Dynamic Strength Analysis for Hydraulic Excavators

Masashi KAWABATA, Dr. Etsujiro IMANISHI, Mechanical Engineering Research Laboratory, Technical Development Group, KOBE STEEL, LTD.

This report presents a nonlinear dynamic analysis of hydraulic excavators using a component mode synthesis method with retransformed physical coordinates. After the vehicle structure is modeled with the finite element method, the FE model is transformed to a super-element using a component mode synthesis method. This method, when applied to the rough road traveling test and the cabin bench test, enables the accurate evaluation of dynamic stress on the structure being studied. This simulation technique, when performed in an advanced design stage, allows the speedy development of new products.

Introduction

For machinery, such as hydraulic excavators and construction machinery, it is becoming important to pre-evaluate performance at the stage of trial manufacturing to create new products, reflecting market needs. In order to make the pre-evaluations possible at design stages, simulation technologies are indispensable to estimate various issues related to dynamics, which may occur during actual operation.

In the development of new construction machinery, failures in quality, which may occur in actual operation sites, are prevented through pre-evaluations of durability of body frames and various other components, by executing trial tests including rough road traveling tests and bench tests of components, such as forced vibration test. In those tests for pre-evaluations, strength evaluations must take dynamics into consideration.

Kobe Steel developed a multipurpose tool, a non-linear dynamic analysis code SINDYS[1] based on the Finite Element Method (FEM) and has been applying the tool for the pre-evaluations. This article introduces, as case examples for dynamic strength analyses, a rough road traveling analysis of a hydraulic excavator and a dynamic strength analysis of a cabin.

1. Dynamic strength analysis method

The non-linear dynamic analysis code SINDYS is a tool enabling dynamic simulations of mechanical systems, simultaneously considering structural, hydraulic and control units. Generally, large scale finite element models are prepared for strength analyses of complex structures, such as construction machinery. Such models are not practical because the methods of directly executing time integrations on such models, for dynamic analysis require excessive amounts of calculation time. It is more efficient to calculate on models in which linear portions are reduced to lower dimensions. SINDYS has a newly developed conversion program, named COMET, between itself and a multipurpose FEM code, NASTRAN. COMET allows retrieval of modal data in FE models and has static reduced matrix as parts of elements.

The following explains a modal synthesis method[2] of retranslating into physical coordinates, as a way of reducing large scale FE models into lower dimensions.

The dynamic equation, digitized by the FEM, is given by the following formula;

$$\boldsymbol{M}\ddot{\boldsymbol{u}}+\boldsymbol{C}\dot{\boldsymbol{u}}+\boldsymbol{K}\boldsymbol{u}=\boldsymbol{f} \quad \cdots\cdots(1)$$

where, $\boldsymbol{M}$: mass matrix
$\boldsymbol{C}$: damping matrix
$\boldsymbol{K}$: stiffness matrix
$\boldsymbol{f}$: external force vector
$\boldsymbol{u}$: displacement vector.

Equation (1) solves eigenvalue problems negating damping, to determine eigen frequencies and natural modes. Here, the modal matrix, in which natural mode vectors from the first to p^{th} orders are arrayed, is denoted as Φ. Assuming the displacement vector to be a linear combination of the natural modes and the modal coordinates to be $\boldsymbol{\xi}$, the displacement vector is given by the following formula:

$$\boldsymbol{u}=\Phi\boldsymbol{\xi} \quad \cdots\cdots(2)$$

By substituting formula (2) into formula (1), multiplying Φ^T from the left and assuming the damping matrix to be proportionaldamping the following formula is established based on the orthogonality of natural modes;

$$\begin{bmatrix}\ddots & & \\ & m & \\ & & \ddots\end{bmatrix}\ddot{\boldsymbol{\xi}}+\begin{bmatrix}\ddots & & \\ & c & \\ & & \ddots\end{bmatrix}\dot{\boldsymbol{\xi}}+\begin{bmatrix}\ddots & & \\ & k & \\ & & \ddots\end{bmatrix}\boldsymbol{\xi}=\boldsymbol{\Phi}^T\boldsymbol{f} \quad \cdots\cdots(3)$$

where:

$$\begin{bmatrix}\ddots & & \\ & m & \\ & & \ddots\end{bmatrix}=\boldsymbol{\Phi}^T\mathbf{M}\boldsymbol{\Phi},\ \begin{bmatrix}\ddots & & \\ & c & \\ & & \ddots\end{bmatrix}=\boldsymbol{\Phi}^T\mathbf{C}\boldsymbol{\Phi},\ \begin{bmatrix}\ddots & & \\ & k & \\ & & \ddots\end{bmatrix}=\boldsymbol{\Phi}^T\mathbf{K}\boldsymbol{\Phi} \quad \cdots(4)$$

In conventional component mode synthesis, the synthesis takes place after response from formula (3), given external force values and initial conditions. On the other hand, in the modal synthesis method of retranslating into physical coordinates, formula (3) is retranslated in coordinates, including the joints with non-linear portions and the physical coordinates from which responses are expected.

