle and optimization of the design based on the results to increase the life. The front and rear uprights have been designed using the 3D modeling software CATIA and in accordance to the formula SAE rulebook 2018. The analysis parameters are derived from the logics and simulations of the vehicle design parameters using Optimum Dynamics. The study covers both static and fatigue analysis of the components with their respective boundary conditions and the analysis software used to carry out the static and fatigue analysis are Ansys Workbench and Ansys nCode respectively. A comparison on the results obtained from the static analysis and fatigue analysis is made and benefits of fatigue analysis over static analysis are shown.***

Keywords: fatigue analysis, fsae upright, nCode, stress life, fsae

1. Introduction

1.1 About Formula SAE

Formula SAE is a student design competition organized by the SAE International. The event is conducted in various countries all over the world all through the year. The Formula SAE vehicle is conceptualized, designed and fabricated by the students enrolled in a university. The design of the vehicle must comply to the rulebook released by the respective competition.

1.2 Uprights

The uprights are the central component for the suspension of a Formula SAE race car. All suspension components including the control arms, steering arms, springs, shock absorbers, brakes, tires, and in the case of the rear upright the axles are connected to the uprights. All forces that the car will encounter due to the road and tyre interaction will go through the uprights. The uprights must be sufficiently strong in order to withstand many of these forces occurring simultaneously, as well as any forces that may happen as a result of a crash or other kind of emergency without failure. Any failure of the uprights would render the car un-drivable.

1.3 Objective

Static stress analysis is generally used by students all around the world to analyze the design of the component but it does not give the complete picture of the performance of the component. Pushing the limits is necessary while designing a race car and reducing the safety factor of the design is generally the way adopted. The problem with the methodology is that, it is not possible to compensate for all the effects and phenomenon that could result in the failure of the component while setting the value of the safety factor and unexpected failures of components can occur. On the other hand Fatigue analysis based on the stress life approach using the variable amplitude loading simulated for the application could predict how long the component will function with the forces acting on it and helps the designer to replace the component before catastrophic failures occur. Any optimization to the design based on the results of fatigue analysis could lead to a improvement in the life of the component which adds on to the quality of the design. Thus, a study for prediction of fatigue life of the components was done. The parameters used to obtain the results of the analysis are obtained by the respective calculations and analysis conducted by the team members of Formula Manipal, the official Formula Student Team of Manipal University.

2. Literature Review

The stress life approach has been the standard approach for many decades now and is still widely used. This approach is used for the components that operate in the elastic ranges of the material and have long lives, i.e they are considered as high cycle fatigue [2]. Although there are some drawbacks associated with it such as considering all strains as elastic and not taking the plastic behavior of the material in consideration, it is still regarded as the most effective as supporting data for this approach is abundant.

In real case scenarios, the mechanical components of race cars are subjected to variable amplitude loading and the loading pattern can not be predicted with a characteristic equation. Thus, several methods have been developed over the years to take in account the variable amplitudes and using the data generated by physical tests of constant amplitude to predict the life and damage[2].

A linear damage cumulative rule known as the Miner's rule is the most commonly used theory for cumulative damage calculation [7]. This rule is used to calculate the damage on the component due to the variable amplitude loading.

3. Cad Model of the Design

The uprights were designed using the 3D modeling software CATIA. Numerous iterations were made to obtain the design shown below on the basis of the results obtained on static analysis for Von Mises stress, stress flow and maximum deformation. The uprights designed are required to have a high stiffness to resist the camber changes in the wheel assembly.

Figure 1: Cad Model of Front Upright FMX6

Figure 2: Cad Model of Rear Upright FMX6

4. Material Selection

The material selected for the component is Aluminum Alloy 7075-T6 as it has a high strength to weight ratio and better fatigue properties than the other lightweight Aluminum Alloys such as 6061-T6.

