INTERNATIONAL JOURNAL OF ADVANCE RESEARCH, IDEAS AND INNOVATIONS IN TECHNOLOGY
IJARIIT
ISSN: 2454-132X
Impact Factor: 6.078
(Volume 7, Issue 2 - V7I2-1468)
Available online at: https://www.ijariit.com

# FEA analysis of a knuckle upright for a formula student car

*Abhishek Subhashchandra Amritkar*
*abhishek.21810262@viit.ac.in*
*Vishwakarma Institute of Information Technology, Pune, Maharashtra*

## ABSTRACT

***This research study is focused on to provide FEA analysis of a formula student car steering knuckle uprights to check the factor of safety. The model was designed in Solidworks® 2018 version and static structural analysis was carried out in Ansys® Workbench 2019 R3.Steering Knuckle uprights are used to connect steering tie rods suspension arms to wheel assembly(HUB) and also hold brake calipers. Being part of wheel,it is always in motion and sustains too much load. So it needs to be stiff at the same time as lightweight as possible. The material used for knuckle is Aluminium 7075 T6. Total deformation of 0.1752 mm was observed.Results are fairly reliable and can be utilised to go further with the manufacturing of the knuckle uprights.***

***Keywords:*** *Formula SAE - Knuckle Upright - FEA.*

## 1. INTRODUCTION

Formula Student Competition is a worldwide based engineering design competition wherein participating teams have to design, manufacture & compete with a formula styled race car. The main objective for this competition is to check whether the teams have done good engineering practices while the design and manufacturing phase of the vehicle. This competition inspires students to learn new skills apart from curriculum and to think out of the box to create the most lightweight car without breaching the rulebook as well as not losing performance. Designing safe and reliable components is the first priority of the event. For that particular reason, the main load carrying components need to be safer and lighter as it is a race car. To prove the design to the judges at the event, FEA analysis plays a crucial role. The current study is about the FEA analysis(static structural) of steering knuckle uprights with various element order and mesh types for better end results and safer approach towards validation. Steering uprights connect suspension to the wheel and wheel hub. Being the vital component of the wheel, it is always in motion therefore fatigue life becomes an important criteria for the design and to select the right material with required specifications for manufacturing. This research was carried out to check the design and safety of the component in terms of varying forces as well as fatigue life with the help of FEA.

Finite element analysis is a computer based method to predict the working of the model in terms of failure mode. FEA gives us the data about the failure of the component like where it will break and also shows the wear and tear of the part. It does this by breaking the model into small finite elements such as a cube of a triangle depending upon the geometry. Mathematical equations aid the FEA for predicting the behaviour of the model. FEA predicts models with respect to the external effects such as mechanical stress & vibration, fatigue, motion, heat transfer, fluid flow and many more.

To get good quality results in static structural analysis, we should look for finer mesh with element size ranging between 1 mm to 0.01 mm. And also to choose from linear or quadratic becomes important.

**Intermediate Goals**

1. In this study, we are focusing on static structural analysis of steering knuckle uprights.
2. Lightweight components are the key for performance of a race car.
3. Rigidity and reliability are also required for performance of the car.
4. Making components easy to build in-house.
5. More stiffness, more safety, less chances of failure.
6. In case of failure, it should be replaced with ease.

## 2. GEOMETRY

### 2.1 Model

Model was created using Solidworks® 2018 version and further with the aim of creating lightweight component aluminium 7075 T6 alloy was chosen. Aluminium 7075 T6 gives good strength and good fatigue life w.r.t stainless steel

**Fig. 1: Shows the 3D model of knuckle**

### 2.2 Material Properties

**Table 2: Material Properties.**

| Material | Aluminium 7075 t6 |
|---|---|
| Young's Modulus (GPa) | 71.75 |
| Poisson's Ratio | 0.33 |
| Bulk Modulus (KPa) | 70.34 |
| Shear Modulus (KPa) | 26.974 |
| Melting Temperature (ºC) | 477 |
| Tensile Strength (MPa) | 572 |
| Specific heat capacity (J/kg*k) | 714.8 |

Quest for finding alternatives for structural steel to reduce weight , we found aluminium alloy 7075 T6 most suitable and reliable for your area of research.

### 2.3 Model Support

The Knuckle was given support in the upper and lower ball point regions and in the ball bearing area.

