WFL Publisher
Science and Technology
Meri-Rastilantie 3 B, FI-00980
Helsinki, Finland
e-mail: info@world-food.net

Journal of Food, Agriculture & Environment Vol.8 (2): 359-364. 2010
www.world-food.net

Analysis and design optimization of a frontal combine harvester axle using finite element and experimental methods

Majid Khanali *, Ali Jafari, Hossein Mobli and Ali Rajabipour
Department of Agricultural Machinery Engineering, Faculty of Agricultural Engineering and Technology, University College of Agriculture and Natural Resources, University of Tehran, P.O. Box 4111, Karaj 31587-77871, Islamic Republic of Iran.
**e-mail: khanali@ut.ac.ir*

Received 16 January 2010, accepted 29 March 2010.

Abstract
JD 955 combine manufactured by Iran Combine Harvester Manufacturing Company was modified. Total weight and the loads applied on front axle of modified combine were increased. The commercial finite element package ANSYS V9.0 was used for the solution of the problem. Based on finite element analysis, redesign of front axle was carried out for increasing mechanical strength, easy manufacturability and cost reduction. Results of static, modal and transient analysis of proposed front axle under loading due to modified combine showed that the proposed model is suitable to install on the modified combine. Results obtained by numerical method were verified experimentally. The strain measurement was carried out using strain gauges. The calculated values of strain by numerical method were in good agreement with experimental data.

Key words: Front axle, stress analysis, optimization, combine harvester, finite element method.

Introduction
The urgent issues for industrial companies today are how to reduce the time and cost required for developing a new product [1, 2]. Accordingly, they have tried to use the computer's vast memory capacity, fast processing speed, and user-friendly interactive graphics capability to automate and tie together otherwise cumbersome and separate engineering or production tasks, thus reducing the time and cost of product development and production. Computer-aided design (CAD), computer-aided manufacturing (CAM) and computer-aided engineering (CAE) are the technologies used for this purpose during the product cycle [3].

Iran Combine Manufacturing Company (ICMCo) is committed to excellence and competitiveness in the national market. This company had manufactured JD 955 combines many years ago in Iran. ICMCo modified existing product in order to improve its product quality and efficiency while reducing development time in the existing combine [4]. Therefore, the loads applied on the front axle of combine were increased due to the modification in the existing system. Front axle is one of the major and very important components that need a thorough design, as this part experiences the worst load condition of the whole combine. Front axle of JD 955 combines has no field failure reports.

Leon *et al.* [5] used experimental and numerical methods for the stress analysis of a frontal truck axle beam. The results obtained by finite element method were verified experimentally using photo-stress. Mahanty *et al.* [6] performed an experimental and numerical analysis of a tractor's front axle. Based on finite element analysis results, redesign was carried out for the front axle for weight optimization and easy manufacturability. Five different models were proposed based on ease of manufacturing and weight reduction. The results obtained by finite element method were analyzed by thirteen different certification test load conditions. Maly and Bazzaz [7] used experimental and numerical methods for design change from casting to welding for an axle casing. Nanaware and Pable [8] used finite element method for failure analysis of rear axle shafts of 575 DI tractors. Topac *et al.* [9] studied a premature failure that occurs prior to the expected load cycles during the vertical fatigue tests of a truck rear axle housing prototype. In these tests, crack mainly originated from the same region on test samples. To determine the reason of the failure, a detailed CAD model of the housing was developed. Mechanical properties of the housing material were determined via tensile tests. Using these data, stress and fatigue analyses were performed by finite element method. Fatigue crack initiation locations and minimum number of load cycles before failure initiation were determined. Results provided from tests were compared with the analyses.

The main objective in the current study was to analyze the front axle of JD 955 combine under static loading conditions resulted from the applied modifications and investigate the mechanical strength of front axle of combine under new loading condition and arrive at a conclusion whether existing front axle can do the job or total redesign of the component is required. It was shown that the front axle of JD 955 combine is not strong enough to be installed on the modified combine and it is necessary to optimize the existing design of the front axle of JD 955 combine in order to install on modified combine. Therefore, the analysis and optimization of front axle of the modified combine was performed. Dynamic analysis of optimized front axle has to be performed for investigating induced stress and strain due to cyclic loading conditions of field operation. Results obtained by numerical method were verified experimentally in order to investigate the accuracy of finite element analyses.

