THOMSON REUTERS ENDNOTE
[Choi , 4(1): January 2017] ISSN 2348 – 8034
DOI- 10.5281/zenodo.244915 Impact Factor- 4.022

GLOBAL JOURNAL OF ENGINEERING SCIENCE AND RESEARCHES

DYNAMIC VIBRATION CHARACTERISTICS OF ELECTRIC AGRICULTURAL VEHICLE BASED ON FINITE ELEMENT METHOD

Won-Sik Choi[*1], Pandu Sandi Pratama[2], Destiani Supeno[3], Ji-Hee Woo[4], Eun-Sook Lee[5], Sung-Won Chung[6] & Chun-Sook Park[7]

[*1,2,3,4,5&6]Department of Bio-Industrial Machinery Engineering, Pusan National University, Republic of Korea

[7]CEO of Keunwoo Tech Co. Ltd., Republic of Korea

ABSTRACT

An agriculture machines such as electric vehicle which is used in the agricultural field are subjected to different loads and vibration. Due to this vibration there will be certain deformations which affect the performance of the electric vehicle in adverse manner. This paper proposed a vibration analysis of four wheel drive agricultural electric vehicle using ANSYS. The Finite Element Method (FEM) analysis is carried out to study the effect of vibration on the cart in order to ensure the safety. This work helps the electric vehicle developer make a better product at the early design stage with lower cost and faster development time. To do this, firstly, using CATIA, a CAD model is prepared. Secondly, the analysis is to be carried out using ANSYS 15. The modal analysis and random vibration analysis of the frame was conducted. The analysis shows that the proposed design was successfully shows the minimum deformation when the vibration was applied in normal condition.

Keywords: *Agricultural Vehicle, Electric Vehicle, Lift and dump, Vibration analysis.*

I. INTRODUCTION

Electric agricultural vehicle is kind of mechanical and electrical integration of transportation vehicle, and an important tool in agriculture modernization. With the rapid development of economy in Korea, electric vehicle is widely used in agriculture area. According to the market demand, in this research a small sized electric vehicle with lift and dump capability was developed. The structural analysis is one of important step in the development process such as in [1~2]. In agricultural environment is usually used in bad working condition, uneven road surface will lead the vehicles to the forced vibration. If the excitation vibration frequency and the resonance natural frequency of the frame structure are close, the mechanical structure will produce local resonance and deformation. In previous studies conducted by Ferreira and Lopez-Pita [3] and analyses by [4~5], it conclude that if excitation signal frequency and the response frequency of the structure is similar, the mechanical structure will produce resonance and deflection. Therefore, the simulation related to the vibration analysis is needed in vehicle development process. There are several research related to vibration analysis. Vibration analysis can be done experimentally such as in [6~7], but the cost of conducting using this method is expensive compared to simulation based analysis. In [8] random vibration analysis for the chassis frame of hydraulic truck based on ANSYS was reported. Moreover, in [9~10] ANSYS was used to determine the natural frequencies of space frame race car chassis through modal and harmonic analysis. However, different chassis structure produces different dynamic characteristic, therefore it is important to study dynamic characteristic to ensure the structure and vibration failures do not occur during the chassis operating life.

In this research the modal analysis was performed to find out the natural frequency of the electric agricultural vehicle. First natural frequency or resonance is one of the contributing factors for vibration and noise related problems that occur in structures and machineries. With advancement in computation power, finite element analysis simulation is possible and become complement of experiment based analyses. Furthermore, the free vibration analysis was performed to understand the effect of vibration due to daily activities to the frame structure of the vehicle. ANSYS's free vibration analysis can simulate a structural response under forced vibration from uneven load surface and the software efficiently extract the forced excitation amplitude deflection of the structure, which later can be used to predict and prevent damage and large deflection to the structure.

