Investigation of Structural Stress Monitoring System on Excavator under Impact Loads

Liang Peng, Zhenlei Chen*, Yongkang Shi, Ganggang Li and Daiwei Zheng

Faculty of maritime and transportation, Ningbo University, Qi Xing Nan Road 169, Ningbo, China
Email: chenzhenlei@nbu.edu.cn

Abstract. Displacement and pressure sensors were used to measure the displacement and pressure variations of the hydraulic cylinders in an excavator under impact loads. Taking the measured cylinder displacement as a driven boundary condition, the kinematics simulation is completed in the LMS Virtual Lab. Taking the measured cylinder thrust and the simulation results of the kinematics as driven boundary conditions, the dynamic simulation of the excavator is carried out. The CAE dynamic stress analysis of the boom and arm of the excavator is performed using ABAQUS inertia release approach under the loads on the sub-structure obtained from the above multi-body dynamic simulation and the measured data, it provides a reference for the material selection of each part of excavator boom. The good agreement between the analytical and testing results demonstrates that the present method is accurate, efficient, cost effective, and able to monitor the stress status and access the health of an excavator instead of using field testing techniques.

Keywords. Excavator, experiment test, multi-body dynamics simulation, inertia release, dynamic stress simulation.

1. Introduction

Excavator working equipment consists of boom, arm, bucket, boom hydraulic cylinder, arm cylinder, bucket hydraulic cylinder, connecting rod mechanism and so on. All connections between the members are hinged. All kinds of complicated excavation actions are realized through the expansion of each hydraulic cylinder [1]. With the development of the mechanical engineering, the working conditions of the excavator are becoming more and more complex, resulting in that the force of the working device is becoming more and more severe. In view of the different working conditions of excavators, their stress conditions are also different, and the requirements for the design and material selection of various structural parts are not the same. It is necessary to put forward some different welding materials for different parts of the excavator: the steel and welding materials of the welded structure should not only meet the necessary strength and stiffness requirements, but also meet the requirements of low-temperature impact toughness. The stress monitoring of the excavator can better determine the material requirements of excavator under working conditions, so as to meet the needs of field use.

How to properly evaluate and optimize the working device during the design phase is one of the being solved problems in the engineering community. The combination of experimental tests and computer simulations is an effective method to solve the above-mentioned problems.

In literature Lanakie [2] used a predefined time-varying cutting force function as the external load of the excavator, and calculated the load spectrum of each hinge pin point and the stress distribution of the working device, to provide evidence for reducing the dynamic load during the excavation process. Qi
and Song [3] used pressure and angular strain synchronous measurement methods to test the excavating process of the excavator, and obtained the static stress, the tooth tip movement trajectory, the stress spectrum of each measuring point and the excavation resistance load spectrum of each measuring point of the working device combined with the kinetic theory. The results provide a basis for the study of mining theory and structural optimization design. Based on the test data and driven by the maximum theoretical impact force obtained from the force simulation of each hinge point Zhang et al. [4] performed the transient dynamic analysis and fatigue life analysis of excavators using Adams software, in which the static moment balance principle was applied to the dynamic impact process. Ren et al. [5] adopted the dynamic stress and transient analysis method to study the dynamic stress characteristics of the hydraulic excavator working device during normal excavation, in which the dynamic balance equation was established based on the D’Alembert dynamic and static principle. Leel and Chang [6] and Muvengei and Kihiu [7] utilized the bond diagram method to establish the dynamic model of the excavator, in which the model was obtained by simply requesting the speed and angular velocity at the centroid of the rod. Frimpong, et al. [8] developed the vibration analysis model of the excavator, in which the problems were solved by transforming the matrix hybrid graph model into a block diagram structure.

The above literature survey shows that how to apply the test data to improve the accuracy of excavator dynamic stress simulation is one of the main research directions. Especially, the analysis method of the stress and strength of structures under impact loading is worthy of further studying. In this paper, the dynamic simulation of the excavator is carried out. The CAE dynamic stress analysis of the boom and arm of the excavator is performed using ABAQUS inertial release approach under the loads on the sub-structure obtained from the multi-body dynamic simulation and the measured data. The calculation results show that the acceleration information of the component obtained is in good agreement with the measured results, and the stresses and strains of the corresponding key part are also consistent with the test results. The present simulation provides a relatively accurate numerical basis for the strength assessment of large excavator work devices during the design phase.

