Vibration Rapid Analysis and Comparison of Motor Support in Container Crane's Machine Room with Single Degree of Freedom and Finite Element Method Based on Visual Basic

Lu Kai-liang[1], Ye Hao[1], Qiu Hui-qing[2], Zhang Wei-guo[1],
Hao Zhi-yong[1] and Liu Yuan[1]

*[*1]Logistics Engineering College, Shanghai Maritime University*
[2]College of Mechanical Engineering, Tongji University

lkl1984@163.com

Abstract

The excessive vibration of motor support in container crane's machine room affects lifespan of the structure and driving comfort of the operators. According to the result of the vibration site test, the main causes of the machine room vibration in container crane were found out. In order to reduce the vibration response, based on simplifying the system as a single degree of freedom vibration system model and finite element method, two kinds of rapid vibration analysis software were compiled respectively by using Visual Basic (VB), which can parameterize the motor and its support. Meanwhile, their advantages and disadvantages were compared. By analyzing the effects of different motor support structure layouts on its natural frequency and maximum vibration velocity response, it is concluded that increasing and improving the motor support's cross-sectional moment of inertia and lateral stiffness can reduce the system vibration response. The rapid analysis software's computing results are almost the same as ANSYS computing results. Furthermore, motor support structure of the container crane machine room was optimized through the original software. The accordance of vibration test data and analysis results prove the credibility and the reliability of the rapid analysis software and methods.

Keywords: *Container Crane, Motor Support, Vibration, Rapid Analysis, Visual Basic*

1. Introduction

Vibration and noise problems are coming up as the container crane is developing towards a high speed, heavy load direction. In recent years, researches about the container crane vibration problem mainly focused on the dynamic characteristics analysis of the structure or the front girder [1] , the analysis and control of wind-induced vibration of the oblique rod [2, 3], and seismic-isolated device for the machine to resist earthquake [4], *etc*. While related studies to the vibration problems of container crane's machine room, which are closely related with the crane structure safety and staff working comfort, are still not common.

At present, there are many methods or commercial software used in vibration analysis, such as ANSYS based on finite element method (FEM) and ADAMS based on virtual prototype technology, *etc.* However, for some not very complex vibration problems, using commercial software to do analysis is both time-consuming and uneconomical. Therefore, for certain specialized problems, usually self-preparation software or programmes are preferred to do rapid analyses, *e.g.*, Shijing WU, *et al.*, [5] used Matlab to compile a torsional vibration analysis system for vehicle driveline based on the expanded transfer matrix method. Deshan

SHAN, *et al.*, [6] used numerical algorithm based on excitation nonlinear vibration compiling vehicle-bridge coupled vibration analysis software on railway curvilinear continuous beam.

The design of motor, gear box and reel's support or base structures is important to the machine room's vibration, and these supports or bases are similar in structures. Thus, quickly optimizing the design can be achieved through the development of simple and rapid analysis software. The purpose of this paper is to use object-oriented programming language Visual Basic (VB) to compile rapid vibration analysis software for container crane's machine room, which is used to avoid resonance in the design stage. Research ideas are as follows: Firstly, according to the vibration test results, analysing the causes of vibration in the machine room; Secondly, compiling VB rapid vibration analysis software respectively based on single degree of freedom method (SDOF Method) and finite element method (FEM) [7-8]; Finally, using the rapid vibration analysis software to study different motor support structure arrangements' effects on the system vibration, and comparing the analyses results with ANSYS results and measured data, meanwhile, the advantages and disadvantages of SDOF Method and FEM are compared.

2. Vibration Experiment Analysis for Container Crane Machine Room

2.1 Vibration test

According to the theory of simple harmonic vibration, two quantities out of displacement, velocity and acceleration can be obtained by calculation as if vibration frequency and other quantity are known. Vibration velocity can adequately reflect the equipment vibration energy, momentum and the resulting impact effect. In this paper, the motor vibration in a container crane machine was tested on the spot, the structure and size of the motor pedestal are shown in Figure 1. The main size parameters are as follows: Motor pedestal height H=688mm, length L=1540mm, longitudinal plate distance D=360mm, longitudinal plate thickness t_1=12mm; the inner, outside diaphragm plate thickness t_2=t_3=10mm, the inner, outside diaphragm row number N_1=N_2=3, inner and outer side of diaphragm spacing d_1=D_1=20mm, d_2=D_2=290mm, d_3=D_3=1006mm, diaphragm plate elongation l_1=l_2=85mm.