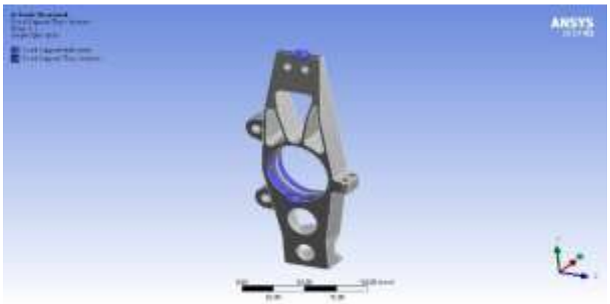


**Fig 2: Shows the fixed supports on the knuckle**

### 2.4 Forces Applied

Forces were applied at the point where steering , brake support, push rods and suspension arms(Double wishbone).

**Fig 3: Shows the braking support with 1500 n load**.

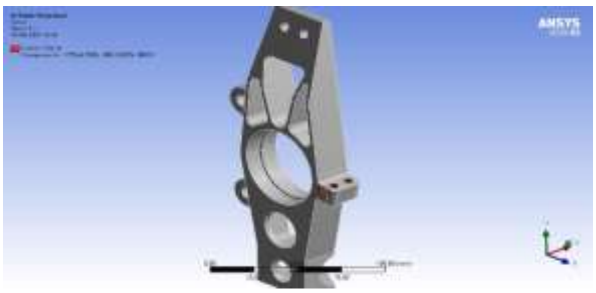


**Fig 4: Depicts force of 1750 n in steering arm**

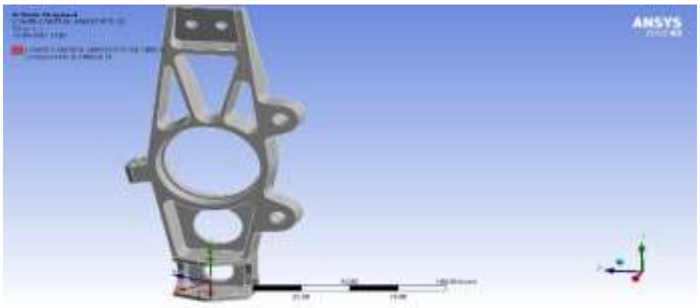


**Fig 5 : Depicts force of 1000 n in lower control arm(LCA**)

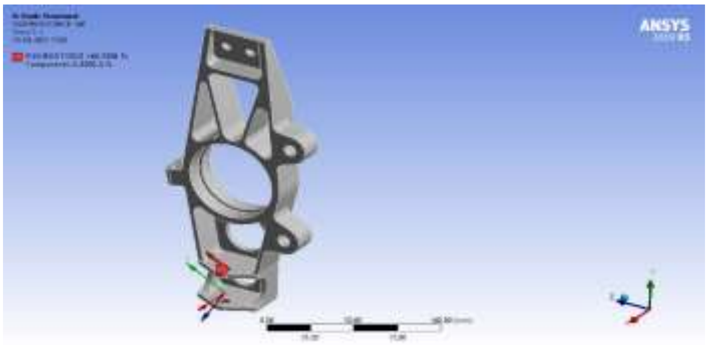


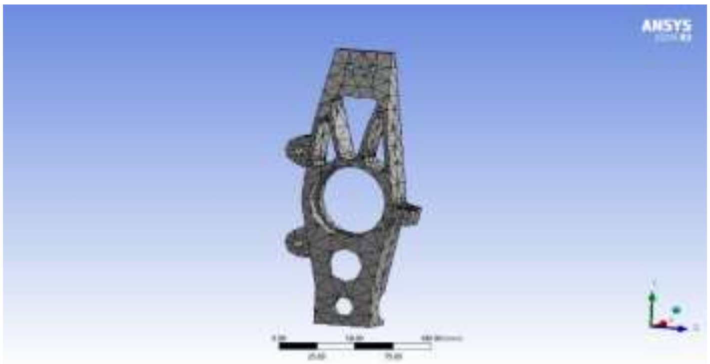
**Fig 6: Depicts force of 5000 n in push rod joint.**

## 3. ANALYSIS

### 3.1 Element size order and mesh types

**TRIAL 1**

| | |
|---|---|
| ELEMENT ORDER | LINEAR |
| SPAN ANGLE CENTER | COARSE |
| ELEMENT SIZE | 4 MM - DEFAULT |
| NODE | 1323 |
| ELEMENT | 3626 |



