Journal of Advanced Science and Engineering Research 1 (2011) 124-136

Dynamic Analysis of Vehicle Arm Based on Finite Element Approach

Hemin M. M, Rahman M. M and Omar R.M.
Faculty of Mechanical Engineering, Universiti Malaysia Pahang, Malaysia.
Automotive Engineering Centre, Universiti Malaysia Pahang, Malaysia.
Universiti Tenaga Nasional, Malaysia.
hemin_hm@yahoo.com

Article Info

Received: 25th July 2011
Accepted: 22th August 2011
Published online: 10th September 2011

ABSTRACT

In the automotive industry, the riding comfort and handling qualities of an automobile are greatly affected by the suspension system. In this paper a suspension arm model based on finite element analysis (FEA) is proposed. First, critical location and suitable materials for the suspension arm has been identified, and then the contact stress analysis and dynamic behavior of lower arm has been investigated. A structural modeling of the lower arm suspension was developed using Solid Works and aluminum alloys (AA7075-T6) are selected as a suspension arm materials. The linear static stress distribution is investigated using the commercial FEA package, and dynamic analysis was performed using NASTRAN software. Two types of nodes, 4 nodes tetrahedral (TET4) and 10 nodes tetrahedral (TET10) has been used in the finite element modeling. According to the results TET10 are able to capture the higher stresses than TET4 for the same global length. In addition TET10 mesh size 0.1 mm (54178 elements) has been chosen for dynamic analysis because of predicted higher maximum stresses. This model will provides a solid foundation for further study of failure life analysis of the suspension arm components.

Keywords: **lower arm, finite element analysis, natural frequency**

1. Introduction

Stress analysis activities vary depending on the function and maturity of the phase, an important benefit of performing stress analyses is the ability to determine design sensitivities and to conduct trade studies. One of the basic tasks in dynamic analysis of the various constructions is to evaluate the displacements of the construction as the time dependent functions when the time varying loads are given.

Finite element packages have been readily available, and their utility has increased with the development of fast computers. The finite element method provides a relatively easy way to model the system.

Conle and Mousseau (1991) used vehicle simulation and the finite element results to generate the fatigue life contours for the chassis components using automotive proving ground load history results combined with the computational techniques; They concluded that the combination of the vehicle dynamics modeling, finite-element analysis, and fatigue analysis are the viable techniques for the fatigue design of the automotive components. Kim et al. (2002) were studied a method for simulating vehicles dynamic loads, but they add durability assessment, for their multibody dynamic analysis they use DADS and a flexible body model. Dynamic stress analysis was performed using MSC. NASTRAN. The fatigue life was then calculated using a local strain approach; The result fatigue life, shows the majority of the fatigue damage occurred over a frequency range that depend on terrain traveled (service or accelerated test course). Nadot and Denier (2003) have been studied fatigue phenomena for nodular cast iron automotive suspension arms. The authors found that the major parameter influencing fatigue failure of casting components are casting defects.

Baek et al. (1993) proposed an integrated computational durability analysis methodology, the multi-body dynamic simulation software, DADS was used to calculate dynamic loads of a mechanical component that is modeled as a rigid body in the multi-body mechanical system. Recently the suspension arm get more attention by many research like(Attia, 2002) study dynamic analysis of the double wishbone motor-vehicle suspension system using the point-joint coordinate's formulation the mechanical system is replaced by an equivalent constrained system of particles and then the laws of particle dynamics are used to derive the equations of motion.

Gopalakrishnan and Agrawal (1993) carried out durability analysis of full automotive body structures using an integrated procedure, in which the dynamic simulation software ADAMS was used to generate loading histories, and the Inertia Relief Analysis of MSC.NASTRAN was used to analyze the model and to get displacements and stresses. Then, the Fatigue Life Analysis Procedure (FLAP) was used to analyze the durability for selected critical areas from the full model. . In this paper, MSC.NASTRAN finite element techniques have been used as a tool to model the mechanical properties of the suspension arm firstly.

Three-dimensional linear tetrahedron solid elements (TET10) and tetrahedral elements (TET4) are used for the initial analysis based on the loading conditions. Convergence of stress energy was considered as the criteria to select the mesh size, Developed the structural model and identify the critical locations and predict the dynamic behavior of suspension arm.