Assuming the physical coordinates to be ub, which include r ($r<p$) pieces of joints and expected responses, the following formula is established in a mode coordinates $\boldsymbol{\xi}$;

$$\boldsymbol{u}_b = \Phi_r \boldsymbol{\xi} \quad \cdots (5)$$

The following formula is established by leaving the mode coordinates from the $r+1$ to p^{th} orders, as they are, and assuming them to be $\boldsymbol{\xi}_n$;

$$\begin{Bmatrix} \boldsymbol{u}_b \\ \boldsymbol{\xi}_r \end{Bmatrix} = \begin{bmatrix} \boldsymbol{\Phi}_r \\ \mathbf{0} \quad \mathbf{I} \end{bmatrix} \boldsymbol{\xi} \quad \cdots (6)$$

Here, I is a unit matrix. If the matrix on the right side is regular, then the following equation is derived from formula (6);

$$\boldsymbol{\xi} = \mathbf{R} \begin{Bmatrix} \boldsymbol{u}_b \\ \boldsymbol{\xi}_r \end{Bmatrix} \quad \cdots (7)$$

where;

$$\mathbf{R} = \begin{bmatrix} \boldsymbol{\Phi}_r \\ \mathbf{0} \quad \mathbf{I} \end{bmatrix}^{-1} \quad \cdots (8)$$

By substituting formula (7) into formula (3) and multiplying R from the left, the following formula is obtained.

$$\mathbf{R}^T \begin{bmatrix} \ddots & & \\ & m & \\ & & \ddots \end{bmatrix} \mathbf{R} \begin{Bmatrix} \ddot{\boldsymbol{u}}_b \\ \ddot{\boldsymbol{\xi}}_r \end{Bmatrix} + \mathbf{R}^T \begin{bmatrix} \ddots & & \\ & c & \\ & & \ddots \end{bmatrix} \mathbf{R} \begin{Bmatrix} \dot{\boldsymbol{u}}_b \\ \dot{\boldsymbol{\xi}}_r \end{Bmatrix} + \mathbf{R}^T \begin{bmatrix} \ddots & & \\ & k & \\ & & \ddots \end{bmatrix} \mathbf{R} \begin{Bmatrix} \boldsymbol{u}_b \\ \boldsymbol{\xi}_r \end{Bmatrix} = \mathbf{R}^T \boldsymbol{\Phi}^T \boldsymbol{f} \quad \cdots (9)$$

The above equation is the conversion of the equation expressed in the modal coordinates, leaving the physical coordinates, has the same degree of freedom and is expressed by the physical coordinates $\boldsymbol{u}_b$ and remaining higher order mode coordinates $\boldsymbol{\xi}_r$. The responses from ub and $\boldsymbol{\xi}_r$ are determined from the above equation, the modal coordinates $\boldsymbol{\xi}$ are determined from formula (7) and, after that, the response from the original physical coordinate is determined from formula (2). On the other hand, the element force $\boldsymbol{f}_e$ is given by the following formula using an element force matrix Φ_f, in which the first to $\boldsymbol{p}^{th}$ order element force vectors are arrayed.

$$\boldsymbol{f}_e = \Phi_f \boldsymbol{\xi} \quad \cdots (10)$$

As shown in **Figure 1**, to integrate the multipurpose code NASTRAN and SINDYS, Kobe Steel developed a program for modal synthesis, named COMET, and a program for regenerating FE modal stress based on the dynamic analysis results from SINDYS, named COMSIN, all of which are configured into one dynamic strength analysis system.

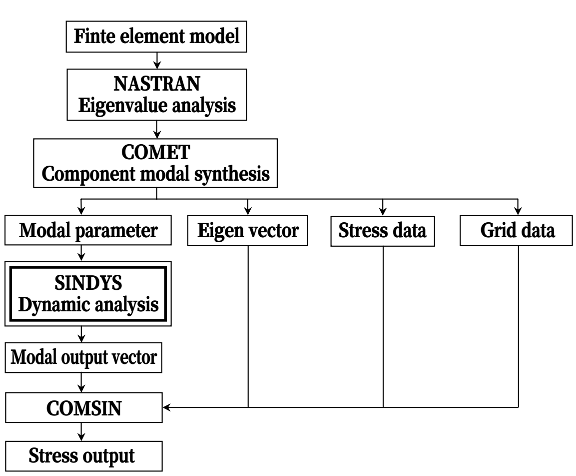

Fig. 1 System flow of dynamic strength analysis

2. Analysis case examples

2.1 Rough road traveling analysis

The following describes a case example, in which the dynamic strength analysis method is applied to the strength analysis of the upper-structure main-members of a hydraulic excavator, traveling on a rough road. In the rough road traveling test, as shown in **Photo 1**, a hydraulic excavator is driven on a road with placed blocks to evaluate durability of the frame and other components of the vehicle. As shown in **Figure 2**, FE models were prepared for the main structures, consisting of cabin, upper swing body and lower structure, and eigenvalue analysis was executed using NASTRAN. This model has a degree of freedom of 10,000 or more and uses small to several tens of pieces of modal data. A simplified model was used, in which masses of heavy loads, such as attachments and tanks, are placed towards the center of gravity, so