ISSN 2348 – 8034
Impact Factor- 4.022

II. METHOD & MATERIAL

Proposed System

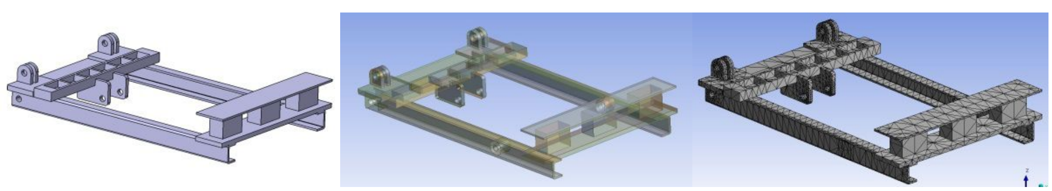
The developed electric vehicle is shown in Fig. 1. The dimension of the cart was calculated optimally to maximize the capacity and the material strength. The vehicle has ability to lift up and to dump the load. The cart was design with total body weight 300kg and total load capacity 300kg. This vehicle powered by 24V 400W DC motor with gear ration 30:1. For lifting and dumping, 2 hydraulic cylinders with diameter 50mm were used. In standard condition the height of trunk is 770 mm, but in lift condition the height of trunk is 1240 mm. This configuration makes sure the product can be transfer easily from the electric vehicle to the other transport vehicle as shown in Fig.1a.

a. Electric vehicle in lift and dump position *b. upper side frame of dumper*

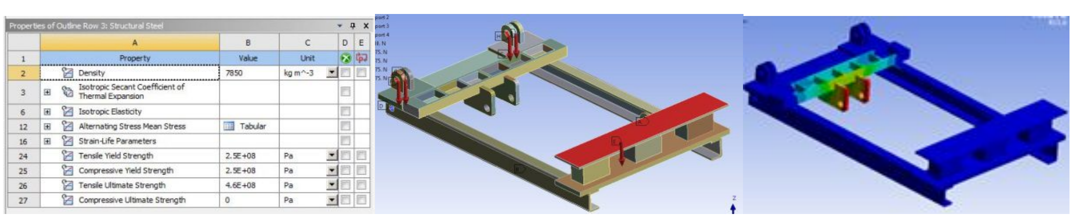
Fig. 1. The developed electric vehicle

Method

This paper focused on the upper side frame of dumper as shown in Fig. 1b since this part is the most affected part during the vehicle operation. The procedure of static analysis includes following steps as shown in Fig. 2. Firstly, the 3D model of important component of the vehicle was created. In this research focused on the middle frame since this part affecting hardly by the load and lifting condition. Secondly, the model was simplified to obtained geometry model. Thirdly, the mesh of the model was generated. Fourthly, the materials properties were defined. Fifthly, the boundary condition was defined. Finally, solve the problem, visualized and read the results.