**TRIAL 2 :**

| | |
|---|---|
| ELEMENT ORDER | LINEAR |
| SPAN ANGLE CENTER | MEDIUM |
| ELEMENT SIZE | 4 MM - DEFAULT |
| NODE | 1522 |
| ELEMENT | 4325 |

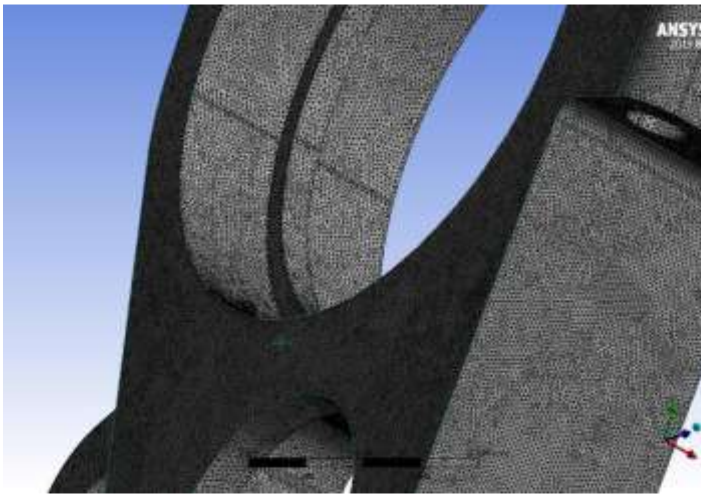
 

**TRIAL 3**

| | |
|---|---|
| ELEMENT ORDER | LINEAR |
| SPAN ANGLE CENTER | FINE |
| ELEMENT SIZE | 4 MM - DEFAULT |
| NODE | 2524 |
| ELEMENT | 7350 |

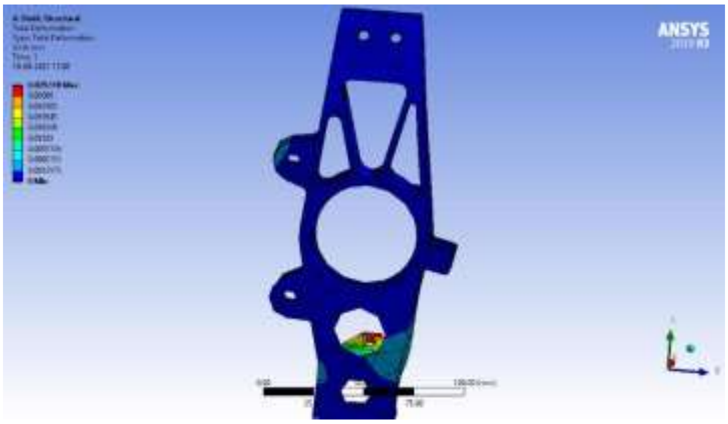
**TRIAL 4**

| | |
|---|---|
| ELEMENT ORDER | LINEAR |
| SPAN ANGLE CENTER | FINE |
| ELEMENT SIZE | 0.1 MM |
| NODE | 472588 |
| ELEMENT | 1790467 |