Photo 1 Rough road traveling test

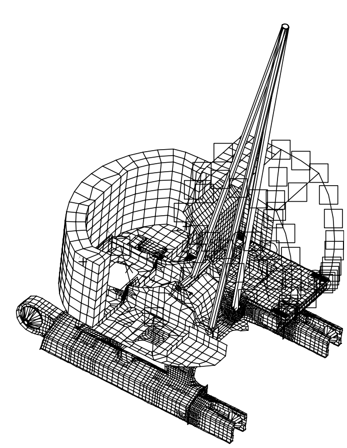

Fig. 2 Finite element model of hydraulic excavator

that the loads are imposed as a concentrated mass. The contacts between the road surface and the lower structure are represented by the idlers and sprockets on the left and right, under an assumption that the crawler portion has a high enough rigidity, In this model the physical coordinates to be left, in the lower structure, are degrees of freedom of the idlers and sprockets on the both sides (rotary portions in the front and back) and degrees of freedom of points to be evaluated for responses. Based on the data, the left term of formula (7) gives a dynamic equation, which is scaled down to the degree of freedom by the number of the modes.

An outline of the analysis model according to SINDYS is shown in **Figure 3**. The behavior during riding over the rough road blocks is determined separately by a structural analysis software and the responses, obtained for the idlers and sprockets, are given to SINDYS as excitation inputs, to execute a behavior analysis. The contacts with the road surface are modeled using a piecewise linear spring, as shown in Figure 3. The responses from the physical coordinate, along with the responses from the modal coordinate, are obtained as a result of the behavior analysis using SINDYS, which are retranslated into the time response of stress.

As an example, a stress response of a main frame element is shown in **Figure 4**. Note that the vertical and horizontal axes are normalized. The stresses obtained by this method are compared with the criteria for the actual rough road tests for the judgment of acceptance, enabling pre-evaluation of strength in design stages.

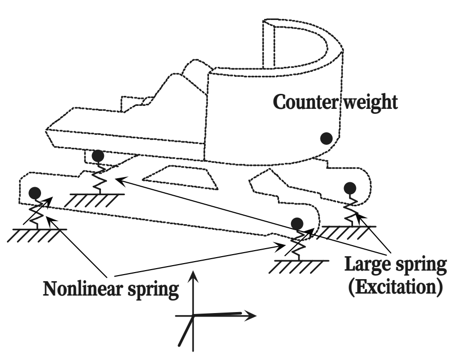

Fig. 3 Simulation model in SINDYS

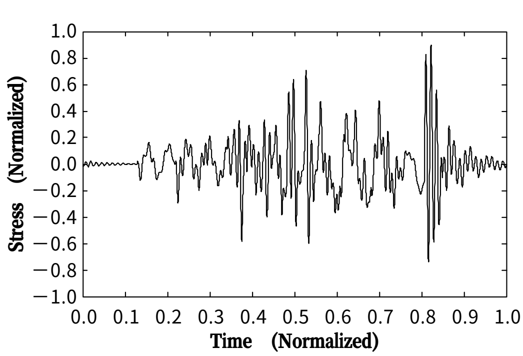

Fig. 4 Stress response of rough road simulation

2.2 Dynamic strength analysis of cabins

Durability of cabins is evaluated by durability tests using vibration exciters as shown in **Photo 2**. In general, methods for vibration excitation include sinusoidal wave excitation at a constant frequency, shock wave excitation and excitations by actually measured waveforms.

The following explains an example in which a random waveform, with a specific power spectrum density (PSD), is applied for the excitation. The frequency of the random waveform and intensity of PSD are set individually for each model, based on the accelerations and strains actually measured on the machines.

The cabins of hydraulic excavators are supported by a highly-viscous fluid mount, as shown in **Figure 5**. In its characteristics there exists a frequency dependence and amplitude dependence[3)], which cannot be expressed by any simple model, in which linear springs and dampers are connected in parallel. In order to take such non-linearity into account, a FE model having a large degree of freedom, as shown in **Figure 6**, was reduced to a lower dimension, by the modal synthesis method described above, and modeled as elements for SINDYS.