2. Motion For Suspension System Of Automobile

Natural frequency is the rate of energy interchange between the kinetic and the potential energies of a system during its cycle motion. As the mass pass through the static equilibrium position, the potential energy is zero (Dimarogonas, 1996). The natural frequency is expresses as Eq. (1).

$$w_n = \sqrt{\frac{k}{m}} \tag{1}$$

Where: w is natural frequency, k is coefficient of spring and m is mass

The chassis natural frequency is used the suspension rate and chassis mass and expressed as in Eq. (2);

$$w_n = \sqrt{\frac{k_s}{m_c}} \tag{2}$$

Where: w_n is natural frequency for the car, k_s is coefficient of spring, and m_c is mass of the car

For the wheel natural frequency ωw, it's necessary to take into account Ks and Kt because the wheel oscillates the suspension and tire springs. Although these two springs are on opposite side of the wheel/hub/knuckle mass, the mass would feel the same force if the two springs were in parallel on one side of the mass. In other words, the two springs, Ks and Kt, are in parallel and their composite rate is their sum.

$$w_w = \sqrt{\frac{k_s + k_t}{m_w}} \tag{3}$$

Where w_w is the natural frequency of the wheel, k_s is the coefficient of spring, k_t is coefficient of tire and m_w is mass of the wheel

Automobile suspension arm is two-degree of freedom system. The two-degree of freedom car suspension model is illustrated in Fig.1 (Milliken, 2002).

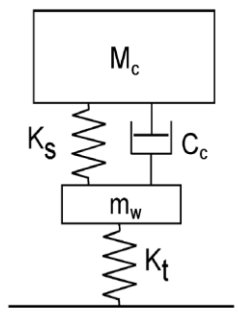


Fig.1. Quarter car passive suspension arm

3. Model Description

Vehicle suspension is a mechanism locating between the sprung mass (vehicle body) and the unsprung masses (wheels) of the vehicle. The suspension provides forces between these two masses of the vehicle according to certain state variables of the vehicle. A good car suspension system should have satisfactory road holding ability, while still providing comfort when riding over bumps and holes in the road. When the bus is experiencing any road disturbance the bus body should not have large oscillations, and the oscillations should dissipate quickly.

A simple three-dimensional model of suspension arm as shown in Fig.2, was modeling by used Solid Works software. Fig.3 shows the overall dimensions for the model.

