Jurnal Mekanikal
December 2008, No. 26, 76 - 85

STRESS ANALYSIS OF HEAVY DUTY TRUCK CHASSIS AS A PRELIMINARY DATA FOR ITS FATIGUE LIFE PREDICTION USING FEM

Roslan Abd Rahman, Mohd Nasir Tamin, Ojo Kurdi*

Faculty of Mechanical Engineering,
Universiti Teknologi Malaysia,
81310 UTM, Skudai,
Johor Bahru

ABSTRACT

This paper presents the stress analysis of heavy duty truck chassis. The stress analysis is important in fatigue study and life prediction of components to determine the critical point which has the highest stress. The analysis was done for a truck model by utilizing a commercial finite element packaged ABAQUS. The model has a length of 12.35 m and width of 2.45 m. The material of chassis is ASTM Low Alloy Steel A 710 C (Class 3) with 552 MPa of yield strength and 620 MPa of tensile strength. The result shows that the critical point of stress occurred at the opening of chassis which is in contact with the bolt. The stress magnitude of critical point is 386.9 MPa. This critical point is an initial to probable failure since fatigue failure started from the highest stress point.

Keyword: *Stress analysis, fatigue life prediction, truck chassis*

1.0 INTRODUCTION

The age of many truck chassis in Malaysia are of more than 20 years and there is always a question arising whether the chassis is still safe to use. Thus, fatigue study and life prediction on the chassis is necessary in order to verify the safety of this chassis during its operation. Stress analysis using Finite Element Method (FEM) can be used to locate the critical point which has the highest stress. This critical point is one of the factors that may cause the fatigue failure. The magnitude of the stress can be used to predict the life span of the truck chassis. The accuracy of prediction life of truck chassis is depending on the result of its stress analysis. The more accurate result of stress analysis the more valid the predicted life of object. In this study, the stress analysis is accomplished by the commercial finite element packaged ABAQUS.

The automotive industry (vehicles and components) represents a strategic and important business sector in Malaysia. With the eventual trade liberalization of ASEAN Free Trade Area (AFTA), local automotive manufacturers and vendors shall require cars and components of world class standard. Noise and vibration are

* Corresponding author: E-mail: ojokurdi@yahoo.com

key elements in such standard. The automotive industry in Malaysia is much relying on foreign technology. Truck chassis, which is important structure of lightweight commercial vehicle, is mostly designed and imported from foreign country. In order to change this trend, it is necessary to develop and built Malaysian own chassis design. Study and research on truck chassis is thus required to achieve this goal.

The chassis of trucks is the backbone of vehicles and integrates the main truck component systems such as the axles, suspension, power train, cab and trailer. The truck chassis is usually loaded by static, dynamic and also cyclic loading. Static loading comes from the weight of cabin, its content and passengers. The movement of truck affects a dynamic loading to the chassis. The vibration of engines and the roughness of road give a cyclic loading. The existing truck chassis design is normally designed based on static analysis. The emphasis of design is on the strength of structure to support the loading placed upon it. However, the truck chassis has been loaded by complex type of loads, including static, dynamics and fatigue aspects. It is estimated that fatigue is responsible for 85% to 90% of all structural failures [1]. The knowledge of dynamic and fatigue behavior of truck chassis in such environment is thus important so that the mounting point of the components like engine, suspension, transmission and more can be determined and optimized.

Many researchers carried out study on truck chassis. Karaoglu and Kuralay [2] investigated stress analysis of a truck chassis with riveted joints using FEM. Numerical results showed that stresses on the side member can be reduced by increasing the side member thickness locally. If the thickness change is not possible, increasing the connection plate length may be a good alternative. Fermer et al [3] investigated the fatigue life of Volvo S80 Bi-Fuel using MSC/Fatigue. Conle and Chu [4] did research about fatigue analysis and the local stress-strain approach in complex vehicular structures. Structural optimization of automotive components applied to durability problems has been investigated by Ferreira et al [5]. Fermér and Svensson [6] studied on industrial experiences of FE-based fatigue life predictions of welded automotive structures.