Materials and Methods

Geometry of current front axle: All vehicles are subjected to both static and dynamic loads. It is very difficult to determine the exact types and locations of loads that are applied on the component. Current front axle of combine is considered as a support for front wheels, hydraulic cylinders of the head of combine, gearbox, bodywork or superstructure etc. The front axle of combine is an assembly of lower and upper boxes, right and left side bodyworks bases, right and left connection plates, hydraulic cylinders supports and gearbox supports (Fig. 1).

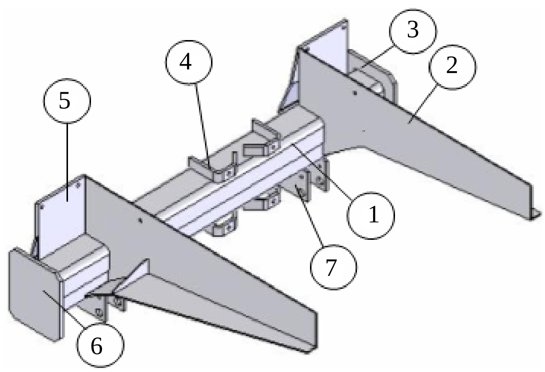


Figure 1. Three dimensional model of front axle of JD 955 combine: (1) lower and upper boxes, (2) right side bodywork base, (3) right connection plate, (4) gearbox supports, (5) left side bodywork base, (6) left connection plate, (7) hydraulic cylinders support.

Static loads on front axle: Static loads that are applied on front axle of JD 955 combine are as follows: vertical right bodywork and left bodywork loads, vertical loads on the gearbox supports, head lifting loads from hydraulic cylinders. A simplified diagram of all kinds of load is shown (Fig. 2). The magnitudes of these forces were determined by experimental and theoretical methods.

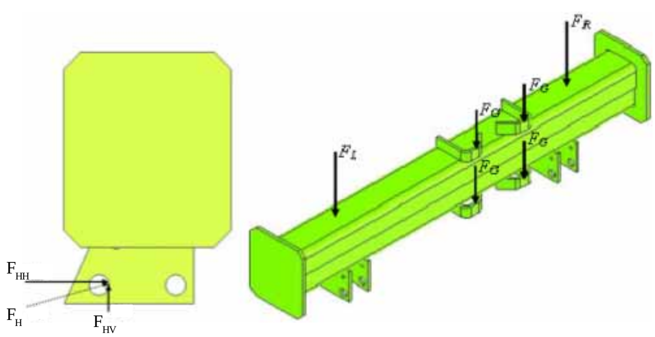


Figure 2. Loads applied on front axle of JD 955 combine: F_H head lifting loads from hydraulic cylinders, F_{HH} horizontal component of F_H, F_{HV} vertical component of F_H, F_G vertical load on the gearbox supports, F_R vertical right bodywork load, F_L vertical left bodywork load.

Stress analysis of front axle of JD 955 combine harvester under static loading: The commercial finite element package ANSYS V9.0 was used for the solution of the problem [10]. Several simplifications of the model structure have been made with the purpose of reducing the analysis time and size of model. To build the finite element model, the front axle was modeled using SOLID 82, a two dimensional elements, and SOLID 95, a hexahedral three dimensional element. At first the cross section areas of the lower box, upper box, hydraulic cylinder supports and connection plates were meshed using SOLID 82 elements and then cross section areas were extruded using the SOLID 95 elements. The individual components have been coupled together so that there is no free motion between components.

The next step was the definition of the boundary conditions. All degrees of freedom are constrained at the nodes placed on the connection plates of the model. Then the loads that had to be previously described and defined were applied on the model.

The static analysis of front axle was carried out after several sensitivity analyses; elements with average size of 10 mm were used. The finite element model of front axle of JD 955 combine consisted of 19,000 elements and 123,000 nodes (Fig. 3).

