International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 07 Issue: 07 | July 2020 www.irjet.net p-ISSN: 2395-0072

Static Analysis and Geometry Optimization of Rear Wheel Trailing ARM

NADIKOPPULA PARDHASARADHI[1], M Y PRASADA RAO[2]

[1]PG Student, Dept. of Mechanical Engineering, MVGR College of Engineering (A), Vizianaragam, AP, India.
[2]Assistant professor, Dept. of Mechanical Engineering, MVGR College of Engineering (A), Vizianaragam, AP, India.

---***---

Abstract - *This paper introduces modeling and analysis of the failure of the rear wheel trailing arm bracket case study at FIRDO Engineering, Bangalore. On the application of bump loading type as 1G, 2G, and 3G condition, the modeling done using SOLIDWORKS, and Pre-processing done using ANSA pre-preprocessor- BETA CAE Systems. Post-processing and solving done using ABAQUS FEA SIMULIA by Dassault systems, the maximum von Mises stress for existing rear trailing arm in 1G bump load condition was 220MPa. Due to less factor of safety, the prevailing rear-trailing arm failed statically and dynamically. From the analysis of The von Mises stress under 3G bump, load conditions were 80.79MPa. Hence, the design is statically safe. Static analysis performed for 3G bump load of modified design of Rear-wheel Trailing Arm made from AISI STEEL 1018 material for thickness increased 2mm to 3.5 mm. The revised model is much safer than the original.*

Key Words: **Trailing arm, Suspension system, FEA, CAE, Analysis, ANSA, and ABAQUS.**

1. INTRODUCTION

The suspension system is the most valuable segment of a vehicle where its principle work is to reduce the vibrations from being transmitted to the car from the road, to protect the safety of the vehicle in pitching or rolling. To maintain the safety of the travelers from sudden shock loads, also to furnish directing solidness with high stability and to give great road holding while at the time of driving, braking, and turning. This paper is the study of geometric and dimensional optimization of the rear-wheel trailing arm suspension system. This model designed in Solidworks software, pre-processor ANSA does meshing, and post-processing and analyzed using Abaqus.

1.1 TYPES OF SUSPENSION SYSTEM

There are two types of suspension system:

a. DEPENDENT TYPE: - Independent type of suspension load on one side of the suspension leads to movement on the other side of the suspension system.
   Ex: Wishbone, Solid axle, etc.
b. INDEPENDENT TYPE: - In the separate type of suspension system movement on one wheel not transmitted to the other wheel of the system. This type of suspension used in many automobiles for both rear-wheel and front-wheel suspension system
   Ex: Solid axle leaf spring and solid axle coil spring etc.

1.2 WORKING PRINCIPLE OF TRAILING ARM

A trailing arm bracket is the main component of the automobile rear-wheel suspension system. There are various types of suspension systems has utilized for both front-wheel and rear-wheel suspension system. In vehicles trailing arm suspension is free suspension configuration, uses a long hollow solid to locate the wheel. Components of vehicle suspension are springs, dampers, and connectors trailing arms of the body with wheels. Springs are adaptable components, which act as the supply of vitality during movement. The point when vehicles move on the road, energy has discharged with the activity of damper and power changed over into heat energy; thus, the vibration has reduced.

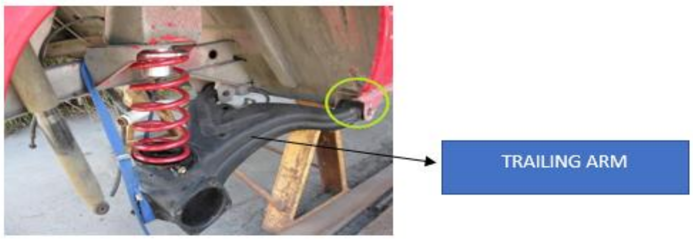

Fig -1: Rear wheel trailing arm bracket

1.3 PROBLEM STATEMENT

FIRDO Engineering services, Bangalore has been working on R&D of Design of HONDA Car of US-based manufacturer. Here the Honda US gives the design of the car, and FIRDO services do pre and post-processing of the vehicle design. However, failure of the design of the Trailing-arm bracket FIRDO asked for R&D work of the vehicle design. It has failed due to deformation by excessive load acted upon the trailing arm bracket, which Leads to the camber effect of the vehicle and causes tire wear.

1.4 OBJECTIVE

Main objective of work is to analyze the trailing arm geometry, dimensions for the modified and original design. Now trailing arm can withstand more load comparatively with the unique design. The specific objective is to analyze trailing arm using Computer-aided Finite element analysis.