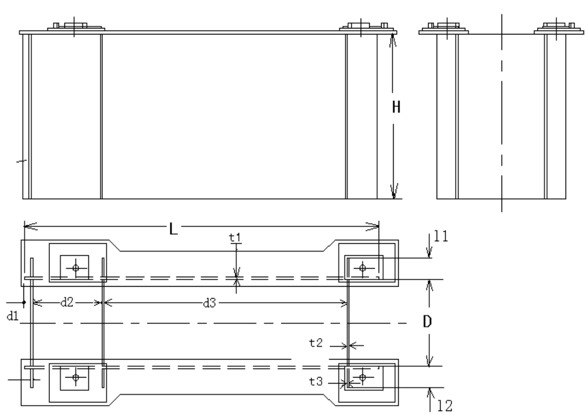


Figure 1. Structure and Dimension of Motor Support

Test position was in the middle of the motor height direction. Vibration velocity tests were carried out at points A, B, C in directions of *x*, *y*, *z* (as shown in Figure 2). The test results are shown in Table 1.

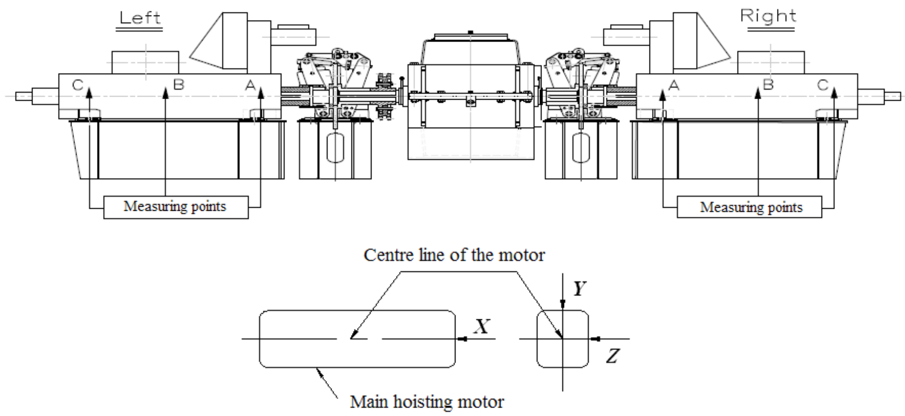


Figure 2. Measuring Points of the Vibration Test

Table 1. Vibration Velocity Test Results

Direction	Maximum value of vibration velocity (mm/s)									
	100% speed*		90% speed		75% speed		60% speed		45% speed	
	Left	Right	Left	Right	Left	Right	Left	Right	Left	Right
x	0.5	0.7	0.1	0.2	0.2	0.3	0.5	0.2	0.3	0.2
y	0.7	1.5	0.5	0.9	0.3	0.5	0.1	0.3	0.2	0.2
z	5.4	7.2	2.1	3.8	1.4	0.9	0.4	0.5	0.2	0.3

* Rated speed of left and right motors is 1968 r/min

2.2 Causes analysis

From Figure 1, it can be seen that: 1) The maximum vibration velocity value is 7.2 mm/s, this value is greater than the allowable vibration velocity value of 4.5 mm/s required by the users; 2) Motor vibration response in the lateral direction of *z* is significantly higher than that in the other two directions. The motor is installed on the support, so we can take the motor and the support as a system to study the reasons resulting in vibration.

By using rotor dynamics [9,10] analyses, the following reasons cause the motor rotor vibration: 1) Because of errors in the motor manufacture procedure, the imbalance of rotor mass causes forced vibration, the higher rotation speed is, the larger excitation force is; 2) Motor body is too long, the distance of bearings is so far, and the diameter of rotor shaft is small, thus the critical speed is low; 3) The length of motor support is long and height is high, so the lateral stiffness of motor support is not enough.

For container crane makers, rational design of motor support is the key to reduce machine room's vibration. The easiest way to do this is to improve its lateral stiffness through the reasonable layout of the motor support structure. Basing on two kinds of calculation methods of simplifying the system as a SDOF forced vibration model or FEM, we used Visual Basic (VB) to compile vibration rapid analysis software which can parameterize and model the motor pedestal, calculating the natural frequency and the maximum vibration velocity response by different motor excitations. Designers only need to input the parameters of the motor and support to software, the vibration magnitude and motor support design schemes can be quickly judged.