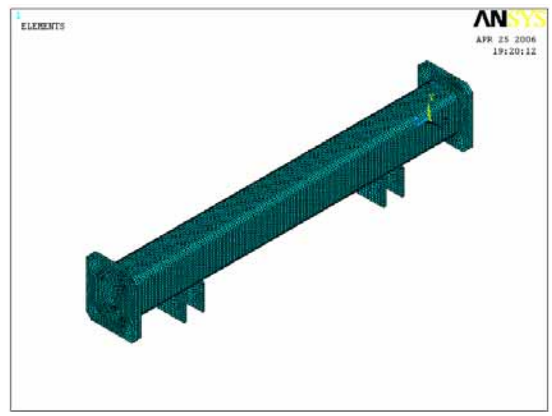


Figure 3. Finite element model of front axle of JD 955 combine.

After obtaining the solution, the results of analysis can be reviewed using post processing to determine maximum induced stress and its location. In designing parts to resist failure, it is assumed that the internal stresses do not exceed the strength of the material. If the material to be used is ductile, then it is the yield strength that designer is usually interested in, because a permanent deformation would constitute failure. The distortion energy theory is also called the Von Mises theory, which is the most suitable theory to be used in ductile materials [11].

Von-Mises stress σ' is calculated by using the formula:

$$\sigma' = \left[\frac{(\sigma_1-\sigma_2)^2+(\sigma_2-\sigma_3)^2+(\sigma_1-\sigma_3)^2}{2}\right]^{1/2}$$

where σ_1, σ_2 and σ_3 are principal stresses associated with the three principal directions. According to distortion energy theory, allowable stress in order to avoid fracture is equal to yield stress strength. Factor of safety can be calculated by dividing yield stress to maximum Von-Mises stress.

Optimization of front axle of modified combine: The results of stress analysis of front axle of JD 955 combine harvester under static loading showed that the front axle of JD 955 combine is not strong enough to be installed on the modified combine. Consequently the optimization of the current model of axle was carried out. During the optimization process the evaluation of the different alternative combination of product design parameter is carried out to achieve one or several objective functions. Furthermore a search strategy has to be defined to find the combination of design parameters that fulfills the objective functions.

Three different optimization procedures exist: a) size, b) shape and c) topology. In this study the procedure applied was for the size optimization. For this purpose, based on the resulting stress map, the thickness of the lower and upper boxes was increased, strengthening the place where maximal stress appeared. In order to eliminate the effect of eccentric loads due to vertical bodywork bases (Fig. 1), the horizontal bodywork bases were used instead of vertical ones (Fig. 4).

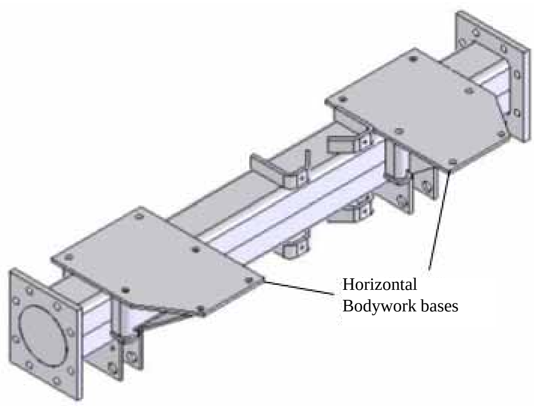


Figure 4. Three dimensional model of optimized front axle.

Material data: The current front axle and optimized front axle were made from St 37-2 with following material properties: E = 200,000 MPa Young's modulus, $\nu = 0.3$ Poisson's ratio, $\rho = 8000$ kg/m^3 density, $\sigma_y = 235$ MPa yield stress and $\sigma_u = 340$ MPa limit stress.

Static analysis of optimized front axle: The finite element model of the optimized front axle required in the analyses was composed via commercial finite element package ANSYS V9.0 preprocessing environment. The optimized front axle was modeled with SOLID 82 and SOLID 95 that have been previously described. Then all degrees of freedom are constrained at the nodes placed on the connection plates of the model. The next step was the definition of the loads that had to be previously described and defined. The static analysis of front axle was carried out after several sensitivity analyses; elements with average size of 10 mm were used. The size of the finite model is approximately 15,000 elements and 100,000 nodes.