2. Multi-body Dynamic Model of Excavators

Figure 1 is a multi-body dynamic model of a 24.5 ton excavator based on LMS Virtual Lab software. The model involves more than 30 components, such as bucket, dipper, boom, rotary platform, car, track and so on. In addition, according to the relative motion relations among the components, the motion pairs, constraints and the initial oil cylinder length between the components are also set up in the model. The weights of the excavator platform and the cab accessories are located on the platform in the form of mass points. The elastic contact model, as shown in figure 1, is used between the track and the ground [9].

![Figure 1](image_url)
According to the mechanical system model, the Lagrange equation of the system is established, which together with corresponding constraint equations for each rigid body are listed in six generalized coordinates as follows,

\[
\frac{d}{dt} \left( \frac{\partial K}{\partial \dot{q}_i} \right) - \sum_{i=1}^{n} \frac{\partial V_i}{\partial q_i} \lambda_i = F_j, \quad (1)
\]

\[
\psi_i = 0 \quad (i=1,2,\ldots,n), \quad (2)
\]

where \( K \) is the kinetic energy; \( q_i \) is the generalized coordinates of system description; \( \psi_i \) is the constraint equation of the system; \( F_j \) is the generalized forces in the direction of generalized coordinates, and \( \lambda_i \) is the Lagrange multiplier array of \( n \times 1 \).

Equations (1) and (2) can be written as follows:

\[
\begin{bmatrix} F \\ \psi \end{bmatrix} = 0, \quad (3)
\]

where 0 is the zero matrix, \( F=(\dot{q},\dot{q},q,t) \) and \( \psi=f(q,\dot{q},t) \)

Define kinetic energy as:

\[
K = \frac{1}{2} r^T \dot{r} + \frac{1}{2} u^T I u, \quad (4)
\]

where \( T \) is the time, \( r \) is the locational coordinate, \( u \) is the angular coordinate, and \( I \) is the moment of inertia. Equation (3) can be merged into a concise matrix, that is,

\[
M \ddot{x} + \psi^T \lambda = Q^*, \quad (5)
\]

where \( \ddot{x} = [\ddot{x}_1,\ddot{x}_2,\ldots,\ddot{x}_n] \), \( \psi = [\psi_{x1},\psi_{x2},\ldots,\psi_{xn}] \), \( M^{**} \) are the 6 \( \times \) 6 generalized quality diagonal matrices and 6 \( \times \) 1 generalized arrays, respectively, that is,

\[
M = diag \left[ M_1, M_2, \ldots, M_n \right],
\]

\[
Q^* = [Q_1^{**}, Q_2^{**}, \ldots, Q_n^{**}].
\]

3. Experimental Test

In order to establish a virtual simulation monitoring system for the structural strength assessment of the excavator, the test data should be obtained during the loading process of a certain impact condition, in which the displacement of the cylinder and the pressure of the cylinder are applied to the analysis of multi-body dynamic. The test data of the stresses and strains and the acceleration at the key parts are used to verify the virtual simulation results.

Figure 2 shows the flow chart of the strength monitoring system of excavators, which is illustrated by a positive load impact case shown in figure 3. At the beginning of the working condition, the bucket teeth are about 300 mm from the surface of the steel plate. When the engine rotates at full speed, the cylinder is driven to quickly fall boom. The bucket teeth are impacted on the ground to support the body, and the body and the ground forms a 15-degrees angle at this time. After holding up the state for three seconds, the body is lowered by adjusting the pressure of the cylinder for three times, and at each time it suddenly stops and keeps 1 second to continue to decline until it last drops in the contact surface.
Displacement and pressure sensors were used to record the displacements of each hydraulic cylinder (figure 4) and the pressure changes, respectively. Straight and floral strain gauges are used to measure the strain values in various different directions. The resistance specifications of the strain gauges employed are 120 $\Omega$ or 350 $\Omega$ [10].
4. Multi-body Dynamic Analysis of the Excavator

4.1. Kinematic Analysis

Take the displacements of the oil cylinder measured in the time domain as the displacement boundary of each cylinder and use the Virtual.Lab kinematic analysis module to simulate the kinematics of the excavator under the impact condition as shown in figure 1. The motion trajectory of each component in the three-dimensional space can be obtained. Figure 5 shows the horizontal and vertical displacements and rotation angle of bucket teeth. Figure 6 shows a change map of the centroid acceleration on the excavator boom.