3. Rapid Analysis with SDOF Method

3.1 Natural frequency calculation

The motor support is fixed on the ground which can be seen as a cantilever beam. Motor is equivalent to concentrated mass at the free end. Ignoring the structural damping of the steel plate pedestal, the system can be simplified as a non-damping SDOF forced vibration system, as shown in Figure 3. The excitation is inertial centrifugal force which is caused by imbalanced rotation of the rotor.

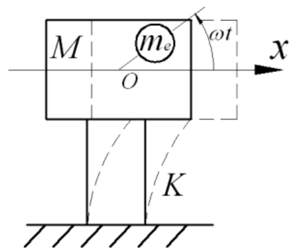


Figure 3. SDOF Forced Vibration System Model

Assuming that eccentric mass of motor rotor is m_e, eccentricity is e, angular velocity of motor rotor is ω, mass of motor rotor is m_{m}, mass of pedestal is m_{s}, then the system equation of motion is

$$M\ddot{x} + Kx = m_e e\omega^2 \sin\omega t \tag{1}$$

Where, equivalent mass is

$$M = 33/140 m_{\mathrm{s}} + m_{\mathrm{m}} \tag{2}$$

And equivalent stiffness is

$$K = 3EI_z / H^3 \tag{3}$$

In Equation (3),

E—Young’s elastic modulus of materials, Pa;

I_z —The cross-sectional moment of inertia of pedestal, m^4;

H —Height of pedestal, m.

The natural frequency of the system thus can be obtained as follows.

$$\omega_{\mathrm{n}} = \sqrt{K/M} = \sqrt{\frac{3EI_z}{H^3(33/140 m_{\mathrm{s}} + m_{\mathrm{m}})}} \tag{4}$$

3.2 Maximum vibration velocity response

By solving system equation of motion (1) can obtain the maximum vibration velocity response as follows.

$$v_{\max} = \frac{m_e e \lambda^2 \omega}{\left|1-\lambda^2\right| M} \tag{5}$$

Where, frequency ratio is $\lambda = \omega / \omega_n$.

According to Equation (4) and (5), the natural frequency and vibration response of the system can be calculated out quickly when the support design is modified.

4. Rapid Analysis with FEM

It is neither economic nor fast to use the commercial FEM software to solve the motor support vibration problem. According to the FEM theory and algorithm [11], the original software is compiled by using VB, dividing the structure into proper elements to guarantee the calculation efficiency.

4.1 Element meshing

The motor support structure is simple, and mainly composed of longitudinal plate and a lateral plate. We used eight nodes hexahedron element type to mesh the structure, as shown in Figure 4.

The number of elements and nodes are in accordance with the order of lateral plate first, inside transverse diaphragm later to rank. After numbering the elements and nodes in above rules, works such as assigning node coordinates, element node extraction and so on can be continued.

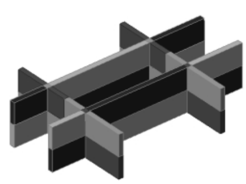


Figure 4. Schematic Map of Element Meshing

4.2 System equation of motion

Because the meshing is a little bit sparse, we used consistent mass matrix to form the element mass matrix $\boldsymbol{M}_e$, the element stiffness matrix $\boldsymbol{K}_e$ and the unit load matrix $\boldsymbol{Q}_e$.

Ignoring the damping, the system equation of motion is:

$$\boldsymbol{M}\ddot{\boldsymbol{x}}(t) + \boldsymbol{K}\boldsymbol{x}(t) = \boldsymbol{Q}(t) \tag{6}$$

In the equation: $\boldsymbol{M}$—Mass matrix of system; $\boldsymbol{K}$—Stiffness matrix of system; $\boldsymbol{Q}(t)$ —Nodal load vector of system; $\ddot{x}(t)$ —Acceleration matrix of system; $\boldsymbol{x}(t)$ —Displacement matrix of system.

On the basis of nodal stiffness ratio, the motor mass and excitation force was assigned to the nodes of the two longitudinal plates on the motor support surface to get the matrices of $\boldsymbol{M}$, $\boldsymbol{K}$, $\boldsymbol{Q}(t)$.

4.3 Natural frequency and dynamic response

The system characteristic equation is:

$$\left|\boldsymbol{K}\text{-}\boldsymbol{\omega}_{\mathbf{n}}{}^{2}\boldsymbol{M}\right| = \mathbf{0} \tag{7}$$

Where, $\boldsymbol{\omega}_{\mathbf{n}}$ —Natural frequency matrix of system.