Dynamic analysis of optimized front axle: During use in the field, ground profile is an input of vertical excitation to combine. Therefore, the front axle is subjected to cyclic loading conditions of field operation. There are two general classes of vibrations, free and forced. Free vibration takes place when a system oscillates under the action of forces inherent in the system itself, and when external impressed forces are absent. The system under free vibration will vibrate at one or more of its natural frequencies, which are properties of the dynamical system established by its mass and stiffness distribution. Vibration that takes place under the excitation of external forces is called forced vibration. When the excitation is oscillatory, the system is forced to vibrate at the excitation frequency. If the frequency of excitation coincides with one of the natural frequencies of the system, a condition of resonance is encountered and large dangerous oscillations may result. Thus, the calculation of the natural frequencies is of major importance in the study of vibrations [12].

Modal analysis of optimized front axle: Modal analysis is used to determine the vibration characteristics (natural frequencies and mode shapes) of a structure or a machine component while it is being designed. For performing modal analysis, finite element model of optimized front axle was used. The applied boundary condition in this analysis was as similar as boundary conditions applied on the static analysis. After obtaining the solution, the results of analysis can be used to determine the natural frequencies and their appropriate mode shapes. When a vehicle such as combine is traveling over a rough road, the value of excitation frequency (frequency of harmonic loads that applied on front axle) is calculated by the formula:

$$f = \frac{V}{\lambda}$$

where, V is the speed of the vehicle measured in m per second; λ is the wavelength of the harmonic motion, measured in m. In the above equation the quantity f is obtained in cycles per second.

Considering the typical row spacing range from 150 to 400 mm in drill seeders [13], the field ground profile is modeled with a sinusoidal wave with a wavelength of 0.3 m and amplitude of 0.15 m. considering the combine speed ranges from 1.2 to 20 km/h, the maximum value of corresponding frequency is calculated as below:

$$f = \frac{20}{3.6 \times 0.3} = 18.52 Hz$$

Therefore, a frequency range from zero to 20 Hz is considered for frequency range of harmonic loads.

Transient analysis of optimized front axle: When a dynamical system is excited by a suddenly applied non-periodic excitation, the response to such excitation is called transient response, since steady state oscillations are generally not produced [12]. In transient analysis, front axle of combine is considered as a beam that was subjected to two dynamic right and left bodyworks loads (Fig. 5).

The beam was modeled with a two-dimensional element, BEAM3. Some geometrical attributes of cross-section area of axle is used as real constants of this element. After this step, the dynamic loads were applied on the model. A dynamic loading condition consists of two load steps (Fig. 6).

In the first load step, the force increased during the rise time, the value of force at the end of the first step, reached the maximum value. During the second load step, value of load remained constant at maximum value. Rise time and maximum value of two dynamic loads applied on the axle were calculated by considering some assumptions. Assuming the combine falls from 0.5 m height, its speed when reaching the ground, was calculated as follows:

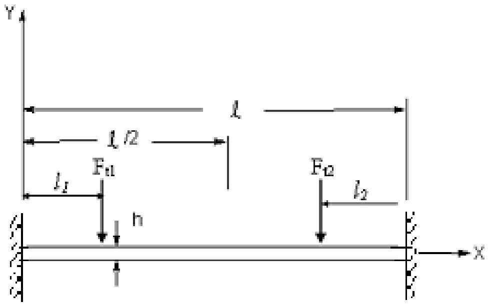


Figure 5. Beam model of front axle subjected to transient loads.

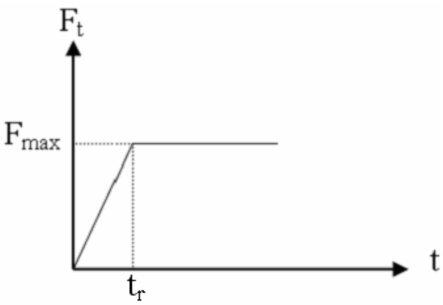


Figure 6. Load versus time curve in dynamic loading.

$$V_i = \sqrt{2gh} = 3.13 \, {}^{m}\!/_{s}$$

where, V_i is the combine speed when reaching the ground in metres per second and h is the fall height in metres.

Assuming that the combine tires are compressed by 0.1 m, the rise time can be calculated using following equation:

$$t_r = \frac{h'}{\bar{V}_i} = \frac{2h'}{V_i} = 0.06s$$

where t_r is the rise time in second; h' is the tire compression in metre and $\bar{V}$ is the average speed during rise time in metre per second. Above equation includes the assumption that the final speed of the combine during rise time is zero.