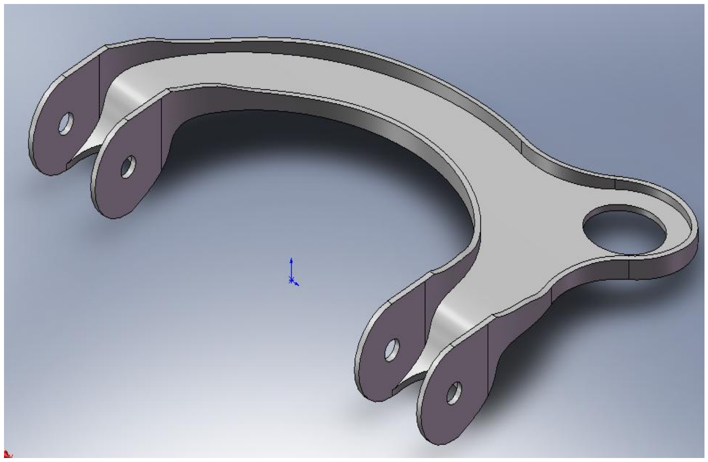


Fig.2. Structural mode

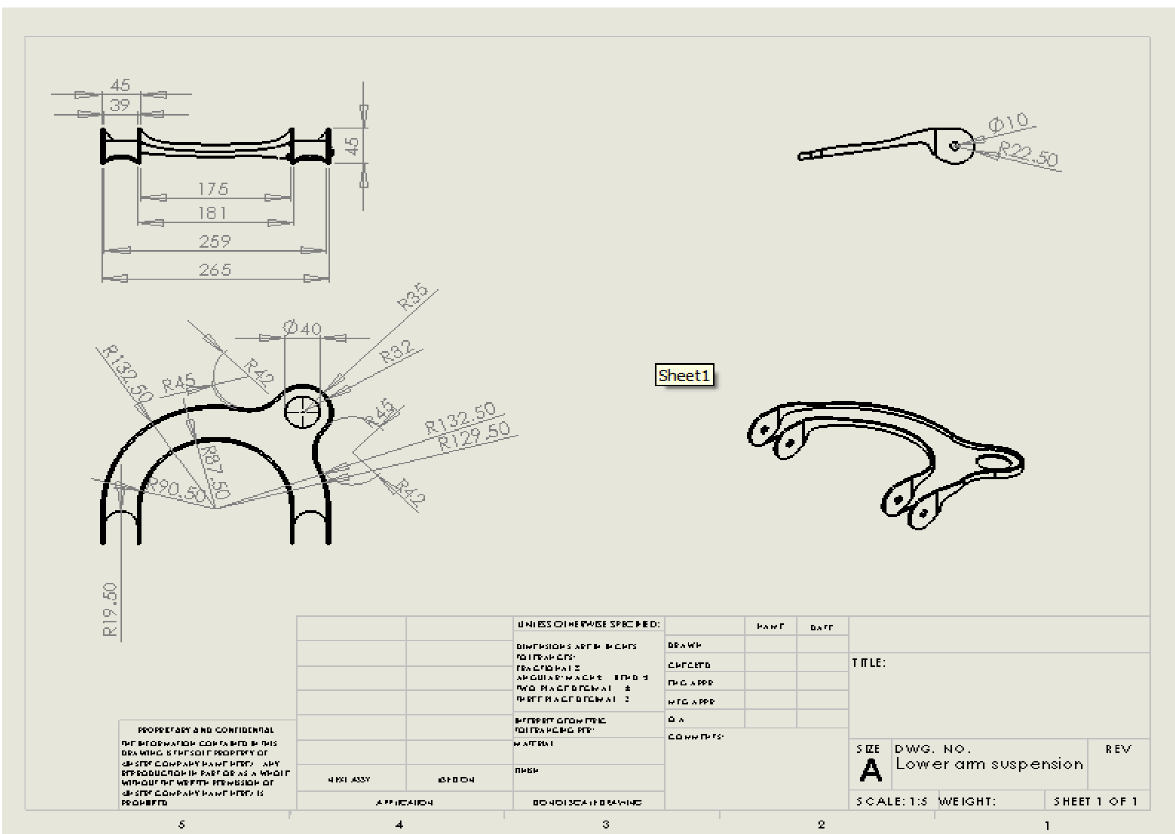


Fig.3. Overall dimension of suspension arm model

4. Mechanical Properties

Material model and material properties play an important role in the result of FE method. The material properties are one of the major inputs, which is definition of how the material behaves under the cyclic loading conditions. The materials parameters required depend on the analysis methodology being used. The mechanical properties of 7075-T6 aluminum alloy are shown in Table 1.

Table 1: Mechanical properties of aluminum alloy 7079-T6

Material	Young's Modulus (GPa)	Poisson's ratio	Tensile strength(MPa)	Yield strength (MPa)
Aluminum alloy AA7079-T6	72	0.33	503	572

5. Results And Discussion

5.1 Modeling and Simulation

The lower arm suspension is one of the important parts in the suspension system. A specific area of constraint has been set into the design in order to get a precise result. Two types of nodes, 4 nodes tetrahedral (TET4) and 10 nodes tetrahedral (TET10) has been used in the finite element modeling using MSC. PATRAN. These analyses were preformed iteratively at different mesh global length until the appropriate accuracy obtained. The convergence of the stresses was studied as the mesh global length was refined in the analysis. The mesh global length of 0.1 mm was chosen and the pressure of 8 MPa was applied at the end of the bushing that connected to the tire. The other two bushing that connected to the body of the car are constraint. The pressure that has been applied is based on Al-Asady et al. (2008). The three-dimensional FE model, loading and constraints of suspension arm is shown in Fig.4.

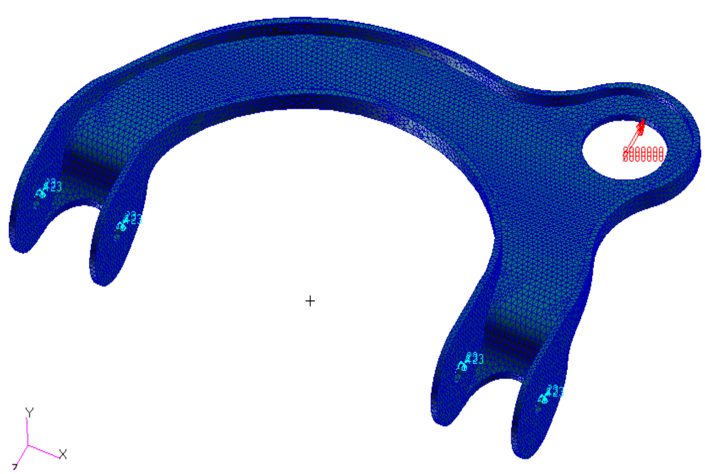


Fig.4. Three-dimensional FE model, loading and constraints

5.2 Effects of the Mesh Types

In the finite element modeling and analysis, a mesh study has been constructed by refining the mesh in order to obtain the accuracy of the calculated result which depends on the competitive cost (CPU time). For this analysis, the auto tetrahedral meshing approach is employed for the meshing of the solid region geometry. By using the tetrahedral meshing, a high quality meshing for boundary representation can be obtained from the solid model that is imported into the analysis.