The properies use for the analysis were as follows:

Table 1: Material Properties of Al 7075-T6

Properties	Al 7075-T6
Density	2.793 g cm^{-3}
Young's Modulus	71 Gpa
Poisson's Ratio	0.33
Bulk Modulus	69.62 Gpa
Shear Modulus	26.69 Gpa
Tensile Ultimate Strength	565.37 MPa

The log-log plot of the alternating stress vs number of cycles used for the analysis is as follows:

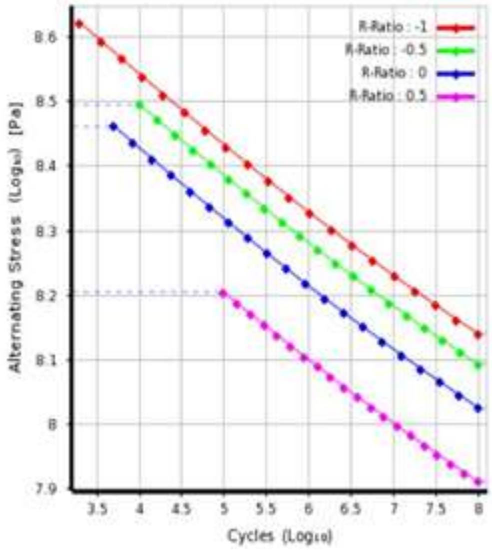

Figure 3: S-N Curve of Al 7075-T6 (nCode Material Library)

5. Static Analysis

5.1 Static Loads and Boundary Conditions

The parameters of the vehicle used for the calculation of forces for the static analysis of the front and rear uprights are:

Mass of the Car and Driver = 290kg
Weight Distribution = 40/60
Height of Center of gravity = 275mm
Wheel Base = 1540mm
Track Width Front = 1200mm
Track Width Rear = 1100mm
Deceleration = 1.8gs
Lateral Acceleration = 2gs
Longitudinal Acceleration = 0.9gs
Bump force = 1000N
Wheel Diameter = 457mm

The loads on the tire in the x,y,z co-ordinates were calculated using the following formulae [3].

$$Load\ Transfer = \frac{Weight \times Acceleration \times CG\ height}{Track\ Width\ or\ Wheelbase} \quad (1)$$

$$ad\ in\ x = Longitudinal\ Acceleration \times Load\ on\ wheel \quad (2)$$

$$ad\ in\ y = Lateral\ Acceleration \times Load\ on\ wheel \quad (3)$$

$$ad\ in\ z = Load\ due\ to\ load\ Transfer + Bump\ force \quad (4)$$

$$aking\ moment = Load\ in\ x\ during\ deceleration \times \frac{Wheel\ Diameter}{2} \quad (5)$$

The loads calculated for the Front and Rear uprights are as follows:

Table 2: Calculated Loads for the front and rear uprights

Maximum Loads	Front Upright	Rear Upright
Tire X	1848 N	1737
Tire Y	-2052 N	-2164
Tire Z	2026 N	2028
Braking Moment	422075 Nmm	397078 Nmm

The assumption to carry out the analysis was that the wheel has traveled the maximum it can at the maximum load case due to bump load. The upper and lower ball joint mounting points and the tie and toe rod mounting points were given as cylindrical supports with only tangential movement free. The von mises stress and total deformation models were solved.

5.2 Static Analysis Results

The results of the static analysis showed that the factors of safety for the front and rear uprights were 1.13 and 1.26 respectively. The maximum deformation for the front upright was 0.59587mm and for the rear upright was 0.33695mm.

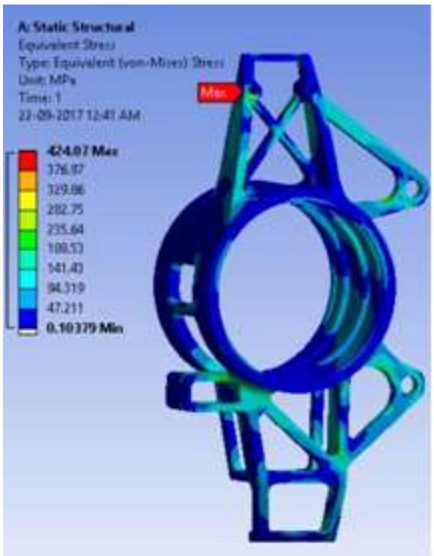

Figure 4: Static Stress analysis result for front upright (Maximum=424.07Mpa)

