Numerical investigation of dynamic parameters of a reinforced concrete dome

A. Alikina
NPO Saturn, JSC, Russia, e-mail: ann8809@yandex.ru

Abstract

Design of spatial reinforced concrete (RC) structures to dynamic loads is based on structural dynamic parameters. Experimental research on dynamic response of full scale RC structures is very limited. Therefore developing proper numerical techniques, allowing calculation of valuable dynamic parameters is extremely important. The purpose of the present study is to investigate efficiency of existing software and its suitability for modeling spatial RC structures. The object of the research is a dome over a public building. It is a shallow thin walled spherical RC shell. The numerical results are compared with available experimental data. ANSYS software was used for the analysis. The affect of different structural components on natural vibration parameters like mode shapes and frequencies was studied. Comparison of numerical and experimental results enabled to verify the accuracy and adequacy of the structural scheme.

Realization of numeral experiments allows to understand properly the qualitative characteristics and to interpret the impact of various parameters in order to obtain reasonable recommendations for mock-up study or full scale experiments. It is shown that qualitative modeling of the system can be used for further investigation of spatial RC structures response to different dynamic loadings.

Keywords: sphere dome, reinforced concrete structures, natural vibration frequencies and mode shapes, finite-element model.

Introduction

Seismic resistance of existing buildings can be estimated by testing a full-scale structure or a scaled model. Alternatively numerical analysis can be performed. Conducting full-scale experiments is cost consuming therefore numerical methods are more popular. Contemporary progress in numerical methods enables to analyse complicated spatial structures. It allows the engineer to closely approach the actual behavior of thin concrete shells by performing geometrically and physically nonlinear analyses.

For proper analysis of thin shells, knowledge about their design history is necessary. These data can be useful for design and construction of new structures with similar schemes. Krivoshapko et al. [1] have discussed useful data for early period reinforced concrete (RC) shells. Well-known examples on early RC shells were presented and use full information about them was given.

Iskhakov et al. [2] studied the dynamic characteristics of the spherical RC shell in Dushanbe. The dynamic load was applied to the shell using a vibration machine, located at the center of the dome. The study presents results of full-scale structures monitoring under dynamic loadings and further analytical investigation. In the tested structure two natural vibration modes in vertical direction were identified. The obtained experimental data were generalized analytically: natural vibration periods of RC domes were calculated. Comparison of experimental and theoretical results shows that the specified analytical expressions allow predicting of the corresponding dynamic parameters with satisfactory accuracy.
Iskhakov and Ribakov [3] shows that a span of a spatial structure is practically independent of its thickness and is a function of its geometry. Analytical and experimental approaches are presented. Numerical examples on surface geometry and calculation of various typical shells, based on analytical methods are given. Additionally, results of experimental, analytical and numerical evaluation of a real RC dome are described.

An important aspect for the first stage of finite elements techniques is the choice of an element, which will model the investigated structure. For modeling of a spherical shell, shell or solid elements can be used. A numerical model using the Finite Element Method (FEM) for the nonlinear static and dynamic analysis of reinforced concrete (RC) beams, plates and shells is presented by Tamayo et al. [4]. Static analysis of RC shells up to failure load was carried out using 9-node degenerated shell finite elements, while 20-node brick elements were used for dynamic applications. The steel reinforcement was considered to be smeared and represented by membrane finite elements. Various benchmark examples were solved using this numerical model and comparisons with other published data were performed. It was reported that the numerical results obtained numerical modeling are satisfactory for both static and dynamic loading.

Tamayo et al.[5] have proposed a three-dimensional numerical model for nonlinear dynamic analysis of an RC containment shell under seismic load. A 20-noded brick finite element is used to model the concrete part, whereas reinforcing steel bars are modeled using incorporated membrane elements. It is reported that the amplitude is lower in the nonlinear analysis than in the linear case because cracking affects the stiffness of the structure, thus changing its fundamental period. It was concluded that the response to seismic excitation is strongly dependent on the structural dynamic characteristics, as the energy, absorbed by the system, depends both on the forcing and the natural frequencies.