Filho *et. al.* [7] have investigated and optimized a chassis design for an off road vehicle with the appropriate dynamic and structural behavior, taking into account the aspects relative to the economical viability of an initial small scale production. The design of an off-road vehicle chassis is optimized by increasing the torsional stiffness, maintenance of center of gravity, total weight of structure and simpler geometry for reduction of production cost. The integration of computer aided design and engineering software codes (Pro/Engineer, ADAMS, and ANSYS) to simulate the effect of design changes to the truck frame has been studied by Cosme et al [8].

Chiewanichakorn et al [9] investigated the behavior of a truss bridge, where an FRP deck replaced an old deteriorated concrete deck, using experimentally validated finite element (FE) models. Numerical results show that the fatigue life of the bridge after rehabilitation would be doubled compared to pre-rehabilitated reinforced concrete deck system. Based on the estimated truck traffic that the bridge carries, stress ranges of the FRP deck system lie in an infinite fatigue life regime, which implies that no fatigue failure of trusses and floor system would be expected anytime during its service life.

Ye and Moan [10] have investigated the static and fatigue behavior of aluminium box-stiffener/web frame connections using Finite Element Analysis (FEA) to provide a connection solution that can reduce the fabrication costs by changing the cutting shapes on the web frame and correspondingly the weld process meanwhile sufficient fatigue strength can be achieved. FE based fatigue was used to locate the critical point of probable crack initiation and to predict the life in a door hinge system [11].

In this study, stress analysis of heavy duty truck chassis loaded by static force will be investigated to determine the location of critical point of crack initiation as a preliminary data for fatigue life prediction of this truck chassis.

2.0 FINITE ELEMENT ANALYSIS OF TRUCK CHASSIS

2.1 Basic Concept of FEM

The finite element method (FEM) is a computational technique used to obtain approximate solutions of boundary value problems in engineering. Simply stated, a boundary value problem is a mathematical problem in which one or more dependent variables must satisfy a differential equation everywhere within a known domain of independent variables and satisfy specific conditions on the boundary of the domain [12].

An unsophisticated description of the FE method is that it involves cutting a structure into several elements (pieces of structure), describing the behavior of each element in a simple way, then reconnecting elements at nodes as if nodes were pins or drops of glue that hold elements together (Figure 1). This process results in a set of simultaneous algebraic equations. In stress analysis these equation are equilibrium equations of the nodes. There may be several hundred or several thousand such equations, which mean that computer implementation is mandatory [13].

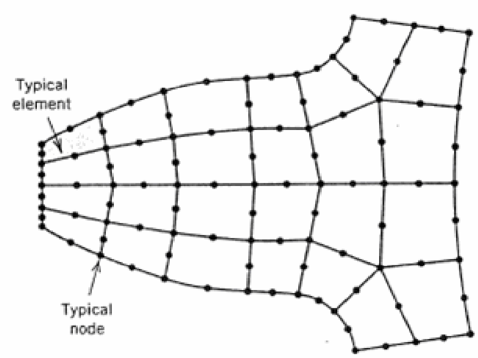

Figure 1: A coarse –mesh, two-dimensional model of gear tooth. All nodes and elements lie in plane of the paper [13]

2.2 A General Procedure for FEA

There are certain common steps in formulating a finite element analysis of a physical problem, whether structural, fluid flow, heat transfer and some others problem. These steps are usually embodied in commercial finite element software packages. There are three main steps, namely: preprocessing, solution and postprocessing. The preprocessing (model definition) step is critical. A perfectly computed finite element solution is of absolutely no value if it corresponds to the wrong problem. This step includes: define the geometric domain of the problem, the element type(s) to be used, the material properties of the elements, the geometric properties of the elements (length, area, and the like), the element connectivity (mesh the model), the physical constraints (boundary conditions) and the loadings [12].

The next step is solution, in this step the governing algebraic equations in matrix form and computes the unknown values of the primary field variable(s) are assembled. The computed results are then used by back substitution to determine additional, derived variables, such as reaction forces, element stresses and heat flow. Actually the features in this step such as matrix manipulation, numerical integration and equation solving are carried out automatically by commercial software [13].