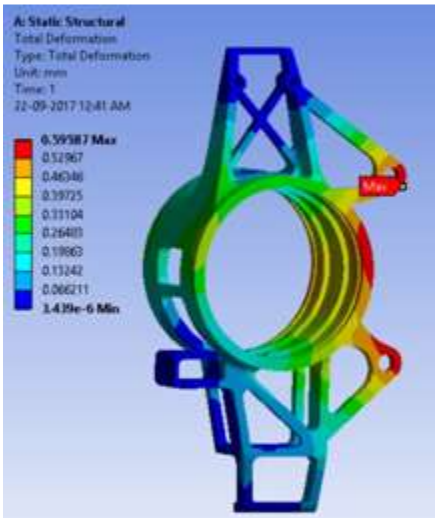

Figure 5: Total Deformation analysis result for front upright (Maximum=0.59587mm)

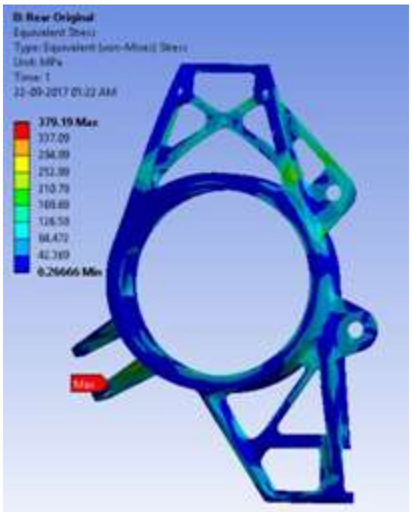

Figure 6: Static Stress analysis result for rear upright (Maximum=379.19Mpa)

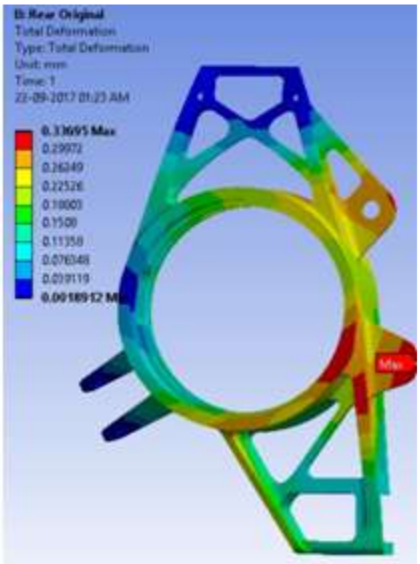

Figure 7: Total Deformation analysis result for rear upright (Maximum=0.33695mm)

6. Fatigue Analysis

6.1 Fatigue Analysis Loads and Boundary Conditions

For the fatigue analysis, the data of the loads on the tire for a time period was needed. This was found out by using the parameters of the vehicle with the tire data model as an input in Optimum Dynamics. The track data of the Formula Student Germany Endurance was used as the track input for the loads on the tires. The results of the simulation were used to create a load history for the uprights to be analysed.

To carry out the analysis, the model was broken down into fourload cases for the tire X,Y,Z forces and the braking moment for the brake caliper mounting position. In each of the load cases, a standard value of load was used as an input and the corresponding values in the load history were taken as its ratios. The load history obtained after taking the ratio is called the load ratios which were used as an input in the time series glyph of Ansys nCode. The load channels were mapped to the load cases made. The material properties were changed to machined to account for the reduction of fatigue life due to the machining processes. Each of the cycle in the analysis refers to a lap of the track for which the forces have been computed. Each of the laps has 5629 loading cycles.

The boundary conditions for the mounting points remained the same as for the static analysis. The SN Method setting used was Multi R-Ratio Curve, the Mean Stress Correction method for damage calculation was set to Interpolate.

Figure 8: Load Cases made as separate static structural modules connected together with a SN Time Series module of Ansys nCode