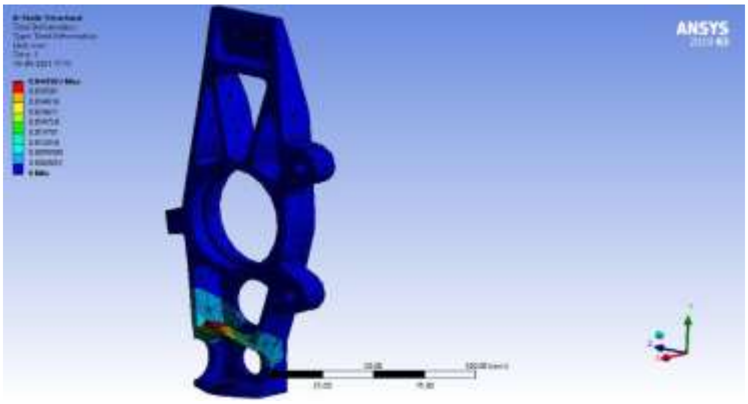


### 3.2 Analysis Results

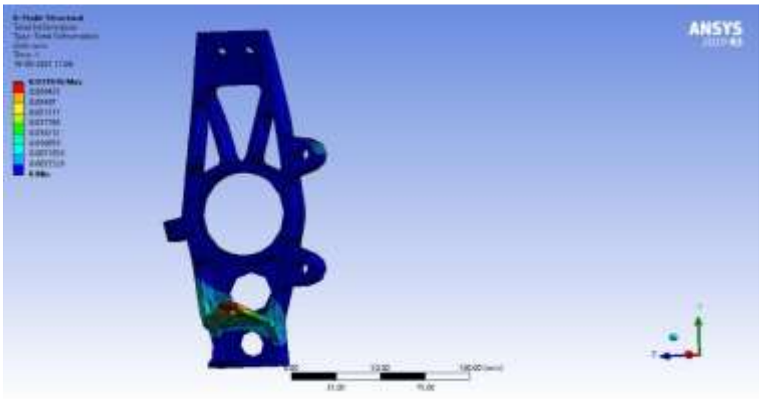
#### 3.2.1. Total Deformation



**TRAIL 1**

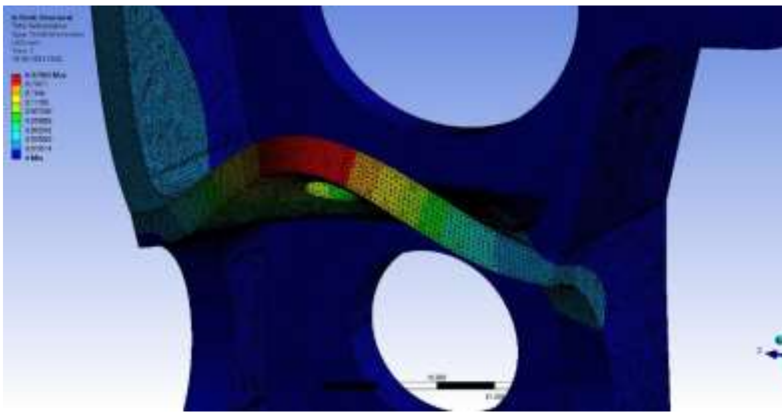


**TRAIL 2**



**TRAIL 3**

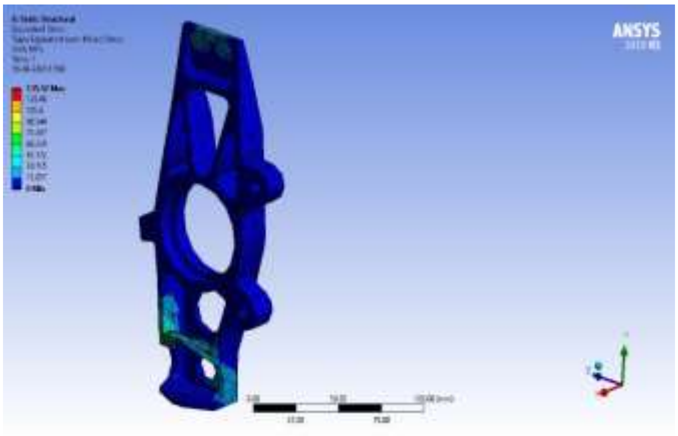
**TRAIL 4**



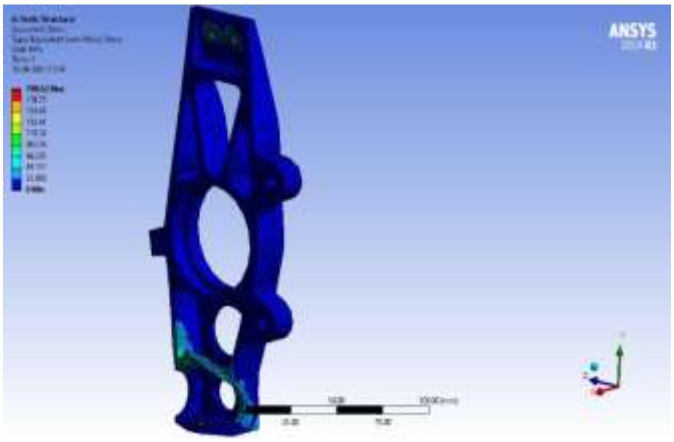
#### 3.2.2. Equivalent Stress

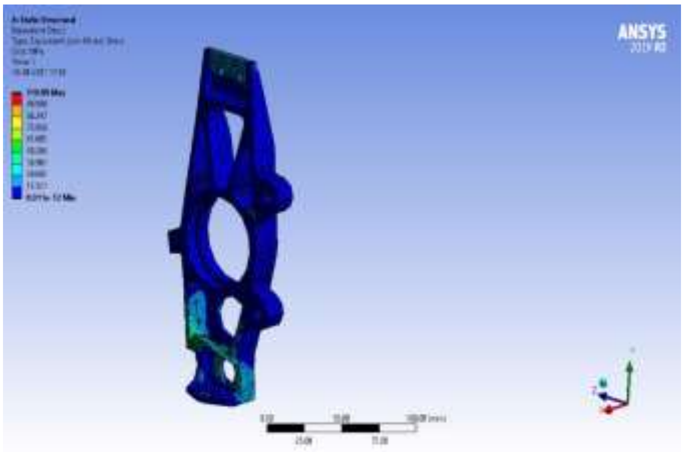
**TRIAL 1**



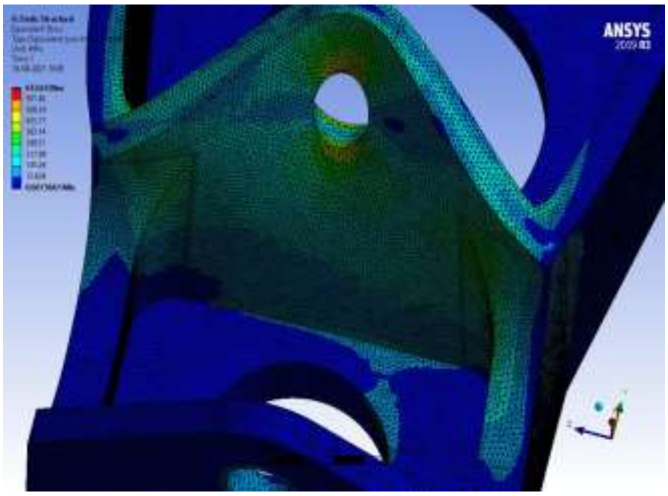
**TRAIL 2**



**TRAIL 3**

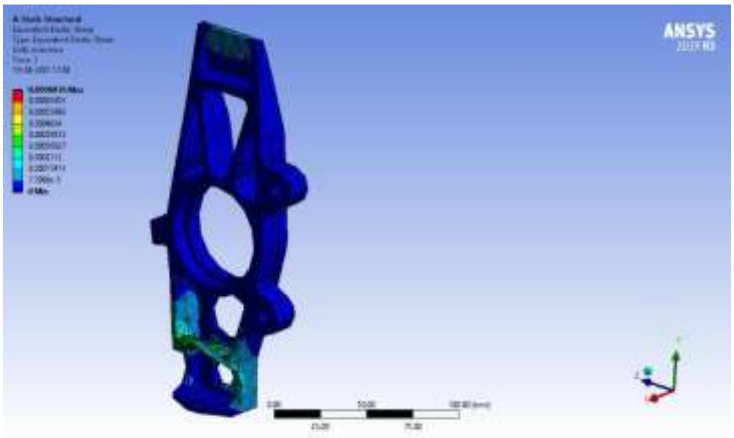


**TRIAL 4**

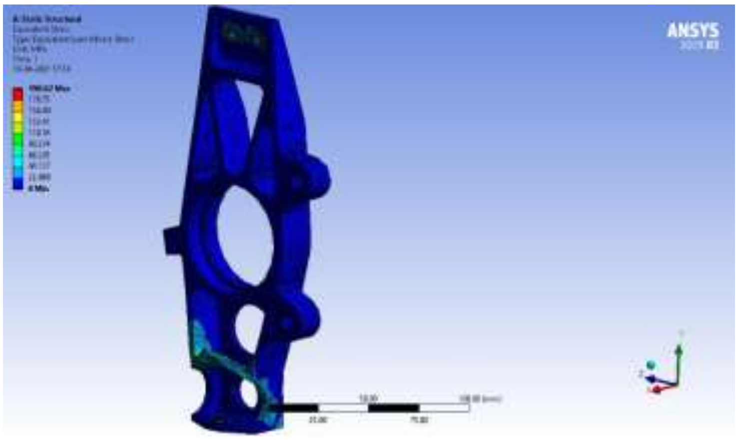


### 3.2.3. Maximum Shear Elastic Strain

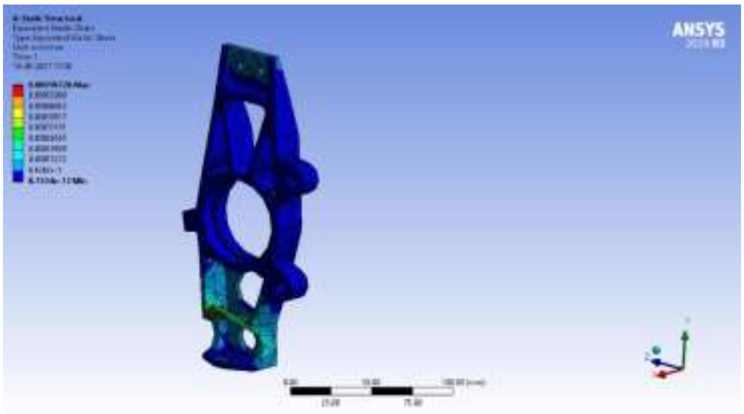
**TRAIL 1**



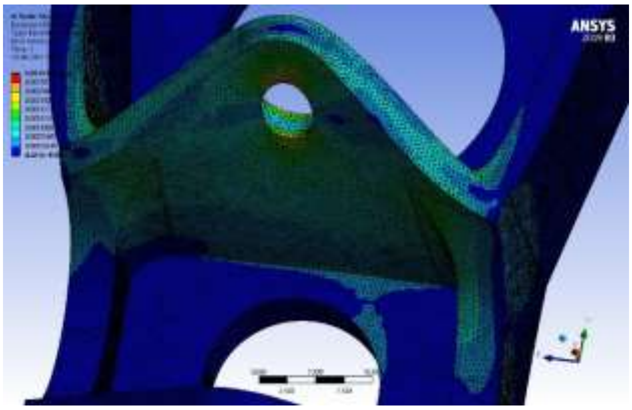
**TRAIL 2**



**Trail 3**



**Trail 4**



**3.3 Result**

| Analysis → Trails ↓ | Total deformation (mm) | Equivalent stress (mpa) | Equivalent elastic stress(mm/mm) |