To calculate the maximum value of dynamic load, the following equation can be used:

$$F_{max} = m_f \times \frac{\Delta V_i}{\Delta t} = m_f \times \frac{3.13}{0.06} = 52m_f = 5.3m_f g$$

where F_{max} is the maximum value of dynamic load, m_f is the applied static mass on the front axle and g is the acceleration due to gravity. This equation means that dynamic load is 5.3 fold as the static load.

In dynamic loading conditions, the vertical acceleration of lumped mass of the vehicle body due to the road surface roughness can be six times as much as the acceleration of gravity. This means that the maximum dynamic loads can be increased six times as much as the corresponding loads in static loading conditions. The amount of dynamic acceleration for tractors and agricultural machinery is considered 6 g [14]. This issue approximately verifies the mentioned assumptions and calculations for obtaining the rise time and maximum value of dynamic loads applied on the front axle. In this study to increase the design factor of safety, the maximum value of dynamic load is considered six times as much as the applied static load. Subsequently, the defined boundary conditions applied to the model were analyzed using post processing. The results of analysis were used to identify value of maximum induced stress.

Experimental validation of the FEM analysis: The results obtained by finite element method were verified experimentally. The stress measurement was carried out using strain gauges. In this study four characteristic points A, B, C and D on the top box of front axle of JD 955 combine were selected for the stress measurement by strain gauges (Fig. 7). The measured strains were compared with the values of calculated strains at same points on the front axle of JD 955 combine by finite element method for verification purpose.

Results and Discussion

The equivalent Von Mises stress distribution of the front axle of JD 955 combine under static loading is provided from the finite element analysis (Fig. 8). The maximum Von Mises stress appears on the upper box and near to the right and left connection plates (Fig. 8). The calculated maximum von Mises stress is 170 MPa; 72.3% of the yielding point of material. The value of factor of safety was 1.38 and found to be less than required value. Calculated value of factor of safety is very low and obviously this value decreases under cyclic loading conditions of field operation. This means that the front axle of JD 955 combine doesn't satisfy the safety conditions for maximum loading even if it is exerted statically and there is a need to optimize the existing design of the front axle of JD 955 combine in order to install on modified combine.

The results of the analysis of the meshed model of optimized front axle under static loading, with elements of average size of 10 mm are given as Von Mises stress distribution (Fig. 9). The maximum Von Mises stress was 55 MPa. Consequently the value of factor of safety was 4.27. The present study clearly indicated that the optimized front axle was strong enough to be installed on the modified combine.

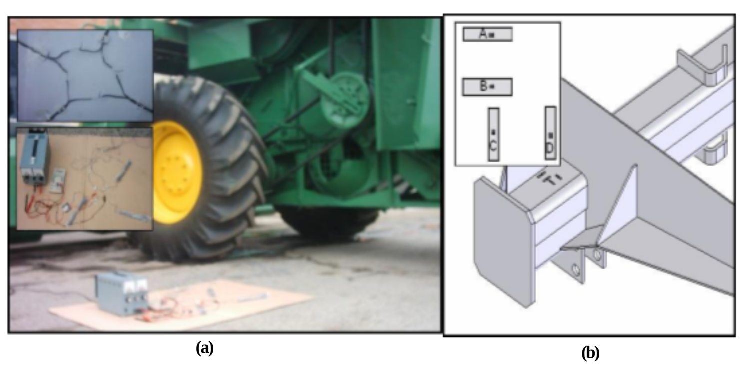


Figure 7. Experimental setup of strain measurement. (a) Wheatstone bridge circuit and strain measurement apparatuses (electric power supply, voltmeter, active and dummy strain gauges, resistance and connecting wires). (b) Arrangement of strain gauges on the front axle of JD 955 combine.