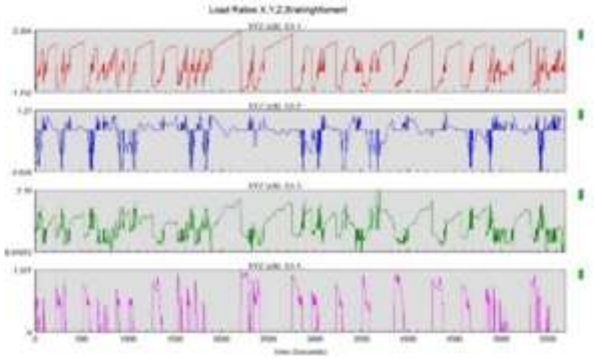

Figure 9: Load Ratios used for the Fatigue Analysis of the Uprights

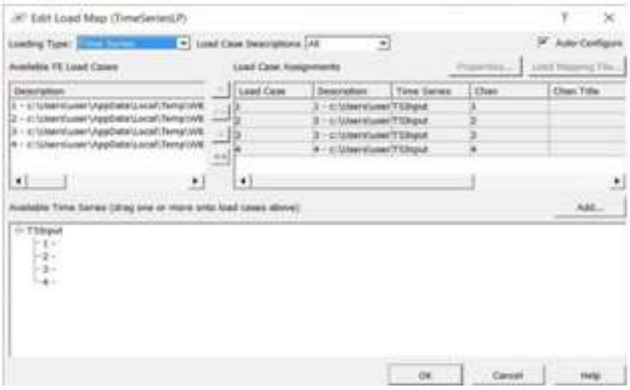

Figure 10: Load ratio channels were mapped to the different load cases

Figure 11: Assignment of the Surface Roughness type as Machined

Figure 12: SN Engine configuration of the Fatigue Analyis

6.2 Fatigue Analysis Results

The fatigue analysis was conducted for the designs using the boundary conditions and loading mentioned above. The minimum life results for the front and rear uprights were 365.9 cycles and 1165 cycles respectively. As each of the cycles contains 5629 loading cycles, the life of the component is still in the high cycle fatigue range and stress life analysis is thus valid. The maximum cumulative damage results are also shown below.

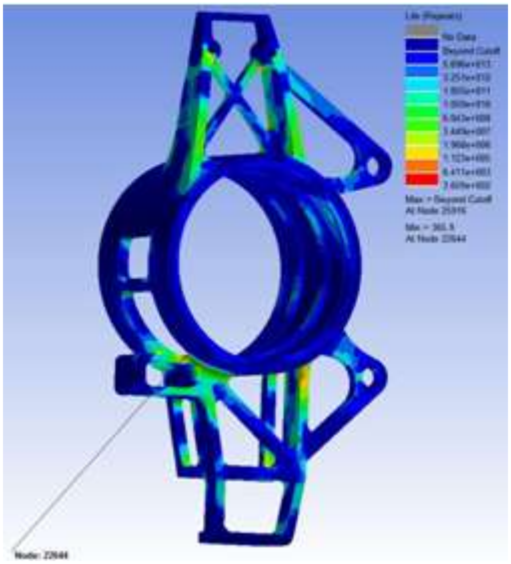

Figure 13: Stress Life of Front Upright (Min:365.9 cycles)

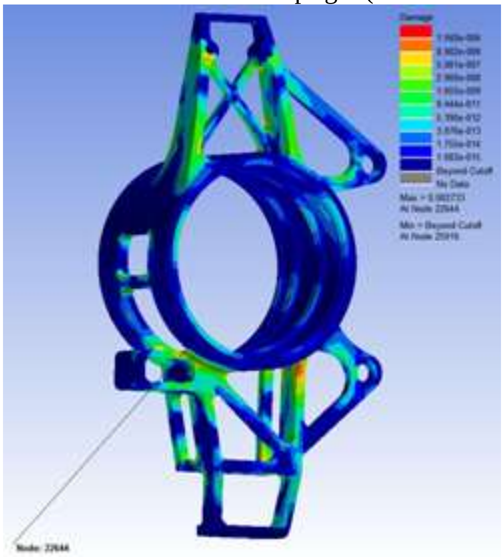

Figure 14: Damage Result of Front Upright

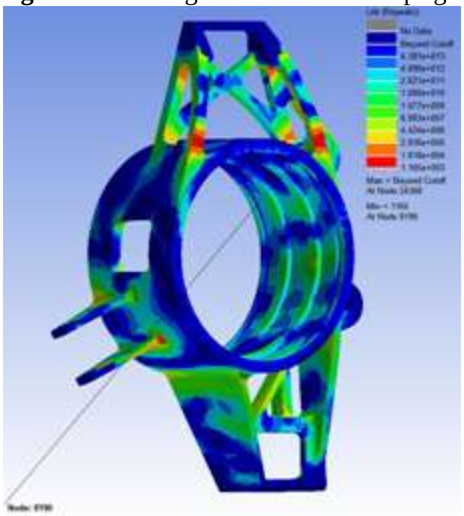

Figure 15: Stress Life of Rear Upright (Min:1165 cycles)