a. Virtual 3D model *b. Geometry model* *c. FEM model*



d. Define material *e. Boundary condition* *f. Visualization result*

Fig. 2. Simulation analysis procedure

III. RESULT AND DISCUSSION

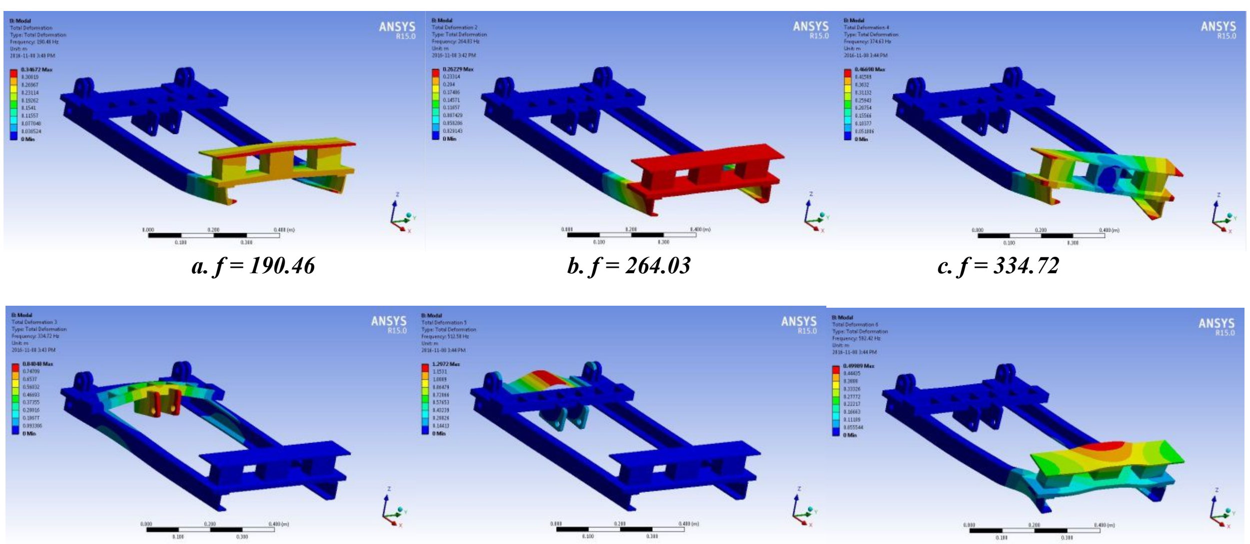
Modal Analysis

Modal analysis is a kind of linear analysis technology, used to determine the structure of the natural frequency and vibration mode. ANSYS provides seven modal extraction methods, this paper uses Block Lanczos method to extract six order modal of chassis frame that have pre-stressed firstly. In modal analysis, the only effective load is zero displacement constraints, if in a certain degree of freedom (DOF) specifies a non-zero displacement constraints, the program will replace the degree of freedom by zero displacement constraints.

The calculation result of modal analysis is shown in Table 1. Modal analysis results in Table 1 shows that the frame of 1 ~ 6 orders modal natural frequency is 190 ~ 600 Hz range. Fig. 3 shows the deformation effect on the frame when the natural frequency occurred.

Table 1. Each order's natural frequency and vibration mode characteristics of the frame

Number	Inherent frequencies	Characteristics of modal
1	190.46	Vertical upward along Z axis on the front side
2	264.03	Vertical upward along Z axis on the front side
3	334.72	Vertical upward along Z axis on the bottom side
4	374.63	Twisting the front side along X axis
5	512.58	Vertical upward along Z axis on the rear side
6	592.42	Bending on the front side



a. f = 190.46 *b. f = 264.03* *c. f = 334.72*

a. f = 190.46 *b. f = 264.03* *c. f = 334.72*

Fig. 3. Simulation result of modal analysis

Random vibration analysis

Random vibration analysis is used to determine the structure response under random loading. ANSYS uses the power spectral density (PSD) spectrum as random vibration analysis of the load input. Power spectral density is a kind of probability statistics method, and is the root mean square value of random variables, including a measure of

ISSN 2348 – 8034
Impact Factor- 4.022

the random vibration energy and frequency information. Power spectrum can be displacement, velocity, acceleration or force power spectral density and other forms. Based on the former modal analysis, power spectral density uses displacement power spectral density, the spectrum value according to the national standard "Vehicle Vibration Input Pavement Level Representation Method" (GB7031-1986), shows in Table 2.

Table 2. Time power spectral density

f/Hz	1.05	2.1	4.15	8.3	16.6	33.2
Gd(f)/10-6 m2/Hz	78.96	19.74	4.93	1.23	0.31	0.08

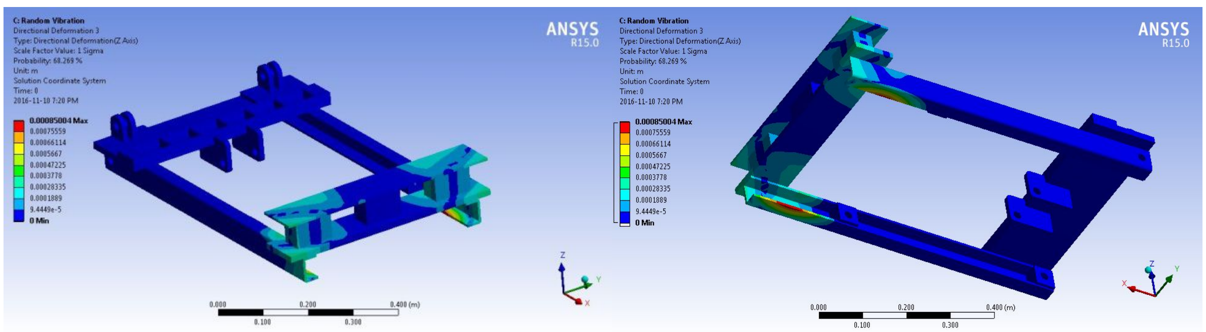


Fig. 4. Simulation result of random vibration analysis

Fig. 4 shows that the maximum deformation is 0.00085004m. This value is very small therefore the effect can be neglected during long period of operation. The most affected area is the front side of frame. Table 3 shows that the vibration frequency induces by the pavement road are between 1.05 Hz until 33.2 Hz. However, the natural frequency of frame is 190 ~ 600 Hz range. Therefore, the vibration induce by the pavement road is far from the vehicle natural frequency.