The tetrahedral elements (TET10) and tetrahedral elements (TET4) are used for the mesh analysis. From the mesh analysis, TET10 gives a higher value of nodes compare to the TET4 type of mesh as shown in Figures 5-6. By setting the pressure and the constraint on the lower arm design, the first analysis result shows that TET10 mesh predicted higher von Mises stress than the TET4 mesh as various mesh global length shown in Figures 7-8. Then, Variation of maximum principal stresses, Tresca and Von Mises against the global mesh length for TET4 and TET10 are shown in Figures 9 and 10 respectively.

According to the result, comparison between TET10 and TET4 based on Mises, Tresca, and maximum principal stresses are Tabulated in Table 2 and 3 for TET 4 and TET10 respectively and the overall results show that the TET10 mesh give a higher stresses concentration compared to TET4 for the same mesh global length. Thus, TET10 is used for dynamic analysis.

From the stress analysis, the result shows that the white area of the design is the lowest predicted stress acted on the lower arm suspension design. Therefore, the area can be made as a guide in the future process of modifying or optimizing the design. It is also important to make sure that the critical points on the design which have the highest predicted stress should be look carefully in the process of modifying and optimizing the design in order to avoid any failure in the future usage of the lower arm design.

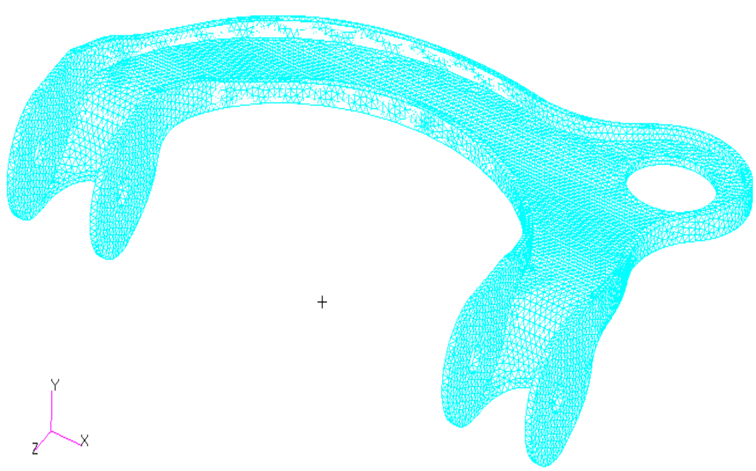


Fig.5. TET4, 54 141 elements and 15 098 nodes

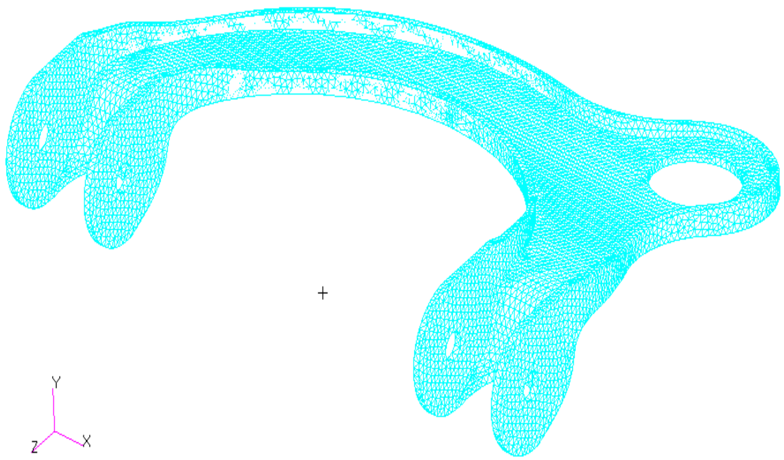


Fig.6. TET10, 54 178 elements and 96 080 nodes

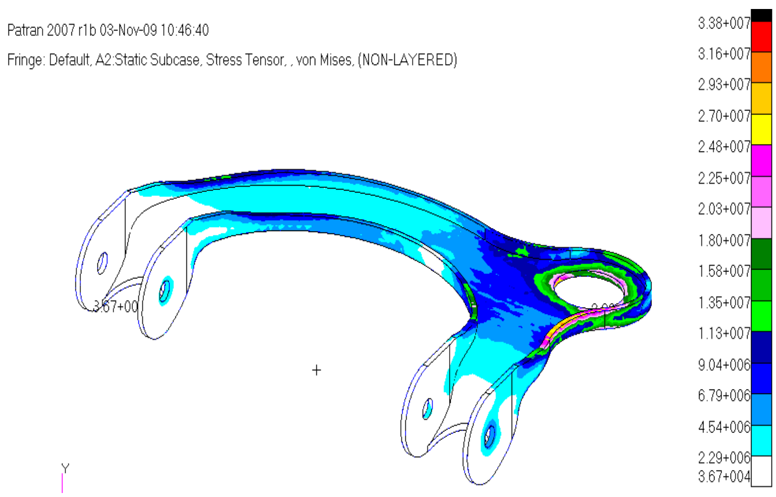


Fig.7. von Mises stresses contour for TET4

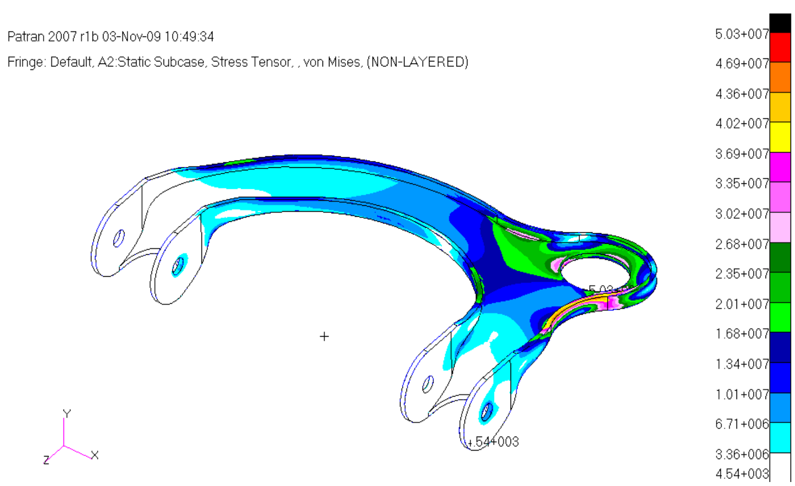


Fig.8. von Mises stresses contour for TET10

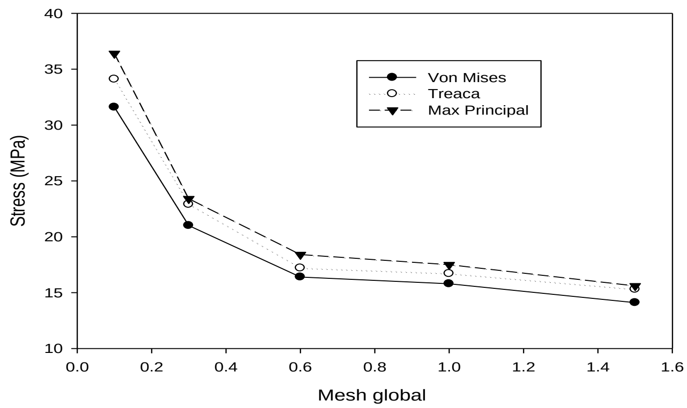


Fig.9. Stress contour for TET4

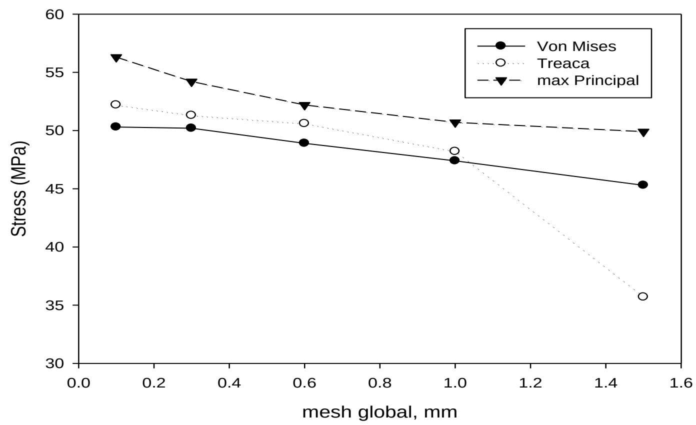


Fig.10. Stress contour for TET10

Table 2: Variation of stresses concentration at the critical location of the suspension arm for TET10 mesh

Mesh size (mm)	Total nodes	Total Elements	Von Mises (MPa)	Tresca (MPa)	Max Principal Stress (MPa)
0.1	96080	54178	50.3	52.2	56.3
0.3	10041	4676	50.2	51.3	54.2
0.6	5889	2665	48.9	50.6	52.2
1.0	5436	2465	47.4	48.2	50.7
1.5	3186	1409	45.3	35.7	49.9

Table 3: Variation of stresses concentration at the critical location of the suspension arm for TET4 mesh

Mesh size (mm)	Total nodes	Total Elements	Von Mises (MPa)	Tresca (MPa)	Max Principal Stress (MPa)
0.1	15098	54141	31.6	34.1	36.4
0.3	1775	4643	21.0	22.9	23.4
0.6	1055	2621	16.4	17.2	18.4
1.0	977	2429	15.8	16.7	17.5
1.5	561	1320	14.1	15.3	15.6