2. LITERATURE REVIEW

A. Dr.A.V.VANALKAR & Mr.Sushilkumar P.Taksande: In this paper configuration and investigation of vehicle front suspension lower arm has done to examine the pressure condition and to choose the appropriate materials for the front suspension lower arm. The principal goals of this examination to decide essential areas and Strain circulations of the segment. The paper intends to finish Finite Element Analysis of the front suspension lower arm, which comprises the pressure improvement stacking and examination for disfigurement.

B. Z. Husin, M.M. Rahman et al.: In this paper, entire research work centers on the limited component-based stress life forecast of the lower suspension arm, exposed to variable load stacking. The targets of this examination are to foresee the stress life of the lower suspension arm utilizing pressure life and strain-life strategies, to examine the impact of the mean pressure. The lower suspension arm created using CAE.

C. Vijay Solanki: This paper considers the von-misses stress investigation of immediate suspension connect completed for static deflection, and von-misses stress form plot has plotted. The result shows that the extreme go is 0mm to 8.22mm. What's more, von-misses pressure shows the least estimation of 0.86Mpa, and the Maximum is 3752Mpa. The maximum von-misses stress is less than the yield of material, and some remedial activity performed at where the connection is associated with the body. After the pressure created on the arm, it has discovered that some metal thickness ought to be incremented at the frame association point to maintain a strategic distance from the disappointment.

D. Lihui Zhao: In this paper Structure, advancement strategies under static load conditions have broadly studied in the car industry for lightweight and execution improvement of present-day vehicles. Notwithstanding, these static burden conditions couldn't speak to all the extreme circumstances of car parts exposed to complex loads shifting with time, particularly for the lower control arm of the front suspension. In this examination, dynamic streamlining of a lower control arm has performed by brushing conventional static load advancement methods and multi-body elements by Equivalent Static Load (ESL).

E. Nima Al- Asady: This paper studies that the decrease of advancement time has been a significant test looked via automobile makers to remain sober in their industry. Durability assessment is one of the most tedious components in the advancement procedure. An innovative computational methodology dependent on limited component examination (FEA) for coordinated durability in a car lower suspension arm part has introduced. Direct FEA of the lower suspension arm has led to decide critical locations and strain dispersions of the segment.

3. LOAD CALCULATION:

In the vehicle below, all the load distributions have represented, and Duygu Geller Külcü method used for load calculation. Numerical data has rendered below

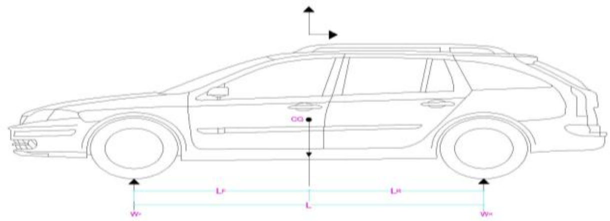

Fig -2: Line diagram of automobile with forces

L, Wheel base length= 2660 mm

LR, Distance from rear wheel axle to CG = 0.45*L= 1197mm

LF, Distance from front wheel axle to CG = 0.55*L = 1463mm

Total weight of the car= 1845 Kg

$$W=mg \qquad \text{-------- (1)}$$

Using equations below load acting on front and rear wheels has calculated

$$W_{FA}= W\frac{Lr}{L} \qquad \text{-------- (2)}$$

$$W_{RA}= W\frac{Lf}{L} \qquad \text{-------- (3)}$$

For one wheel, static load calculated using

Load on the front wheel represented as

$$W_{FAW} = W_{FA}/2 \qquad \text{-------- (4)}$$

Load on the rear wheel represented as

$$W_{RAW} = W_{RA}/2 \qquad \text{-------- (5)}$$

Force acting on the trailing arm bracket has calculated using above formulae

$$W=mg= 1845*9.81=18099.45 \text{ N}$$

$$W_{FA} = W \frac{Lr}{L} = 18099.45 \frac{1197}{2660} = 8144.75 \text{ N}$$

$$W_{RA} = W \frac{Lf}{L} = 18099.45 \frac{1463}{2660} = 9954.69 \text{ N}$$

Load acting on wheels separately calculated using

$$W_{FAW} = W_{FA}/2 = 4072.37 \text{ N}$$

$$W_{RAW} = W_{RA}/2 = 4977.34 \text{ N}$$

We are calculating for rear wheel suspension hence load acted upon has considered as 5400N.