The final step is postprocessing, the analysis and evaluation of the result is conducted in this step. Examples of operations that can be accomplished include sort element stresses in order of magnitude, check equilibrium, calculate factors of safety, plot deformed structural shape, animate dynamic model behavior and produce color-coded temperature plots. The large software has a preprocessor and postprocessor to accompany the analysis portion and the both processor can communicate with the other large programs. Specific procedures of pre and post are different dependent upon the program [12].

2.3 Truck definition and classification

Generally, truck is any of various heavy motor vehicles designed for carrying or pulling loads. Other definition of the truck is an automotive vehicle suitable for hauling. Some other definition are varied depending on the type of truck, such as Dump Truck is a truck whose contents can be emptied without handling; the front end of the platform can be pneumatically raised so that the load is discharged by gravity.

There are two classifications most applicable to Recreational Vehicle tow trucks. The first one is the weight classes, as defined by the US government, ranging from Class 1 to Class 8 as listed in Table 1 and Table 2. The second is classified into a broader category:

- Light Duty Truck
- Medium Duty Truck
- Heavy Duty Truck

Table 1: Classification and classes of truck

Weight Class	Minimum GVWR (lbs)	Maximum GVWR (lbs)	VIUS * Category	Common Category
Class 1		6,000	Light-duty	Light Duty
Class 2	6,001	10,000	Light-duty	Light Duty
Class 3	10,001	14,000	Medium-duty	Light Duty
Class 4	14,001	16,000	Medium-duty	Medium Duty
Class 5	16,001	19,500	Medium-duty	Medium Duty
Class 6	19,501	26,000	Light-heavy	Medium Duty
Class 7	26,001	33,000	Heavy-heavy	Heavy Duty
Class 8	33,001		Heavy-heavy	Heavy Duty

Table 2: Vehicle manufacturer truck classification

Category	Class	GVWR[1]	Representative Vehicles
Light	1	0 - 27 kN (0 - 6,000 lbs.)	pickup trucks, ambulances, parcel delivery
	2	27 - 45 kN (6,001 - 10,000 lbs.)	
	3	45 - 62 kN (10,001 - 14,000 lbs.)	
Medium	4	62 - 71 kN (14,001 - 16,000 lbs.)	city cargo van, beverage delivery truck, wrecker, school bus
	5	71 - 87 kN (16,001 - 19,500 lbs.)	
	6	87 - 116 kN (19,501 - 26,000 lbs.)	
	7	116 - 147 kN (26,001 to 33,000 lbs.)	
Heavy	8	147 kN and over (33,000 lbs. and over)	truck tractor, concrete mixer, dump truck, fire truck, city transit bus

Notes: Gross Vehicle Weight Rating (GVWR): weight specified by manufacturer as the maximum loaded weight (truck plus cargo) of a single vehicle.

2.4 Model of Truck Chassis

The model is depicted in Figure 2. The model has length of 12.35 m and width of 2.45 m. The material of chassis is ASTM Low Alloy Steel A 710 C (Class 3) with 552 MPa of yield strength and 620 MPa of tensile strength. The other properties of chassis material are tabulated in Table 3.

Table 3: Properties of truck chassis material [14]

Modulus Elasticity E (Pa)	Density ρ (kg/m^3)	Poisson Ratio	Yield Strength (MPa)	Tensile Strength (MPa)
207 x 10^9	7800	0.3	550	620