**Research aims and scope**

The main aim of this research is the identification or calibration of the dome finite element (FE) model. It includes selection of FEM parameters for better correspondence between the analytical and experimental results for the investigated structure. The identification allows increasing the accuracy of the calculation results and their proper using for extending the experimental data. Without comparing the measured and the calculated data it is impossible to assert with a sufficient degree of accuracy, that the obtained dynamic characteristics, such as natural vibration frequencies of structures, are real properties of the structure.

Modal and harmonic analyses of the structure have been carried out in the identification process of the model. The main purpose of the modal analysis is determining the natural vibration frequencies and mode shapes. Experimental results of vibration testing are used to calibrate the finite element model and to define whether there were the correct basic for design assumptions (for example, if correct properties of material and boundary conditions were used).

Harmonic analysis is used for finding of the steady reaction of the structure under sinusoidal loading. Calculations are carried out to find the response of the system to loading with several frequencies. The harmonic analysis is intended for search of the maximum value of the steady-state vibration level.

Modal and harmonic analyses are carried out in the linear stage. Geometrical and physical nonlinearities are neglected.
Design and analysis process

The object of this study is the dome of a public building in Dushanbe. The dome is a shallow thin walled spherical RC shell. The general view, structural scheme and dimensions of the tested structure as well as description of the full-scale dome testing are presented by Iskhakov et al [2].

At the first stage analysis of the available experimental data and preliminary calculations were performed. To identify the finite element (FE) model of the dome, numerically obtained values of natural vibration frequencies of the model were compared to those, obtained experimentally. The model with corresponding frequencies was used for further assessment of the dome. To determine the natural frequencies modal analysis in the ANSYS14.5 software was performed. The first 5 frequencies of the entire natural frequencies array were chosen.

At the next step other parameters, affecting the accuracy of analytical results, were chosen. These parameters include the modulus of elasticity and the reinforced concrete density. Additionally, the influence of mode shapes and proper supports selection for the model were analyzed.

The ANSYS14.5 software allows developing a suitable FE model of the dome, allowing performing the modal analysis and calculating the structural response to harmonic excitation.

The FE model takes into account the volume and class of the reinforcing steel in different directions, the self-weight of the structure and of the vibration machine.

The applied FE model was built using the following types of elements: SOLID65, MATRIX27, and MASS21. The dome was modeled using the SOLID65 element. It allows taking into account the concrete reinforcement. The MASS21 element enables to consider the mass of the vibration machine. For a more accurate convergence of the experimental and numerical frequencies the fixing dome is modeled considering the supports flexibility. The MATRIX27 element was used for this reason.

The geometrical model of a spherical dome is shown in figure 1.

![Figure 1. The view of the geometrical model](image-url)
The FE model has 208298 nodes and 141646 elements. It is shown in figure 2.

SOLID65 is used for 3D modeling of solids with or without reinforcing bars (rebars). In concrete applications the solid capability of the element may be used to model the concrete while the rebar capability is available for modeling reinforcement behavior. The element is defined by eight nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions. Up to three different rebar specifications may be defined.
The geometry, node locations, and the coordinate system for this element are shown in figure 3. The element is defined by 8 nodes and has isotropic material properties. It is made of one solid material and up to three rebar materials.

![Figure 3. Geometry of the SOLID65 element](image)

Results and Discussion

Modal analysis in ANSYS software is the solution of free damped or undamped vibrations problem for a discrete system, described by the following equation of motion

\[[M]{u''} + [C]{u'} + [K]{u} = 0\]  \hspace{1cm} (1.1)

This equation is given in the form corresponding to eigenvalue problem. For the case of undamped vibrations the equation takes the following form

\[[K] - \omega^2[M]{u} = 0,\]  \hspace{1cm} (1.2)

where \(\omega^2\) is the squared natural frequency and \(\{u\}\) is the vector of natural vibration modes.

Modal analysis in ANSYS software is implemented in several methods. The best results can be achieved using Lanczos method [6] and subspace. These methods use full matrix of stiffness and mass of the system, they work very accurately and efficiently, almost without requiring user intervention in the analysis process.