Because of the limited finite element degree of freedoms, the first three orders natural frequencies were solved by the subspace iteration method. Using mode superposition method to solve dynamic response, afterwards, the nodal displacement response is still based on the motor speed frequency ω , and the amplitude can be obtained by matrix calculation. Velocity is equal to amplitude multiplied by ω , and then we can get the maximum vibration velocity $v_{\max}$ by simple comparing method.

5. Comparison of the Two Rapid Analysis Methods

5.1 Comparison of rapid analysis software

As an object-oriented visual design tool based on event-driven programming mechanism, VB, a structured programming language, provides an easy-to-use application integrated development environment.

Compiling procedures of the motor support vibration rapid analysis software are as follows: 1) Establish objects of user interface; 2) Set up object attributes; 3) Compile object event procedure program; 4) Debug program.

Through using interactive user interface, graphics and formula simultaneously, software operation interface was compiled in accordance with above-mentioned methods, as shown in Figure 5, Figure 6. The meanings of symbols in the figures are as the same as previous.

It is very easy to use the two kinds of software: Follow the prompt to input parameters, and click “Natural Frequency” (or “Calculate Result”) button, the computing results will be shown on screen. After seeing the results, clicking "OK" key will close the result box, and the user can return to previous interface to modify the parameters to recalculate new results.

Comparing Figure 5 with Figure 6 shows the differences in the interface: “Rapid analysis software based on FEM” has no information about centroid position, but extra “Lateral diaphragm” interface, through which we input diaphragm elongation and spacing parameter information, compared with “Simple and rapid analysis software based on SDOF method”.

Because in simplified SDOF method, cross-sectional moment of inertia of pedestal is needed only to calculate natural frequency, the position of diaphragm is irrelevant to the results; while in FEM, the diaphragm position is a necessary parameter in the process of solving the system equation.

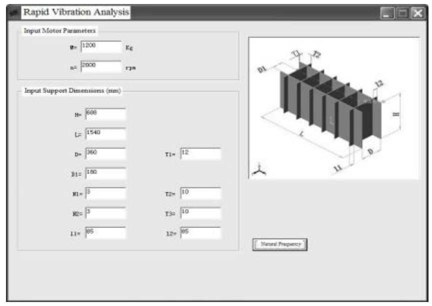
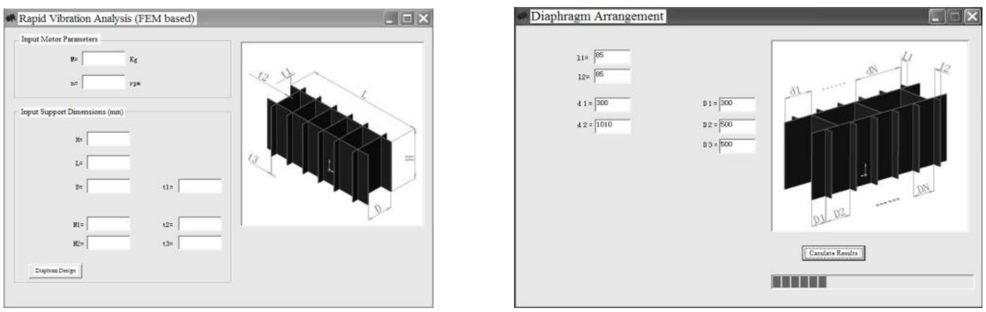


Figure 5. Interface of SDOF Method based Rapid Analysis Software



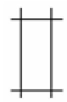
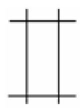
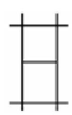
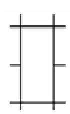
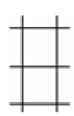
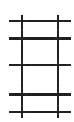
(a) Rapid Vibration Analysis (FEM based)　　　(b) Diaphragm Arrangement

Figure 6. Interface of FEM based Rapid Analysis Software

5.2 Comparison of calculation results

Using two kinds of rapid analysis software above, the motor support vibration response was calculated respectively, and compared with that calculated by general FEM analysis software ANSYS. During calculating, the following parameters are invariant: Motor mass M=1200kg, motor speed n=1968rpm, pedestal height H=688mm, pedestal length L=1540mm, longitudinal plate distance D =360mm, longitudinal plate thickness t_1=12mm, the inner, outer diaphragm thickness t_2= t_3=10mm. The influence on natural frequency f_n and the maximum vibration velocity response v_{max} are studied by changing row number, spacing between inner and outer diaphragm as well as the outer diaphragm elongation. The results are shown in Table 2.