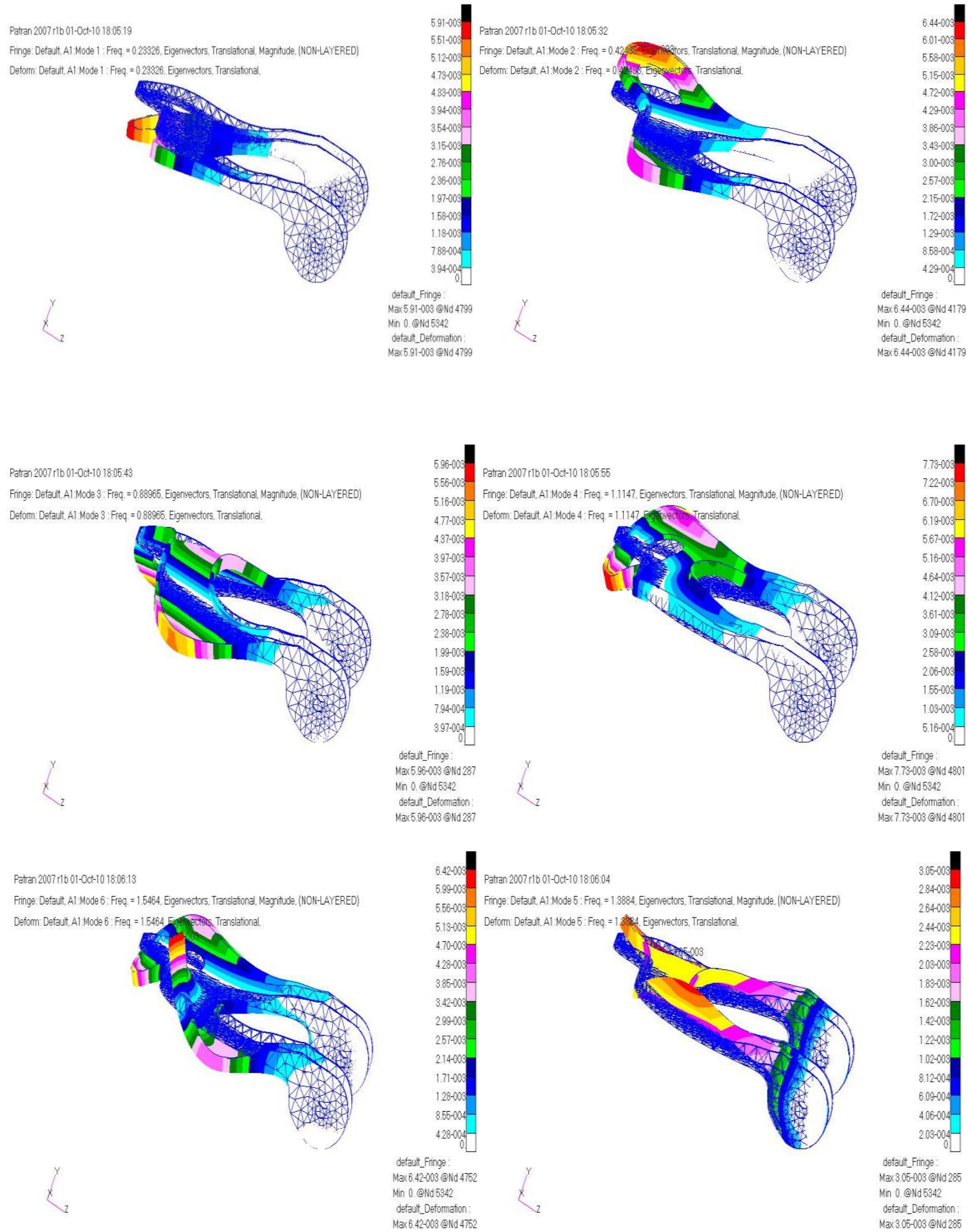
5.3 Identification of Mesh Convergence

The convergence of the stress was considered as the main criteria to select the mesh type. The finite element mesh was generated using TET10 for various mesh global length. Fig.9 shows the predicted results of stresses at the critical location of the suspension arm. It can be seen that the smaller the mesh size capture the higher predicted stresses. It is also observed that mesh size of 0.1 mm (54178 elements) has obtained the maximum stresses, which is almost flatter in nature. The maximum stress obtained of 50.3, 52.2 and 56.3 MPa for von Mises stress, Tresca and Maximum principal stress method respectively.

The maximum principal stress method occurred through the highest stresses along the global length range. Thus TET10 and maximum principal stress method are selected for linear static and dynamic analyses of the suspension arm.