IV. CONCLUSION

In this research, natural frequency analysis and random vibration analysis was done to the middle frame of the electric agricultural vehicle. The simulation result show that the natural frequency value of the first order and second order frame are among sensitive frequency value ranges which can be obtained from modal analysis. Modal analysis results show that the frame of 1 ~ 6 orders modal natural frequencies range are 190 ~ 600 Hz. This should be the key consideration to avoid produce resonance. For the vehicle driving in the pavement road, the random vibration analysis shows that the vibration frequency generated by the road is lower than the first order natural frequency of the frame. Therefore, the operation of the vehicle on the pavement only generated small deformation. The maximum deformation is 0.85 mm. Large displacement and stress mainly concentrated in the bottom front side of the frame by random vibration stress and displacement.

V. ACKNOWLEDGEMENTS

This research was supported by Ministry of Agriculture, Food and Rural Affairs Research Center of Republic of Korea Government.

[Choi , 4(1): January 2017] ISSN 2348 – 8034
DOI- 10.5281/zenodo.244915 Impact Factor- 4.022

REFERENCES

1. *Wang G.S., Si J.P., Chen Y.Y., Guo L.N., and Zhao Y.N., "Structural Stability Analysis of Dump Truck Body Based on ANSYS", Advanced Materials Research, Vol. 588, pp. 217-221, 2012.*
2. *Si F.J.P., Wang G.S., and Ding X.L., "Analysis of Dump Truck Unloading Condition Based on ANSYS", Advanced Materials Research, Vol. 548, pp. 724-729, 2012.*
3. *Ferreira, P.A. and A. Lopez-Pita, "Numerical Modeling of high-speed train/track system to assess track vibrations and settlement prediction" J. Trans. Eng., Vol. 139, pp. 330-337, 2012.*
4. *Gou-Lin, X. and Z. Dai-Sheng. "Analysis of BIW and its shape correlation research". J. Automobile Sci. Technol., Vol.10, pp. 39-43, 2008.*
5. *Yang, Y., W.W. Liou, J. Sheng, D. Gorsich and S. Arepally, "Shock wave impact simulation of a vehicle occupant using fluid/structure/dynamics interactions" Int. J. Impact Eng., Vol. 52, pp. 11-22, 2012.*
6. *He, J. and Z.F. Fu, "Modal Analysis" 1st Edn., Butterworth-Heinemann. Oxford, Boston, ISBN-10: 0080511783, pp. 304. 2001*
7. *Wenlin, W., S. Cheng, L. Mingjun and Z. Jun, "Experimental modal test for structure refinement of an automotive chassis" Proceedings of the International Conference on Measuring Technology and Mechatronics Automation, Mar. 13-14, IEEE Xplore Press, Changsha City, pp. 174-177, 2010..*
8. *Wang H.F, Jia K.K, and Guo Z.P, "Random Vibration Analysis for the Chassis Frame of Hydraulic Truck Based on ANSYS", Journal of Chemical and Pharmaceutical Research, Vol. 6(3), pp. 849-852, 2014.*
9. *Marzuki M.A, Halim M.H.A, Mohamed A.R.N, "Determination of Natural Frequencies through Modal and Harmonic Analysis of Space Frame Race Car Chassis Based on ANSYS.*
10. *Marzuki M.A, Bakar M.A.A., and Azmi M.F.M., "Designing Space Frame Race Car Chassis Structure Using Natural Frequencies Data from Ansys Mode Shape Analysis", International Journal of Information System and Engineeering, Vol. 1, pp. 1-10, 2015.*