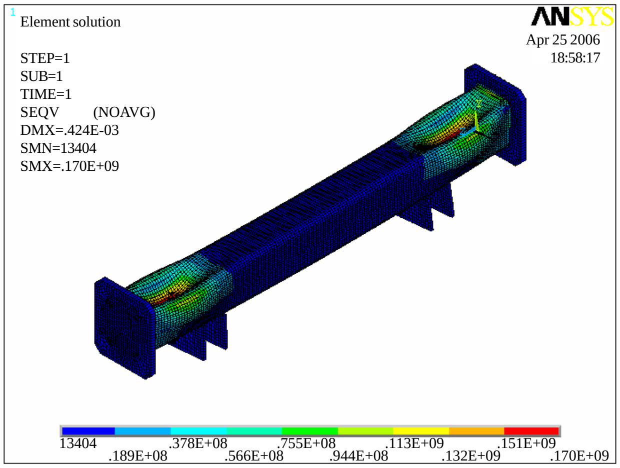


Figure 8. Von Mises stress distribution on front axle of JD 955 combine under static loading.

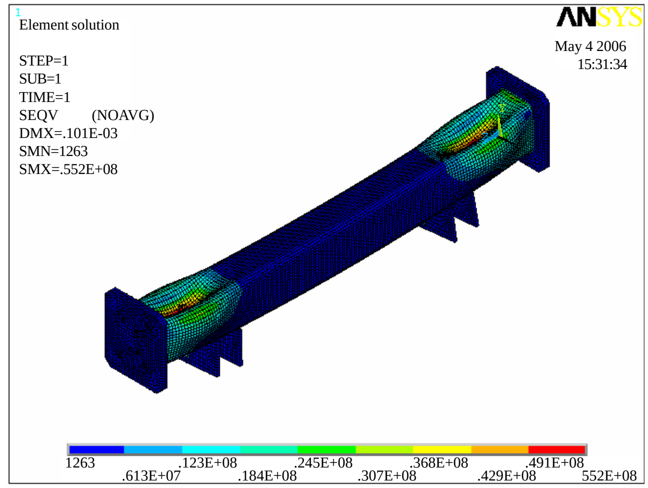
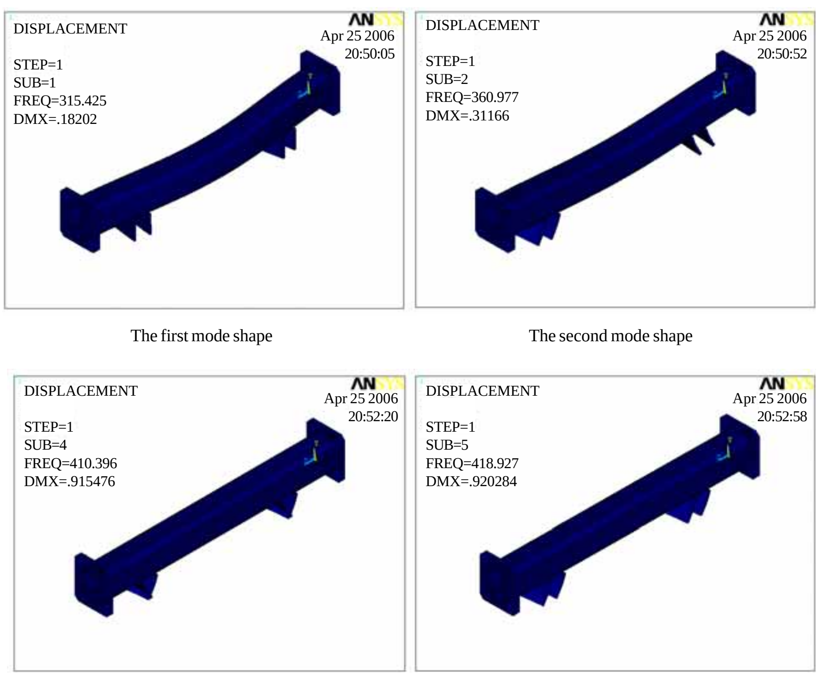


Figure 9. Von Mises stress distribution on front axle.

Considering the combine speed variations and normal wavelength of harmonic motion, the excitation frequency range was calculated to be from 0 to 20 Hz. The first four natural frequencies of the optimized front axle were calculated as:

$f_1 = 315$ Hz, $f_2 = 361$ Hz, $f_3 = 410$ Hz and $f_4 = 419$Hz.



The first mode shape

The second mode shape

The third mode shape

The fourth mode shape

Figure 10. The first four mode shapes of front axle.