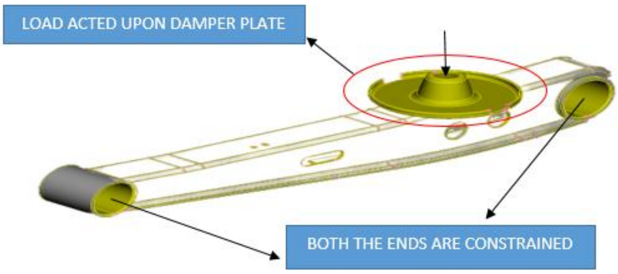

Fig -3: Boundary condition on original CAD

4. ANALYSIS OF ORIGINAL TRAILING ARM:

Analyzing the original trailing arm is mandatory to analyze after making modifications. Where we can compare the results of original and modified trailing arm easily. The material used in the original CAD is AISI1018 with Young's modulus is 210GPa, and Poisson's ratio is 0.3. CAD has created in Solidworks, pre-processing done in ANSA 18.0, and post-processing, the nodal and static analysis have done using ABAQUS 6.14.

4.1 QUALITY PARAMETERS:

Shell quality parameters used in this analysis are standardized values in FIRDO Engineering, and the benefits are

Table -1: Quality criteria and failure conditions

Quality criteria	Failure point
Aspect ratio	5
Skewness	45
Warping	12
Taper	0.35
Jacobian	0.6
Min length	1.8
Max length	5.6
Min angle quads	45
Max angle quads	135
Min angle trias	30
Max angle trias	120

4.2 MID SURFACE EXTRACTION IN ANSA 18.0:

1. Check the thickness of the whole part using middle > multi.
2. Given Offset and created a middle surface using Offset and middle option.

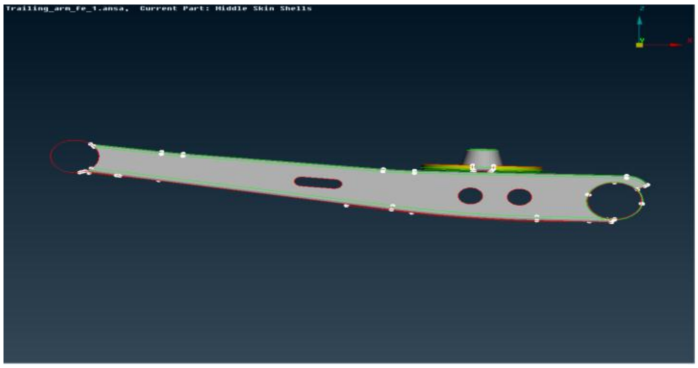

Fig -4: Mid surface extracted from CAD

4.3 MESHING STEPS IN ANSA 18.0:

1. Peruse the IGES document with various resilience settings and evaluate the outcomes.
2. Clean up geometry, close holes, alter or make new Faces and so forth.
3. Optionally, rehash the past advance with the assistance of Automatic Clean-Up usefulness.
4. De-highlighting, evacuating subtleties, for example, little openings and filets.
5. Mesh the parts with a component length of 3 mm.
6. Improve the work by advancing the state of the Macro Areas.

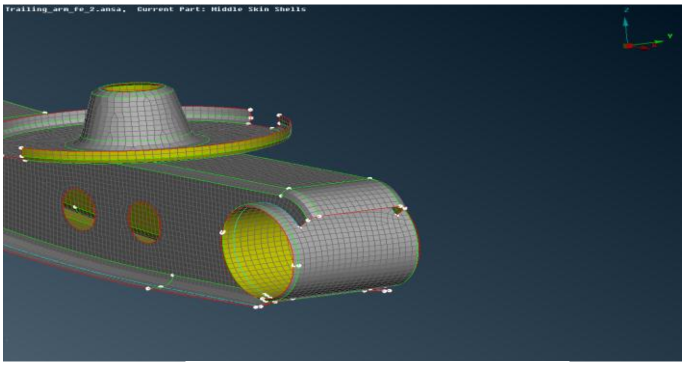

Fig -5: Meshing has given to mid surface.

4.4 QUALITY CHECKING IN ANSA:

1. After meshing has completed, verified the quality parameters if all compelled under the given values or not.
2. Given shell to reliable and checked if the mesh has adequately aligned or not.
3. Quality check is finished and now ready to dump into Abaqus.
4. Add a load of 5400N at the top of the damper plate.