Figure 4. Measured displacement curves of each cylinder.

Figure 5. Spatial position change curve (displacement(a), (b) and angle(c)) of bucket teeth.
4.2. Dynamic Analysis

Figure 7 plots the oil cylinder pressure curve obtained from the experimental tests. In the dynamic simulation analysis, the driving force is applied to the three cylinder positions of the excavator dynamic model, as shown in figure 8. The displacement and angle curves obtained from the above kinematic simulation, which are shown in figure 5, are applied to the bucket teeth. The forces acting on the three hinge joints of A, B and G shown in figure 8 are calculated through the Virtual.Lab dynamic analysis module. Figure 9 shows a force diagram of the hinge point B of the excavator bucket rod from the boom.

Figure 6. The acceleration (X-direction (a), Y-direction (b)) of the boom.

Figure 7. Cylinder thrust change curve of A(a), B(b), G(c) three position.

Figure 8. Force diagram of each component.
5. Calculation of Dynamic Stresses

5.1. Calculation and Analysis Method

Sometimes, there may have difficulty in the use of the time-domain multi-body elastic dynamics model, such as time-consuming, difficult to converge and low efficiency. The finite element transient analysis method of inertia force applied to each component of the excavator needs to calculate the acceleration of each element artificially or programmatically with low calculation efficiency. The inertia release method in ABAQUS can solve the above problems perfectly.

The ABAQUS inertial release method is used to perform the finite element analysis of a structural system with moving components of varying speed and varying loads. The system may be unconstrained or partially constrained. In the release of inertia:

\[ \{U\} = \{U_b\} + \{u\}, \]

where \(\{U\}\) is the total response of the object, \(\{U_b\}\) is the response of rigid body motion at a rigid body of reference point, \(\{u\}\) is the relative response. After the expressions of velocity and acceleration are defined the dynamic equilibrium equation can be approximated as follows by using the finite element method,

\[ [M]\{\ddot{u}\} + [M]\{\ddot{u}_b\} + \{I\} = \{P\}, \]

where \([M]\) is the mass matrix, \(\{I\}\) is the internal force vector, and \(\{P\}\) is the external force vector.

\[ \{\ddot{u}_b\} = \sum_{j=1}^{6} (T)_j \ddot{Z}_j, \]

where \(\ddot{Z}_j\) is the acceleration of reference point, \((T)_j\) is the acceleration vector. The response of interest in the inertial analysis refers to the rigid body response corresponding to the static response and to the dynamic movement of reference points and rigid body motion [11].

5.2. Dynamic Stress Calculation of Excavator Structure under Impact Condition

After completing the kinematic analysis and dynamic analysis of the whole excavator, the ABAQUS inertial release is used to analyze the dynamic stress of each substructure of the excavator under the local
maximum load condition. The stresses and strains obtained by the analysis are compared with the previously obtained test results. The acceleration term is also compared with the acceleration obtained from the test as mentioned above. These comparisons are used to evaluate the rationality and accuracy of the calculation process.

The corresponding finite element model is set up with the boom (or bucket bar) of the excavator, and the stress analysis of the structure is selected as the object to be investigated, in which the load of the structure is provided by the force of the instantaneous hinge point and the tension of the cylinder obtained from the multi-body dynamics simulation. Figure 10 shows the finite element model of the boom (t=119.8 s), and all the forces applied at two hinge joints A and B, the arm cylinder and the bucket rod oil cylinder. The model is analyzed using the ABAQUS inertial release technique.