Table 1. Measured and calculated strains.

Point	Measured strain	Calculated strain	Percent error
A	0.382×10^{-3}	0.340×10^{-3}	11.0
B	0.497×10^{-3}	0.452×10^{-3}	9.0
C	0.764×10^{-4}	0.829×10^{-4}	8.5
D	1.146×10^{-4}	0.960×10^{-4}	16.2

According to the natural frequency values and excitation frequency range, it was concluded that the natural frequency values were much higher than those of the excitation frequency. Therefore, the condition of resonance was not encountered. The first four mode shapes of the front axle are given in Fig. 10.

The results of the transient analysis showed that the maximum bending stress in the optimized front axle was 206 MPa. The value of maximum bending stress due to transient loading conditions was lower than the yield stress strength. Because the transient dynamic loads were applied on the axle suddenly, a permanent deformation could not take place. Therefore, it is concluded that the optimized front axle of combine has enough strength under transient loading conditions.

The amount of measured strain, calculated strain and percent error at points where strain was measured are presented in Table 1. The quantity percent error shows the difference between the measured and calculated strains. The percent error varies from 16.2% to 8.5%, with an average value of 11.18%. The low error percent clearly indicates that the values of strain calculated by finite element method were in good agreement with experimental data.

The front axle of JD 955 combine is not strong enough to be installed on the modified combine and there is a need to optimize the existing design of the front axle of JD 955 combine in order to install on modified combine. Based on the results of FEM analyses of front axle of JD 955 combine, the optimization of the current model of axle was carried out.The optimized front axle of combine has enough strength under static, harmonic and transient loading conditions.

Acknowledgements

We acknowledge the financial support from Research Department of University of Tehran of I. R. Iran under research award number 2225. We are also grateful to all officials and engineers of ICMCO, especially Department of Research and Development (R&D Center), for good collaborations.

References

[1]Beckert, B. A. 1996. Venturing into virtual product development. Computer-Aided Engineering **15**(5):45-50.

[2]Kojima, Y. 2002. Mechanical CAE in automotive design. R&D Review of Toyota CRDL **35**:1-10.

[3]Lee, K. 1999. Principles of CAD/CAM/CAE Systems. Addison-Wesley, Inc.

[4]Ayazi, M. 2004. Design and Manufacturing Process of J.D.1165 Series 300. Master of Science thesis, Tehran University.

[5]Leon, N., Martinez, P. O. and Adaya, P. 2000. Reducing the weight of a frontal axle beam using experimental test procedures to fine tune FEA. 2nd Worldwide MSC Automotive Conference, Dearborn, Michigan.

[6]Mahanty, K. D., Manohar, V., Khomane, B. S. and Nayak, S. 2001. Analysis and Weight Reduction of a Tractor's Front Axle. Tata Consultancy Services, India, Swarup Udgata, International Auto Limited, India.

[7]Maly, J. and Bazzaz, E. 2003. Design change from casting to welding for an axle casing. http://www.aveng.com/Paper_ MSC_03.pdf.

[8]Nanaware, G. K. and Pable, M. J. 2003. Failures of rear axle shafts of 575 DI tractors. Engineering Failure Analysis **10**:719-724.

[9]Topac, M. M., Gunal, H. and Kuralay, N. S. 2008. Fatigue failure prediction of a rear axle housing prototype by using finite element analysis. Engineering Failure Analysis **16**:1474-1482.

[10]ANSYS IP Inc. 2005. ANSYS User's Manual. Ver.9.0. ANSYS IP Inc.

[11]Shigley, J. E. and Mischke, C. R. 1989. Mechanical Engineering Design. 5th edn. McGraw-Hill Book Company.

[12]Thomson, W. T. and Dahleh, M. D. 1998. Theory of Vibration with Applications. 5th edn. Prentice-Hall, New Jersey.

[13]Srivastava, A. K., Goering, C. E. and Rohrbach, R. P. (eds) 1993. Engineering Principles of Agricultural Machines. The American Society of Agricultural Engineers, St. Joseph, Mich.

[14]Mirehea, A. 2005. Dynamic and Vibrational Analysis of Engine Vibration Damper of the Badsan Truck and Design a Vibration Balancer System. MSc. thesis, Tehran University.