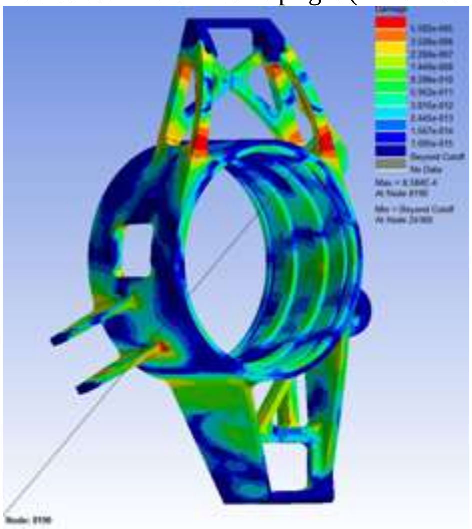

Figure 16: Damage Result of Rear Upright

6.3 Design Optimization and Fatigue Analysis Results

6.3.1 Design Optimization

On careful analysis of the results obtained from the fatigue analysis and using the understanding of stress concentration zones, the dimensions of some zones were modified to increase the life cycle of the component.

Figure 17: Cad Model of the Optimized Front Upright

Figure 18: Cad Model of the Optimized Rear Upright

6.3.2 Fatigue Analysis Results of Optimized Designs

The fatigue analysis was conducted for the optimized designs obtained using the same boundary condition and loading to obtain the new results for stress life. The minimum life results for the front and rear uprights were 410 cycles and 333300 cycles respectively. The maximum cumulative damage results are also shown below.

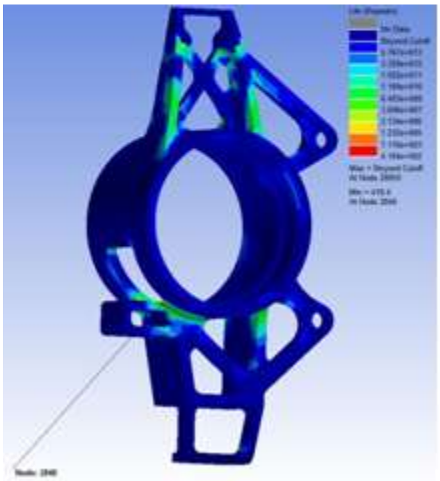

Figure 19: Stress Life of Front Upright (Min:410 cycles)

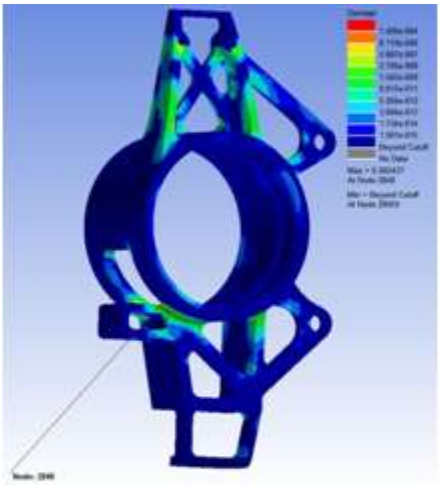

Figure 20: Damage Result of Optimized Front Upright

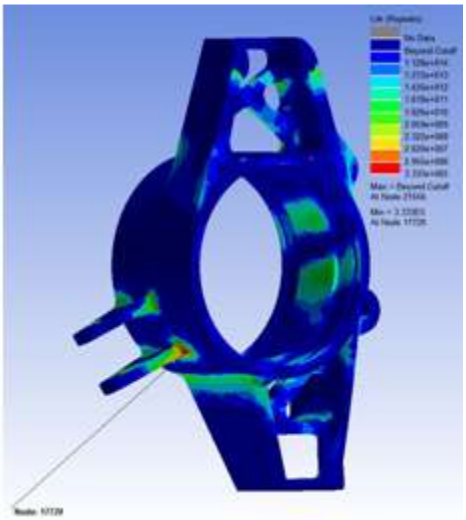

Figure 21: Stress Life of Optimized Rear Upright (Min:333300 cycles)