Figure 10. The finite element model of the boom and its applied forces (t=119.8 s).

5.3. Acceleration Contrast Analysis
Table 1 gives a comparative analysis of the centroid acceleration at time t=119.8 s obtained from the inertial release finite element calculation under the impact condition and the multi-body dynamic calculation results driven by the test data. It is evident that the simulated acceleration is in good agreement with the test result.

Table 1. Comparison and analysis of simulation and test of boom centroid acceleration under impact.

|                | \(a_x\) [m/s\(^2\)] | \(a_y\) [m/s\(^2\)] |
|----------------|----------------------|----------------------|
| Multi-body dynamic | 8.477                | 5.937                |
| Finite element calculation | 8.477                | 5.936                |

5.4. Stress and Strain Analysis of Impact Condition (Boom)
Figure 11 shows the location of the strain test points of the boom, in which the red represents the weld point and the blue is the base material. Table 2 gives the comparison of the strain values at time t=119.8 s under an impact condition between the numerical simulation and experimental test. The results are also shown in figure 12 using the histogram plot. It can be found from table 2 that the accuracy of the testing points 66, 69 and 126 is relatively low. The reason for this is because these three points are located near the shaft hole, which are influenced by the assumption of the axial hole consolidation in the multi body model, for which a substructure contact analysis may be needed in order to improve their accuracy.
Figure 11. Points distribution map of boom.

Table 2. Comparison of the test value and the simulation value of the boom strain.

| Measuring points | Experimental data | Numerical results | Deviation |
|------------------|-------------------|-------------------|-----------|
| 5-1              | 250.2             | 234.7             | 93.80%    |
| 66               | -398.6            | -128.2            | <50%      |
| 69               | 157.1             | 98.6              | 62.80%    |
| 70               | 234.1             | 218.3             | 93.30%    |
| 74               | -278.7            | -243.1            | 87.20%    |
| 77               | 267.4             | 255.6             | 95.60%    |
| 81               | -248.7            | -295.1            | 81.30%    |
| 82               | 12.7              | 10.6              | 83.50%    |
| 85               | 314.1             | 323.7             | 97.00%    |
| 91               | 297.8             | 346.1             | 83.80%    |
| 109              | -277.6            | -265.8            | 95.70%    |
| 110              | -231.2            | -275.8            | 80.70%    |
| 119              | -206.3            | -224.3            | 91.30%    |
| 120              | -337.2            | -360.3            | 93.10%    |
| 126              | -386.3            | -252.7            | 65.40%    |
| 129              | -137.3            | -118.5            | 86.30%    |
| 130              | -104.3            | -107.8            | 96.60%    |
| 143              | -747.5            | -595.9            | 80%       |

Figure 12. Comparison of simulation value and test value of boom strain under impact condition.

5.5. Stress and Strain Analysis of Impact Condition (Bucket Rod)

Figure 13 shows the location of the strain test points in the bucket bar. The comparison of the strain values at time t= 119.8 s under an impact condition between the numerical simulation and experimental
test is given in table 3. The results are also shown in figure 14 using the histogram plot. It can be seen from the figure 12 that the largest strain value obtained in the test is 686.8, while the largest strain value obtained from the simulation is 667, which are very close with the accuracy of 97%. In contrast, the accuracy of the weld spot point 55 is only about 73.6%, lower than the target value 80%, indicating that the simulation model need be improved or the element mesh may need to be refined.

![Figure 13. Points distribution map of bucket rod](image)

Table 3. Comparison of the test value and the simulation value of the bucket rod strain.