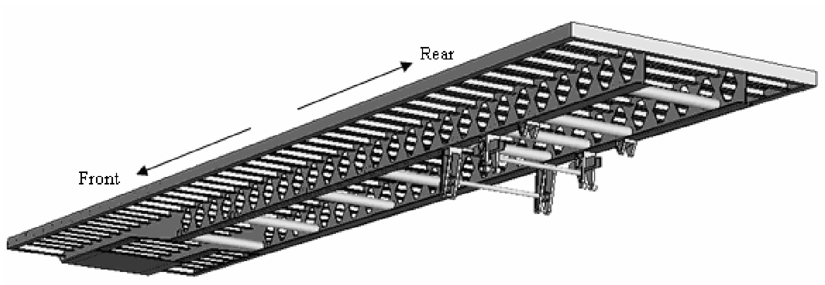

Figure 2: Model of truck chassis

2.5 Loading

The truck chassis model is loaded by static forces from the truck body and cargo. For this model, the maximum loaded weight of truck plus cargo is 36.000 kg. The load is assumed as a uniform pressure obtained from the maximum loaded weight divided by the total contact area between cargo and upper surface of chassis. Detail loading of model is shown in Figure 3. The magnitude of pressure on the upper side of chassis is calculated as 67564.6 N/m^2.

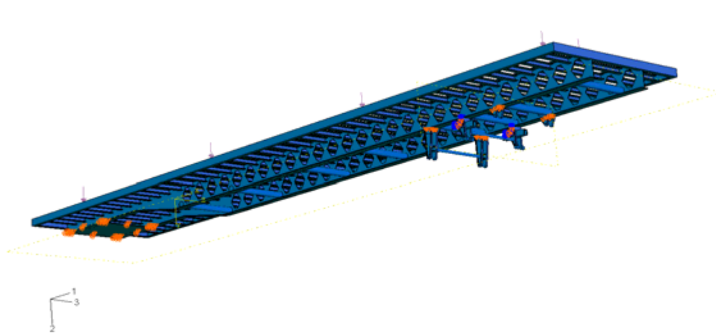

Figure 3: Static load (pressure = 67564.5 N/m^2)

2.6 Boundary Conditions

There are 3 boundary conditions (BC) of model; the first BC is applied in front of the chassis, the second and the third BC are applied in rear of chassis. They are shown in Figure 4. The type of BC 1 is pinned (the displacement is not allowed in all axes and the rotation is allowed in all axes) that represent the contact condition between chassis and cab of truck as shown in Figure 5(a). The BC 2 represents the contact between chassis and upper side of spring that transfer loaded weight of cargo and chassis to axle.

The contact condition of BC 2 in the object is shown in Figure 5. In the BC 2, the displacement only occurred in axis 2 and the rotation respect to all axes is zero. In the position where the BC 3 applied, there is a contact between inside surface of opening chassis and outside surface of bolt. In ABAQUS, this contact is called interaction. In this case, the type of the interaction is frictionless surface to surface contact. In the BC3, the displacement and the rotation is zero in all axes on all of bolt's body. This condition is called fixed constrain. The bolt in BC 3 was assumed perfectly rigid. This assumption was realized by choosing a very high Young's Modulus value of the bolt properties.