A random response analysis was used for the analysis method of SINDYS. The random analysis is a method for frequency response analyses, in which

Photo 2 Oscillation test of cabin

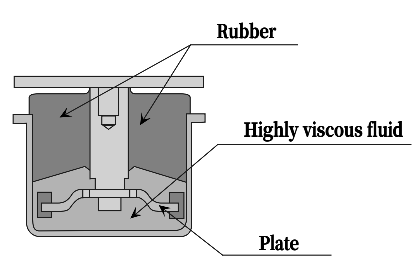

Fig. 5 Configuration of highly viscous fluid mount

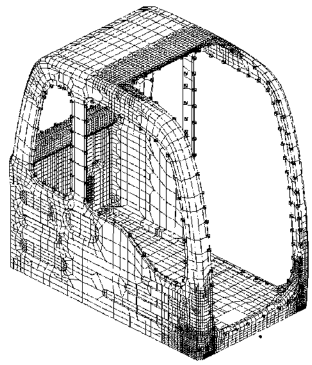

Fig. 6 Finite element model of cabin

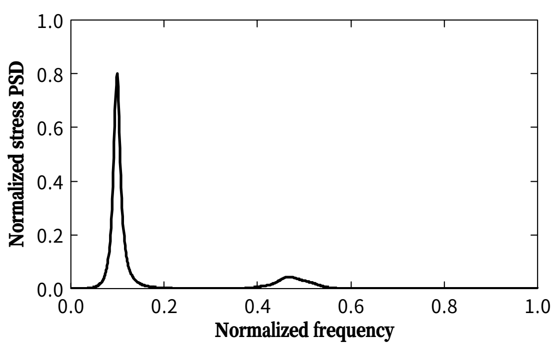

Fig. 7 Example of SINDYS stress PSD output

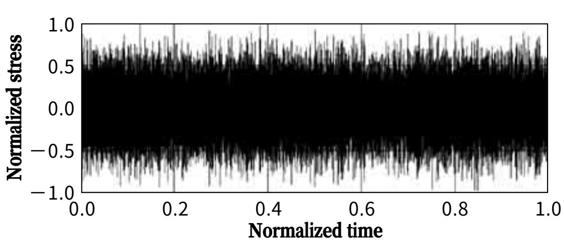

Fig. 8 Stress time history using inverse FFT

the target is set to the random excitation inputs of the PSD.

The resulting stress is obtained in the amplitude of the PSDs corresponding to the frequencies, an example of which is shown in **Figure 7**. In order to evaluate fatigue resistance, however, time response of stresses is required. The reverse FFT analysis of the phase, given randomly to the PSD waveform obtained on the frequency axis, generates time response as shown in **Figure 8**. Damage is calculated by the rain flow method, based on the stress response, to evaluate

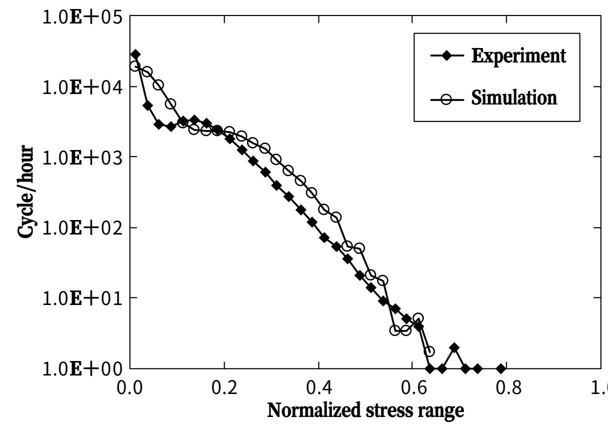

Fig. 9 Frequency distribution for stress range

fatigue life based on the S-N curve of the evaluation portion and modified Miner's rule.

Figure 9 shows a frequency distribution per unit time for the stress range counted by the rain-flow method based on a simulation result and a frequency distribution obtained from experimental stress measurements. Both the results agree well, indicating that lives can be estimated with accuracies high enough for practical applications. In actual pre-evaluations, safety factors are taken into account on the simulated results to further ensure the fitness-for-purpose of the machines to be built.

Conclusions

A method for strength analysis, taking dynamics into account, has been described, and example cases are introduced for the rough road traveling analysis of a hydraulic excavator and dynamic strength analysis of a cabin. Estimations of dynamics, prior to trial manufacturing, and pre-evaluations of strengths ensure perfections of the machines to be built and speeds-up their development.

References

1) T. Fujikawa, et al., *R&D Kobe Steel Engineering Reports*, Vol.37, No.1, p.93 (1987).
2) E. Imanishi, et al., *JSME*, 52-476, p.1232 (1986).
3) Y. Hamazaki, et al. *JSME*, 52-484, p.3207 (1986).