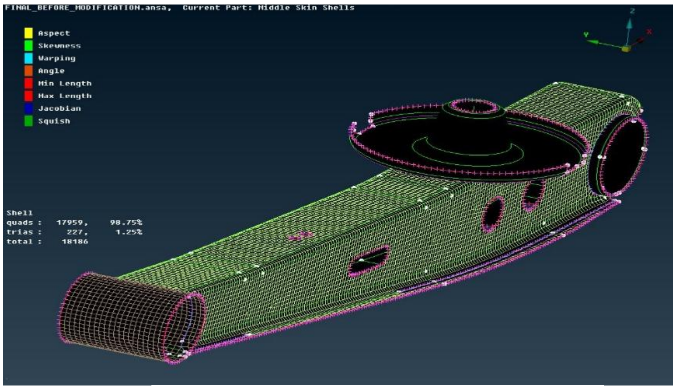

Fig -6: Checking quality using trias percentage.

4.5 POST PROCESSING AND ANALYSIS IN ABAQUS :

1. After meshing has completed in ANSA sets has created for giving connections.
2. Now connections have provided between damper plate to the surface of the trailing arm.
3. Given the degree of freedom as zero at the moving ends.
4. After clicking on visualization results obtained as shown below

Original CAD analyzed and values obtained after the application of boundary conditions for a load of 5400N, and it has noted that Max stress obtained is 220.09 MPa at 1G load condition. Hence, modification of CAD has required near the damper plate.

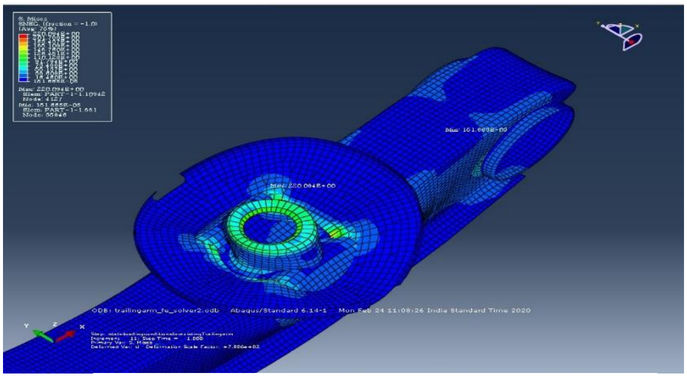

Fig -7: Max and Min failure visualization in Abaqus 6.14.

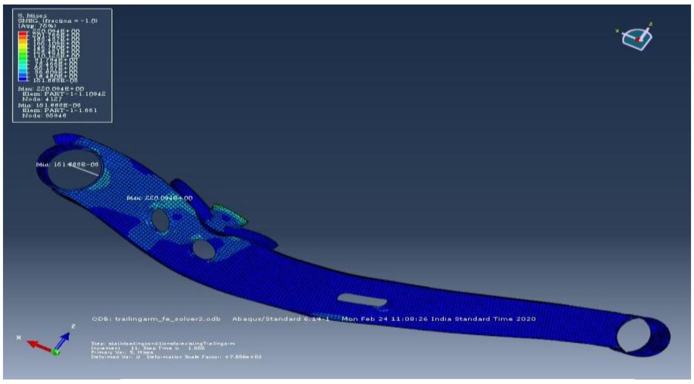

Fig -8: Deformation of the trailing arm under load in Abaqus.

5. DESIGN MODIFICATION AND ANALYSIS:

In a static analysis of trailing arm, we found that Max stress of 1G load is on damper plate, now the dimensions of the damper plate have increased by 1.5mm, and again analysis has performed in Abaqus. By comparing the results from the original component, the life of the element has increased.