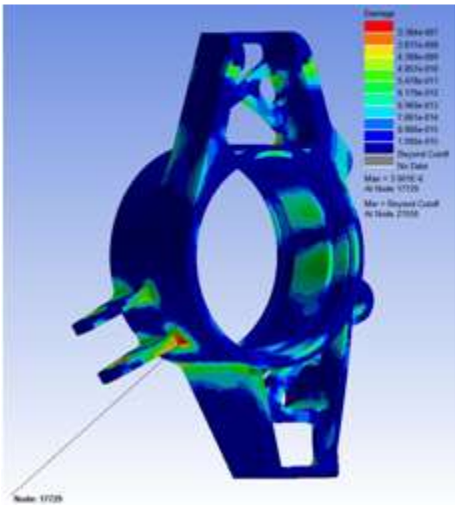

Figure 22: Damage Result of Optimized Rear Upright

7. Result

The Front upright had a factor of safety of 1.13 and the rear upright had a factor of safety of 1.26 from the static analysis. The minumum stress life for the Front and Rear Uprights were 365.9 and 1165 cycles or laps respectively. The minimum stress life of optimized designs of Front and Rear uprights were 410 and 333300 cycles or laps respectively. The mass increase for Front upright was 68grams and for Rear upright was 146 grams.

8. Conclusion

The static and fatigue analysis of the front and rear uprights of the fsae vehicle were conducted. It was observed that the component having the higher stress in static analysis results also had a lower fatigue life. The performance of the components was predicted in the terms of number of laps they will last for. Design optimizations were made based on the results of the initial fatigue analysis and improved life results were obtained. It was observed that the stress life of a component can be greatly enhanced by optimizing the geometry and increasing the thickness of the ribs near zones having low stress life. The minimum life of a component along with the maximum deformation value gives a better estimate of the overall performance of the component in comparison to the results of a basic static stress analysis.

Various physical tests of components have been carried out to estimate the damage an life of automotive components[4] but there is still a lack of documentation and research on fatigue life estimation of automotive components using finite element methods. Considering the results of the study conducted, it can be concluded that major improvements in terms of fatigue life enhancement, performance prediction, vehicle reliability, weight reduction and other design parameters can be achieved by carrying out the fatigue analysis of the components.

References

[1] Milliken, W. F., and Milliken, D. L., “Race Car Vehicle Dynamics”, SAE Inc. Milliken, 1995
[2] Julie A. Bannantine, Jess J. Comer, James L. Handrock, “Fundamentals of Metal Fatigue Analysis”, Prentice Hall Englewood Cliffs, New jersey 07632
[3] Carroll Smith, “Weight, Mass Load and Load Transfer” in Tune to Win, Aero Publishers, INC, 329 West Aviation Road, Fallbrook, CA 92028
[4] S.Schoenborn, H.Kaufmann, C.M.Sonsino, R.Heim,”Cumulative Damage of High-strength Cast Iron Alloys for Automotive Applications’,Procedia Engineering Volume101, 2015, Pages 440-449
[5] Andrew Jen Wong, “Design and Optimization of Upright Assemblies for Formula SAE Racecar”, International Journal of Mechanical and Industrial Engineering (IJMIE), ISSN No. 2231 –6477, Vol-2, Issue-1, 2012, March 2007.
[6] Force Calculation in Upright of a Fsae Race Car Anshul Dhakar and Rishav Ranjan Department of Mechanical Engineering, RV College of Engineering, Bangalore, India
[7] Buxbaum AO (1987) Structure fatigue design. Machine Press, pp.224–232.

Author Profile

Ayush Garg is currently pursuing his B.Tech (2014-2018) in Mechanical Engineering with a minor in Machine Design from Manipal Institute of Technology, Manipal. During 2014-2017, he worked actively in the official formula student team of Manipal University called Formula Manipal as a Vehicle Dynamics engineer and conducted the study on Design and Fatigue of wheel assembly components and Race car Brake Systems Design.