|---|---|---|---|
| **TRAIL 1** | **0.0006936** | **135.52** | **0.029318** |
| **TRAIL 2** | **0.00099928** | **198.62** | **0.44507** |
| **TRAIL 3** | **0.00059726** | **110.89** | **0.031976** |
| **TRAIL 4** | **0.17562** | **653.65** | **0.0032789** |

## 4.CONCLUSION

From the above static structural analysis, we can conclude that finer mesh gives more accurate results as compared to coarse and medium mesh type. In this particular study we found that the forces given were above the actual working condition and considerable results were obtained. From this data, we can conclude that the design is good and with few more iterations it can be made more reliable.

## 5.REFERENCES

[1] Antriksh Mutha, Shubham Thosar and Nikhil Ghodmare Design and Optimization of a Steering Knuckle of FSAE Car.
[2] Razak I.H.A1 , Yusop M.Y.M2 , Yusop M.S.M3 , Hashim M.F4 MODELING, SIMULATION AND OPTIMIZATION ANALYSIS OF STEERING KNUCKLE COMPONENT FOR RACE CAR
[3] Pranav Shinde, Rushikesh Jirage, Kunal Garde
[4] Design and optimization of hub and knuckle for Formula SAE car.
[5] Dumbre, P., Mishra, A.K., Aher, V.S., and Kulkarni, S.S. (2014). Structural analysis of steering knuckle for weight reduction, International Journal of Advanced Engineering Research Studies, April-June.
[6] Bagus Aulia Saputro, Ubaidillah, Dicky Agus Triono, Dzaky Roja Pratama, Sukmaji Indro Cahyono, and Fitrian Imaduddin 'Static Load Simulation of Steering Knuckle For A Formula Student Race Car'
[7] Weight Optimization of Hub and Knuckle Using Topology Optimization P. Chaudhari, R. Khairnar