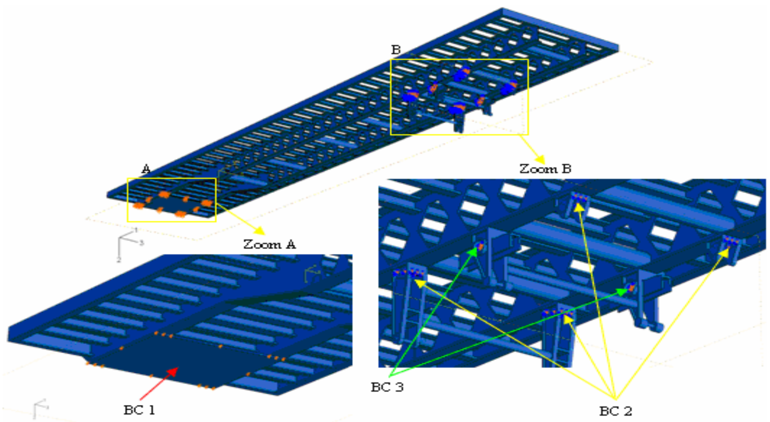

Figure 4: Boundary conditions representation in the model

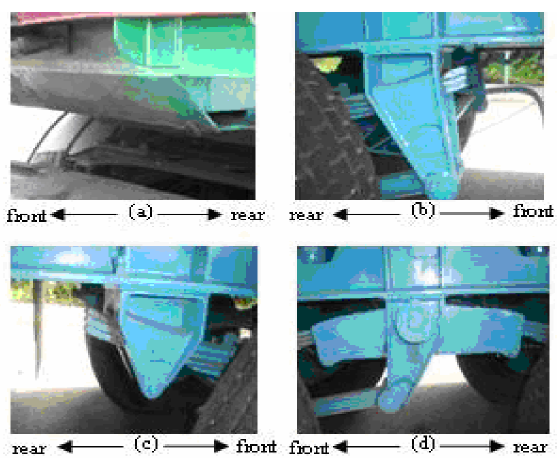

Figure 5: Boundary conditions representation in the object, 4(a) BC 1, 4(b) and 4(c) BC 2, 4(d) BC 3

3.0 RESULTS AND DISCUSSION

The location of maximum Von Misses stress is at opening of chassis which is contacted with bolt as shown in Figure 6. The stress magnitude of critical point is 386.9 MPa. This critical point is located at element 86104 and node 16045. The internal surface of opening of chassis was contacted with the very stiff bolt. The BC 3 is also a fixed constraint, thus it cause a high stress on it. Based on static safety factor theory, the magnitude of safety factor for this structure is 1.43. The formula of Safety Factor (SF) is defined by [11]:

$$SF = \frac{\text{significant strength of material}}{\text{corresponding significant stress from normal load}} \tag{1}$$

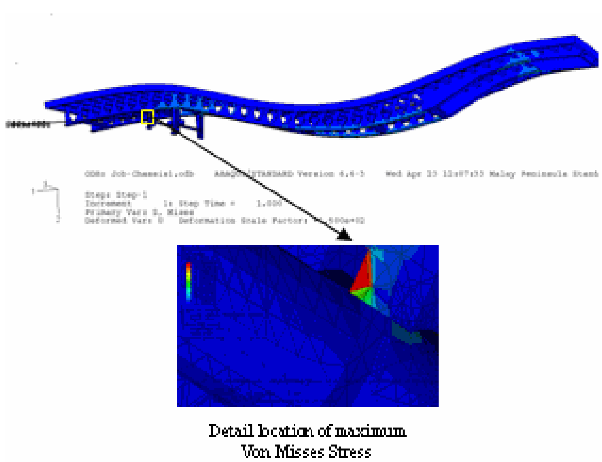

Figure 6: Von Misses stress distribution and critical point location

Vidosic [15] recommends some value of safety factor for various condition of loading and material of structures. He recommends the value of 1.5 to 2 for well known materials under reasonably environmental condition, subjected to loads and stresses that can be determined readily. Based on this result, it is necessary to reduce the stress magnitude of critical point in order to get the satisfy SF value of truck chassis. The truck chassis can be modified to increase the value of SF especially at critical point area.

The displacement of chassis and location of maximum displacement is shown in Figure 7. The magnitude of maximum displacement is 4.995 mm and occurs at middle of chassis. Maximum deflection is occurred at the middle of BC 1 and BC 2.

For validation purpose, the region between BC 1 and BC 2 of chassis where the highest stress occurred is approximated by one dimensional simple beam loaded by concentrated force at mid point. The uniform distributed pressure on this region is replaced by a single concentrated force at mid point. The magnitude of the single force is obtained by multiplying the magnitude of pressure with the total area where the pressure is applied. The result agrees well with this approximation.

The approximation result shows that the displacement of this simple beam is located in the midpoint of beam with magnitude of:

$$\delta_{max} = \delta(L/2) = \frac{PL^3}{48EI} \tag{2}$$

$$= \frac{71145.5(8.1^3)}{48(207 x 10^9) x (8.6x10^{-4})}$$

$$= 4.43 \text{ mm}$$

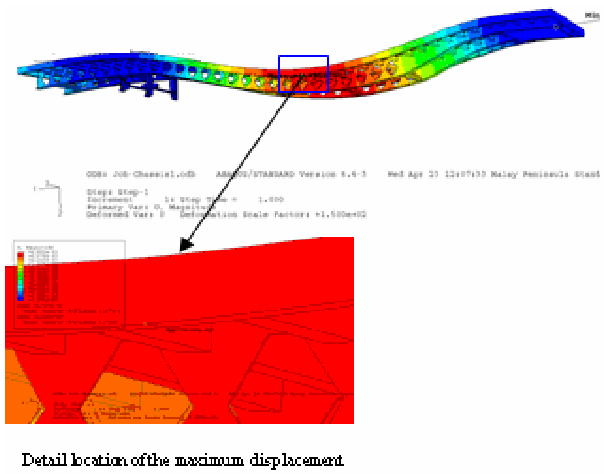

Figure 7: Displacement distribution and the maximum displacement location

The maximum displacement of numerical simulation result is 4.99 mm. The result of numerical simulation is bigger 11.2 % than the result of analytical calculation

4.0 CONCLUSION

Numerical analysis result shows that the critical point of stress occurred at opening of chassis which is in contacted with the bolt. The magnitude of highest stress is critical because the value of SF is below than the recommended value. Since fatigue failure started from the highest stress point, it can be concluded that this critical point is an initial to probable failure. Thus, it is important to take note to reduce stress magnitude at this point. The location of maximum deflection agrees well with the maximum location of simple beam loaded by uniform distribution force.

REFERENCES

1. MSC. Fatigue, 2003. [Encyclopedia]. Los Angeles (CA, USA): MacNeal Schwendler Corporation.

2. Karaoglu, C. and Kuralay, N.S., 2000. *Stress Analysis of a Truck Chassis with Riveted Joints*, Elsevier Science Publishers B.V. Amsterdam, the Netherlands, Vol. 38, 1115-1130.
3. Fermer, M., McInally, G. and Sandin, G., 1999. Fatigue Life Analysis of Volvo S80 Bi-Fuel using MSC/Fatigue, *Worldwide MSC Automotive Conference*, Germany.
4. Conle, F.A. and Chu, C.C., 1997. Fatigue Analysis and the Local Stress-strain Approach in Complex Vehicular Structures, *International Journal of Fatigue.*
5. Ferreira, W.G., Martins, F., Kameoka, S., Salloum, A.S. and Kaeya, J.T., 2003. Structural Optimization of Automotive Components Applied to Durability Problems, *SAE Technical Papers.*
6. Fermér, M. and Svensson, H., 2001. Industrial Experiences of FE-based Fatigue Life Predictions of Welded Automotive Structures, *Fatigue & Fracture of Engineering Materials and Structures* 24 (7), 2001, 489-500.
7. Filho, R.R.P., Rezende, J.C.C., Leal, M. de F., Borges, J.A.F., 2003. Automotive Frame Optimization, *12th International Mobility*
8. Cosme, C., Ghasemi, A. and Gandevia, J., 1999. Application of Computer Aided Engineering in the Design of Heavy–Duty Truck Frames, *International Truck & Bus Meeting & Exposition*, Detroit, Michigan, November 15 – 17.
9. Chiewanichakorn, M., Aref, A.J., Allampalli, S., 2007. Dynamic and Fatigue Response of a Truss Bridge with Fiber Reinforced Polymer Deck, *International Journal of Fatigue*, 29, 1475–1489.
10. Ye, N. and Moan, T., 2007. Static and Fatigue Analysis of Three Types of Aluminium Box-Stiffener/Web Frame Connections, *International Journal of Fatigue,* 29, 1426–1433.
11. Bekah, S., 2004. Fatigue Life Prediction in a Door Hinge System Under Uni-Axial and Multiaxial Loading Condition, *Master Thesis*, Ryerson University, Toronto, Ontario, Canada.
12. Hutton, David, V., 2004. *Fundamental of Finite Element Analysis,* Mc Graw Hill, New York.
13. Cook, Robert, D., 1995. *Finite Element Modeling for Stress Analysis*, John Willey & Sons, Inc, New York.
14. Juvinall, R.C. and Marshek, K.M., 2006. *Fundamental Machine Component Design*, John Willey & Son, Inc., USA.
15. Vidosic, J.P., 1957. *Machine Design Project,* Ronald Press, New York.