5.4 Dynamic Analysis of Lower Arm

Dynamic analysis is focused on the eigen-frequencies and mode shapes. From a physical point of view an initial excitation of an undamped system causes to vibrate and the system response is a combination of eigenmodes, where each eigenmode oscillates at its associated eigen-frequency (Patrik, 2001). Modal analysis is usually used to determine the natural frequencies and mode shapes of a component. It can be used as the starting point for dynamic analysis. The finite element analysis codes usually used several mode extraction methods. The Lanczos mode extraction method is used in this study. Lanczos is the recommended method for the medium to large models. In addition to its reliability and efficiency, the Lanczos method supports sparse matrix methods that significantly increase computational speed and reduce the storage space. This method also computes precisely the eigenvalues and eigenvectors. The number of modes was extracted and used to obtain the suspension arm stress histories, which is the most important factor in this analysis. Using this method to obtain the first 10 modes of the suspension arm, which are presented in Table 4 and the shape of the mode are shown in Fig.11. It can be seen that the working frequency (80Hz) is far away from the natural frequency (233.26 Hz) of the first mode. A sample of the resulting eigenvalue/ eigenvector from the suspension arm is shown in Table 5.


Patran 2007 r1b 01-Oct-10 18:05:19
Fringe: Default, A1:Mode 1 : Freq. = 0.23326, Eigenvectors, Translational, Magnitude, (NON-LAYERED)
Deform: Default, A1:Mode 1 : Freq. = 0.23326, Eigenvectors, Translational,
default_Fringe :
Max 5.91-003 @Nd 4799
Min 0. @Nd 5342
default_Deformation :
Max 5.91-003 @Nd 4799
Patran 2007 r1b 01-Oct-10 18:05:32
Fringe: Default, A1:Mode 2 : Freq. = 0.4
Deform: Default, A1:Mode 2 : Freq. = 0.
default_Fringe :
Max 6.44-003 @Nd 4179
Min 0. @Nd 5342
default_Deformation :
Max 6.44-003 @Nd 4179
Patran 2007 r1b 01-Oct-10 18:05:43
Fringe: Default, A1:Mode 3 : Freq. = 0.88965, Eigenvectors, Translational, Magnitude, (NON-LAYERED)
Deform: Default, A1:Mode 3 : Freq. = 0.88965, Eigenvectors, Translational,
default_Fringe :
Max 5.96-003 @Nd 287
Min 0. @Nd 5342
default_Deformation :
Max 5.96-003 @Nd 287
Patran 2007 r1b 01-Oct-10 18:05:55
Fringe: Default, A1:Mode 4 : Freq. = 1.1147, Eigenvectors, Translational, Magnitude, (NON-LAYERED)
Deform: Default, A1:Mode 4 : Freq. = 1.1147
default_Fringe :
Max 7.73-003 @Nd 4801
Min 0. @Nd 5342
default_Deformation :
Max 7.73-003 @Nd 4801
Patran 2007 r1b 01-Oct-10 18:06:13
Fringe: Default, A1:Mode 6 : Freq. = 1.5464, Eigenvectors, Translational, Magnitude, (NON-LAYERED)
Deform: Default, A1:Mode 6 : Freq. = 1.5464
default_Fringe :
Max 6.42-003 @Nd 4752
Min 0. @Nd 5342
default_Deformation :
Max 6.42-003 @Nd 4752
Patran 2007 r1b 01-Oct-10 18:06:04
Fringe: Default, A1:Mode 5 : Freq. = 1.3884, Eigenvectors, Translational, Magnitude, (NON-LAYERED)
Deform: Default, A1:Mode 5 : Freq. = 1
default_Fringe :
Max 3.05-003 @Nd 285
Min 0. @Nd 5342
default_Deformation :
Max 3.05-003 @Nd 285