Table 2. Comparison of Natural Frequency and Maximal Vibration Velocity of Motor Support Calculated by using Rapid Analysis Software and ANSYS

NO.	Structure sketch	Dimension (mm)	Calculation results (Note: errors compared with ANSYS calculated results in percentage are in the brackets)					
			SDOF Method		FEM		ANSYS	
			f_n (Hz)	v_{max} (mm/s)	f_n (Hz)	v_{max} (mm/s)	f_n (Hz)	v_{max} (mm/s)
a		Elongation: $l_1= l_2$=85 Spacing: d_1=D_1=257 d_2=D_2=1006	102 (29%)	11.2 (24%)	83 (5.1%)	14.2 (3.4%)	79	14.7
b		Elongation: $l_1= l_2$=100 Spacing: d_1=D_1=257 d_2=D_2=1006	111 (13%)	9.3 (3.1%)	100 (2.0%)	9.4 (2.1%)	98	9.6
c		Elongation: $l_1= l_2$=85 Spacing: d_1=D_1=257 d_2= 500 d_3= 496 D_2=1006	109 (4.8%)	9.6 (4.3%)	99 (4.8%)	9.5 (3.3%)	104	9.2
d		Elongation: $l_1= l_2$=85 Spacing: d_1=D_1=257 D_2= 500 D_3= 496 d_2=1006	118 (8.3%)	8.0 (5.9%)	103 (5.5%)	8.8 (3.5%)	109	8.5
e		Elongation: $l_1= l_2$=85 Spacing: d_1=D_1=257 d_2= D_2=500 d_3= D_3=500	124 (9.7%)	7.2 (6.5%)	107 (5.3%)	8.0 (3.9%)	113	7.7
f		Elongation: $l_1= l_2$=100 Spacing: d_1=D_1=20 d_2= D_2=290 d_3= D_3=475 d_4= D_4=531 d_5= D_5=155	174 (4.8%)	3.5 (9.4%)	161 (3.0%)	3.3 (3.1%)	166	3.2

As it can be seen from Table 2, any way to increase the cross-sectional moment of inertia can reduce the dynamic response, such as increasing the diaphragm elongation, or the row number of inner and outer lateral diaphragm, however, the effects are unavoidably different. Among them, increasing both internal and external diaphragms has the best effect, followed by increasing internal diaphragm only. At this point, the two kinds of software results have the same changing trend. One of the FEM software's advantages is able to consider diaphragm spacing influence.

Via comparing computing results by rapid analysis software and ANSYS, it can be seen that the results of FEM and ANSYS are in good agreement, natural frequency and the maximum vibration velocity response errors are all in the range of 2% to 6%. The SDOF method error is slightly larger, which is consistent with the ANSYS result as well.

Having been calculated by rapid analysis software and ANSYS, the maximum vibration velocity of structure form *f* (shown in Table 2) is less than 4.5 mm/s, which meets the design requirement. Therefore, the structure form *f* can be used as the optimized motor support. We tested the optimized motor support by using vibration velocity test method which is shown in Section 2.1, the test results are shown in Table 3.

Table 3. Test Results with Respect to Vibration Velocity for Optimized Motor Support

Direction	Maximum value of vibration velocity (mm/s)									
	100% speed *		90% speed		75% speed		60% speed		45% speed	
	Left	Right	Left	Right	Left	Right	Left	Right	Left	Right
X	0.4	0.6	0.3	0.2	0.2	0.3	0.2	0.2	0.2	0.1
Y	0.4	0.8	0.3	0.7	0.2	0.4	0.2	0.2	0.1	0.2
Z	2.8	3.3	1.5	2.1	0.8	1.2	0.4	0.3	0.1	0.2

* Rated speed of left and right motor is 1968 r/min

The measured vibration velocity of motor support modified in structure is significantly reduced than the original (seen in Table 1). The peak value is 3.3 mm/s, which is consistent with the calculation results of rapid analysis software and ANSYS, further validating the reliability of rapid analysis software's results and methods.

6. Summary

According to the results of the machine room vibration test, the main reasons causing the strong vibration in the container crane machine room are analyzed to be the powerful excitation from the machine room's motor, excessive length of motor body, small diameter of rotor axle and low lateral stiffness of motor support. It will be effective to improve the lateral stiffness of the motor support to suppress vibration.