Numerical values of natural vibration frequencies and mode shapes are shown in table 1 and figure 4. Experimental values of natural vibration frequencies and mode shapes are shown in table 2 and figure 6c. A scheme of the measuring equipment location is shown in fig. 6a. Only the first vibration mode was identified in the horizontal direction (fig. 6b) and two symmetrical modes – in the vertical direction (fig. 6c).

<table>
<thead>
<tr>
<th>The mode shapes number</th>
<th>Natural vibration frequencies (Hz)</th>
<th>Natural vibration period, sec</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>7.55</td>
<td>0.132</td>
</tr>
<tr>
<td>2</td>
<td>8.54</td>
<td>0.117</td>
</tr>
<tr>
<td>3</td>
<td>9.11</td>
<td>0.110</td>
</tr>
<tr>
<td>4</td>
<td>9.70</td>
<td>0.103</td>
</tr>
<tr>
<td>5</td>
<td>9.87</td>
<td>0.101</td>
</tr>
</tbody>
</table>
Analysis of harmonic effects is used to determine the parameters of the steady motion assuming linear behavior of the system under sinusoidal force excitations. Resolving equation for this type of analysis is

\[ \{F(t)\} = \{F_0(\cos(\omega t + \phi) + i \sin(\omega t + \phi))\} \] (1.3)

The displacements change in a sinusoidal form according to the loading frequency. For a given frequency the user has the ability to find the displacements or in the amplitudes and phase.

Typical frequency amplitude at the top of the dome is presented in figure 5. Experimentally obtained dynamic parameters of the dome are presented in table 2.
The numerical value of the natural vibration period is $T_{\text{num}} = 0.101\ldots0.13$ s (table 1), the measured natural vibration period value is $T_{\text{exp}} = 0.106\ldots0.22$ s (table 2).

The numerical peak displacement value is $0.11\ldots0.16$ mm (fig.5), the measured peak displacement value is $0.1\ldots0.37$ mm (table 2).

Table 2. Experimentally obtained dynamic parameters of the dome for the first and the third natural vibration modes in vertical direction (following [2])

<table>
<thead>
<tr>
<th>The mode shapes number</th>
<th>Natural vibration frequencies (Hz)</th>
<th>Natural vibration period, sec</th>
<th>Damping ratio</th>
<th>Peak displacement, mm</th>
<th>Peak velocity, cm/sec.</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>4.54</td>
<td>0.22</td>
<td>0.10</td>
<td>0.37</td>
<td>2.08</td>
</tr>
<tr>
<td>3</td>
<td>9.43</td>
<td>0.106</td>
<td>0.04</td>
<td>0.10</td>
<td>0.63</td>
</tr>
</tbody>
</table>
Figure 6. Location of measuring equipment on the dome (a); natural mode shapes in horizontal (b) and vertical (c) directions (following [2])

Around the natural frequencies of 7.5 Hz and 9.5 Hz resonances are obtained (see fig. 5). Numerical natural vibration mode shapes on the received resonances coincide with experimentally natural vibration mode shapes (1 and 5 mode shapes – in the numerical; 1 and 3 mode shapes – experimentally).

The first natural vibration frequencies were overestimated in comparison with the experimentally obtained values ($f_{exp}$=4.54 Hz, $f_{num}$=7.54 Hz).

The fifth natural vibration frequencies is very close to the third vibration frequency of the experiment in shape and in frequency ($f_{num}$=9.7 Hz and $f_{exp}$=9.43).

Differences between the numerical and experiment values of the frequencies are due to the difference between the actual work of a dome and that proposed in the FEM. In this work calculation was made without taking into account material nonlinearities (inelastic deformations of concrete and reinforcement), possible cracking, geometric nonlinearity, etc.

**Conclusions**

Modal and harmonic analysis of spherical dome were carried out to identify the FE model of the dome. Symmetric shapes were received for the structure. Based on the results of the calculations it can be concluded that for a more accurate calibration of the model physical nonlinearity, possible formation of cracks, modulus of elasticity degradation and other factors should be taken into account.

Qualitative modeling of the system can be used for further investigation of spatial RC structures response to different dynamic loadings.

Mathematical models can be used for predicting further changes of the technical state in time.
References