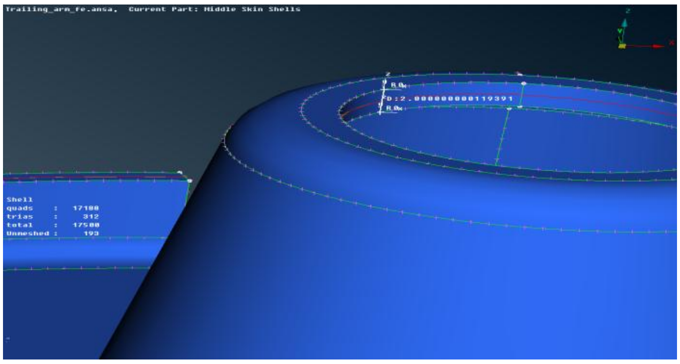

Fig -9: CAD PID before modification

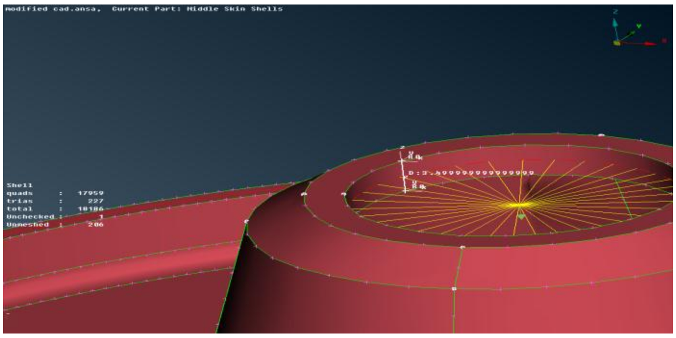

Fig -10: CAD PID after modification

Static analysis has carried out after modification with a load of 5400N and results plotted and compared. From the results seen that max stress has reduced, and the value is within the yield point of the material. Hence, the design is safe.

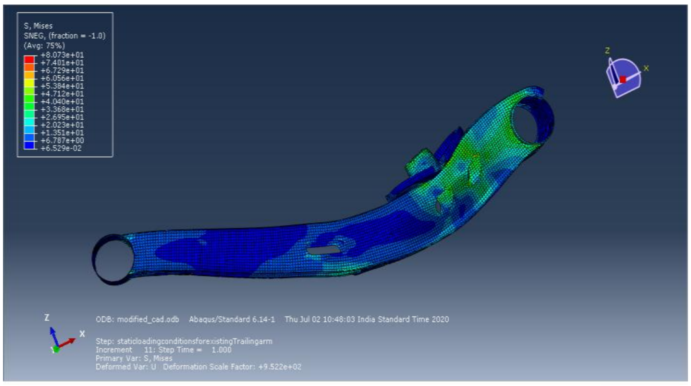

Fig -11: Modified CAD analysis in Abaqus 6.14

6. RESULTS AND DISCUSSIONS:

On comparing, the results obtained from static analysis at the load of 5400N and stress values as shown below

Table -2: Stress values for static analysis

Trailing arm	**Load (N)**	**Observed stress**
Original	5400	220MPa
Modified	5400	80.79MPa

As shown in the above table, the values of stress have changed to very less amount. Hence, the design is safe.

7. CONCLUSIONS

The objective of this work is to reduce the stress acted upon the body by a dimension optimization technique.

1. The value of stress has reduced from 220MPa to 80MPa, keeping the trailing arm in safe conditions.
2. The thickness of the damper plate has increased from 2mm to 3.5mm; at this condition, the design is safe.
3. The value of the yield stress of the damper plate is within the yield limit of the material.
4. The modified design is safe, nodally, and statically.
5. This work also serves as a case study for students in determining the fatigue life of the part.

REFERENCES

1. Bannantine, J.A., Comer, J.J. and Hardrock, J.L., 1990, “Fundamentals of metal fatigue analysis”, Prentice-Hall.

2. Avinash Suresh C. (2016), “Static and fatigue failure analysis of trailing arm bracket of rear suspension system”, International Journal of Innovative Research in Science, Engineering and Technology, 5(7).

3. ThosarAniket (2014), “Design, analysis, and fabrication of rear suspension system for an all-terrain vehicle”, International Journal of Scientific & Engineering Research, 5(11).

4. MSC/NASTRAN version 70.7.2 manual, 2001, The MacNeal-Schwendler Corp., Los Angeles, CA.

5. Vanalkar A.V. (2015), “Design, modeling and failure analysis of car front suspension lower arm”, International Journal of Science Technology & Engineering, 2(1).

6. Davis, J., 1997, Concise metals engineering data book, ASM International, Materials Park, OH.

7. Z. Husin, M.M. Rahman, K. Kadirgama, M.M. Noor, and Rosli A. Bakar. "Prediction of fatigue life on lower suspension arm subjected to variable amplitude loading" *pp. 100-116*

8. A. Ossa, C.C. Palacio b, M.A. Paniagua. "Failure analysis of a car suspension system ball joint." Materials Engineering Research Group, East University, Cra. 49 No. 7 or 50, Medellin, Colombia.

9. Vijaykumar V Solanki and Prof. N.S Patel, "Static analysis and geometric optimization of independent suspension link." Published on April-June 2014. Pp759-766.