By using Visual Basic, two kinds of motor support rapid vibration analysis software, separately based on simplified SDOF method and FEM, have been compiled for the machine room, which can be adopted in analysing the effects of changes in geometric parameters of motor support on the system vibration, and then optimized the motor support in structure. The computing results by rapid analysis software are consistent with ANSYS results and actual test data, but each of the two kinds of software has its advantages: The former is easier to set parameters, and faster in calculating speed; the latter can better reflect the effects of diaphragm arrangement on the system vibration in precision.

Based on SDOF method, "Simple rapid analysis software" will be recommended in optimization design first, and if the results cannot meet the requirements, then modify the design parameters to meet the requirements. Moreover, diaphragm arrangement can be checked and adjusted until the requirements are met, by using "FEM based rapid analysis software".

Acknowledgements

This work is sponsored by Shanghai Top Academic Discipline Project- Management Science & Engineering, this paper is supported by Doctoral Fund of the Ministry of Education Jointly Funded Project (20123121120002), Quality Standards and Metrology Research Project of the Ministry of Transport (2013-419-899-040) and Shanghai Education Committee Project "Shanghai Young College Teacher Training Subsidy Scheme", also supported in part by Ministry of Transport Research Projects (2012-329-810-180), Shanghai Municipal Education Commission Project (12ZZ148, 12ZZ149, 13YZ080).

References

[1] H. Yang, F. Yang, S. Sun and G. Li, "Study on parametric modeling technology of quayside container crape virtual prototype", Hoisting and Conveying Machinery, vol. 39, no. 6, **(2011)**, pp. 66-69.
[2] P. Hang, Y. Wang and M. Gu, "Experimental Research on Mean Wind Loads of a Quayside Container Crane", Journal of Tongji University: Natural Science, vol. 35, no. 10, **(2007)**, pp. 1384-1389.
[3] J. Tang and S. Zheng, "The Effect of Restriction Improvement on the Resonant Velocity of Wind of the Round-section Bar in Quayside Container Crane", Mechanical Research & Application, vol. 19, no. 2, **(2006)**, pp. 63-65.
[4] K. -l. Lu, Y. -f. Huang, W. Yan and J. -l. He, "Numerical Simulation and Structural Model Shaking Table Test for Container Crane Equipped with Seismic-isolated Device", JCIT: Journal of Convergence Information Technology, vol. 8, no. 4, **(2013)**, pp. 977-985.
[5] S. Wu, B. Qian, H. Lu, *et al.*, "Research and Development of Torsional Vibration Analysis System of Vehicle Driveline", Journal of System Simulation, vol. 18, no. 11, **(2006)**, pp. 3100-3103.
[6] D. Shan and Q. Li, "Analysis of Vehicle-Bridge Coupled Vibration on Railway Curvilinear Continuous Beam", China Railway Science, vol. 25, no. 5, **(2004)**, pp. 61-66.
[7] A. F. EI-Bassiouny and M. Eissa, "Dynamics of a single-degree-of-freedom structure with quadratic, cubic and quartic non-linearities to a harmonic resonance", Applied Mathematics and Computation, vol. 139, no. 1, **(2003)**, pp. 1-21.
[8] W. Han and M. Petyt, "Linear vibration analysis of laminated rectangular plates using the hierarchical finite element method—II. Forced vibration analysis", Computer & Science, vol. 61, no. 4, **(1996)**, pp. 713-724.
[9] Y. Zhong, Y. He, Z. Wang, *et al.*, "Rotor Dynamics", Tsinghua University Press, Beijing, **(1987)**.
[10] F. A. Raffa and V. Vatta, "The Dynamic Stiffness Method for Linear Rotor-bearing System", American Society of Mechanical Engineers: Journal of Vibration and Acoustics, vol. 118, no. 7, **(1996)**, pp. 332-339.
[11] K. H. Huebner and E. A. Thornton, "The Finite Element Method for Engineers", Wiley, New York, **(1982)**.

Authors

LU Kai-liang, received his PhD from Tongji University, Shanghai, China. Currently, he is working in Logistics Engineering College, Shanghai Maritime University (SMU) as a lecturer. His research interests include port machine structure and system dynamics, theory and method of dynamic design and optimization of structure, etc. He has 16 publications to his credit both in international and national Journals. He is a committee member of Shanghai Society of Theoretical & Applied Mechanics, member of China Construction Machinery Society (CCMS), and Logistics Engineering Institution, CMES.