| Measuring points | Experimental data | Numerical results | Deviation |
|------------------|-------------------|-------------------|-----------|
| 1                | 104.4             | 91.2              | 87.40%    |
| 3                | 240.9             | 222.6             | 92.40%    |
| 5                | 275.6             | 250.3             | 90.80%    |
| 16               | 104.8             | 86.9              | 82.90%    |
| 17               | 76.5              | 90.6              | 81.60%    |
| 20               | 280.5             | 254.2             | 90.60%    |
| 22               | -88.1             | -68.6             | 77.90%    |
| 23               | 77.3              | 61.6              | 79.70%    |
| 24               | 218.9             | 183.7             | 83.90%    |
| 25               | 260.2             | 209.7             | 80.60%    |
| 29               | 24.6              | 36.5              | 51.30%    |
| 30               | 31.35             | 36.55             | 83.40%    |
| 33               | 686.8             | 667               | 97.10%    |
| 50               | -234.4            | -203.3            | 86.80%    |
| 54               | 314.3             | 301.6             | 96.00%    |
| 55               | 417.3             | 307               | 73.60%    |
| 56               | 329               | 277.6             | 84.40%    |
| 58               | -72.2             | -83.5             | 84.30%    |

![Figure 14. Comparison of simulation value and test value of bucket rod strain under impact condition.](image)
6. Conclusions
In this paper, a Virtual.Lab multi-body dynamic model has been developed for the dynamic analysis of excavators, which is driven by using the test results. The dynamic stresses and strains in the key components of the excavator are analyzed using Abaqus. The comparison of the strain results between the numerical simulation and test demonstrates that the numerical model is able to reproduce the test results and thus the field test of excavator structural system can be gradually reduced or replaced by the computer simulation, which can reduce not only the cost but also improve the efficiency. From the present study the following conclusions can be drawn:

1) By taking the measured displacement of the oil cylinder as the driven factor, the kinematic simulation of the working device of the excavator can be performed by using LMS Virtual.Lab, and the motion trajectory of the bucket teeth, including the displacement, the angle and the acceleration, can be obtained.

2) The dynamic simulation of the excavator can be performed based on the measured cylinder thrust. The accurate interaction forces between individual structural components in time domain can be obtained.

3) The dynamic stresses of the key structural components in the excavator can be analyzed using ABAQUS inertial release method. The present simulation results are found to be in good agreement with the test results.

4) The present study shows that the dynamic stress analysis method under the impact load of the excavator working device is accurate and feasible, and it can be used as a healthy monitoring system as well as a design guidance to access the dynamic characteristics of the excavator.

Acknowledgment
The work presented in the paper was supported by Ningbo University via the Project (2020SRIP1715, 2020SRIP1707) and new-shoot Talents Porgram of Zhejiang province (2020R405077).

References
[1] Bai R 2011 Stress Simulation Analysis and Test Verification for Excavator Boom Zhejiang University, China.
[2] Lanakiev A 2008 Modelling of complex structures using superelements International Journal of Design Engineering 1 396-411.
[3] Chen J, Zhang S Q, Yang Y, Li W B, et al. 2007 Research on the testing method of hydraulic excavator’s digging performance International Conference on Advances in Construction Machinery and Vehicle Engineering Shanghai China 26 August p 266-270.
[4] Zhang W G, Quan L and Cheng H 2011 Transient dynamic analysis on working device of excavator based on practical load Journal of Mechanical Engineering 47 144-149.
[5] Ren Z G, Chen J, Wang S C, et al. 2014 Dynamic stress test and transient analysis of hydraulic excavator Journal of South China University of Technology (Natural Science Edition) 42 22-28.
[6] Lee S J and Chang P H 2012 Modeling of a hydraulic excavator based on bond graph method and its parameter estimation Journal of Mechanical Science & Technology 26 195-204.
[7] Muvengei M and Kihiu J 2012 Bond Graph Modeling of Mechanical Dynamics of an Excavator for Hydraulic System Analysis and Design Journal of Mechanical Science and Technology 1 195-204.
[8] Frimpong S, Hu Y F and Inyang H 2008 Dynamic modeling of hydraulic shovel excavators for geomaterials International Journal of Geomechanics 8 20-29.
[9] Wan X F and Liu L 2010 Introduction and Improvement of LMS Virtual Lab Motion 1st ed. (Xi’an: Northwestern Polytechnical University press) pp 33.
[10] Chen J, Wu J, Li W B, et al. 1997 Finite element analysis and stress excavators Chinese Journal of Construction Machinery 97 661-668.
[11] Hibbitt M, Karlsson J and Pawtucket R I 2006 ABAQUS User’s Manual.