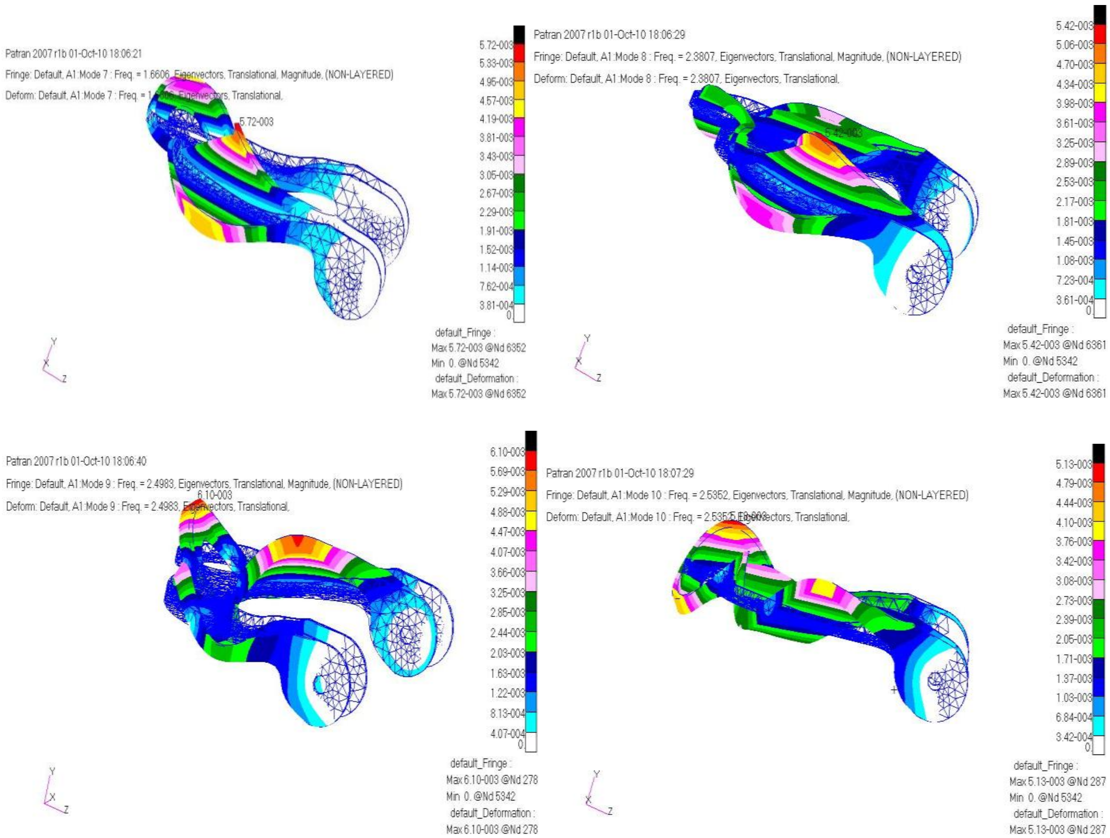


Fig.11. Frequency mode shape of lower arm

Table 4: Natural frequency of lower arm

No. of Mode	**Natural Frequency (Hz)**
1	233.26
2	424.88
3	889.65
4	1114.7
5	1388.4
6	1546.4
7	1660.6
8	2380.7
9	2498.3
10	2535.2

Table 5: Maximum displacements from modal analysis

Mode No	T1(µm)	T2(µm)	T3(µm)
1	146.89	5825.3	10135.5
2	629.77	6349.31	1088.46
3	1234.45	5807.32	950.36
4	1568.87	7615.65	1572.16
5	2854.58	1533.89	490.42
6	1377.12	6320.35	1781.14
7	1262.52	5506.094	1614.21
8	2178.52	5343.608	2623.39
9	1925.07	5991.516	2042.43
10	2586.68	4238.355	2572.20

6. Conclusion

A detailed model of suspension arm has been developed using finite element techniques. The tetrahedral elements (TET10) and tetrahedral elements (TET4) are used for the initial analysis. A TET10 was used for the solid mesh. Sensitivity analysis was performed to determine the optimum element size. it can be seen that the TET10 at mesh size 0.1 capture highest moment levels von Mises stress for this reason used to dynamic analysis. The results of the frequency are shown 10 modes and several deformation shapes and from the results proved that the control suspension arm model has been predicted the dynamic behavior.

References

Al-Asady, N.A., Abdullah, S., Arrifin, A.K., Rahmman, M.M. and Beden, S.M. (2008). Improving the automotive lower suspension arm durability using finite element analysis, Seminar on Engineering Mathematics. Universiti Malaysia Pahang of Department of Mechanical and Materials Engineering.

Attia, H.A. (2002). Dynamic Modeling of the Double Wishbone Motor Vehicle Suspension System. European Journal of Mechanics A/Solids, Volume 21: 167-174.

Baek, W.K., Stephens, R.I. and Dopker, B. (1993). Integrated computational durability analysis. Journal of Engineering for Industry, Transactions of the ASME 115(4): 492-499.

Conle, F.A. and Mousseau, C.W. (1991). Using vehicle dynamic simulation and finite element result to generate fatigue life contours for chassis component. International Journal of Fatigue, 13(3): 195 - 205.

Dimarogonas, A. (1996). Vibration for Engineers. Second Ed. New York: Prentice Hall.

Gopalakrishnan, R. and Agrawal, H.N. (1993). Durability Analysis of Full Automotive Body Structures. SAE International Congress and Exposition, Detroit, MI, March.

Kim, H.S., Yim, H.J. and Kim, C.M. (2002). Computational durability prediction of body structure in prototype vehicles. International Journal of Automotive Technology, 3 (4): 129-136.

Milliken, W.F. (2002). Chassis design: principles and analysis. Warrendale, Pa: Society of Automotive Engineering, Inc.

Patrik, H. (2001). FE-adaptivity and a nonlinear eigen problem algorithm. Department of Structural Mechanics, Chalmers University of Technology, Sweden.