Modal and Harmonic Response Analyses of the Kenyir Dam Intake Section

Azizi Arbain, Ahmad Zhafran Ahmad Mazlan, Mohd Hafiz Zawawi and Mohd Rashid Mohd Radzi

Abstract Modal and harmonic responses are the important analyses that need to be carried out to determine the behavior of a structure by executing data in a real-time. Through these analyses, several parameters of the structure can be identified. These parameters which are natural frequencies, mode shapes and frequency response function (FRF) are critical for monitoring and investigating of the dynamic behavior as well as to avoid unnecessary resonance to the structure. In this paper, the intake section of Kenyir Hydropower Dam in Terengganu, Malaysia is taken into consideration for the modal and harmonic response analyses. Numerical study using ANSYS software is performed for the real scale size of the intake section with a proper boundary condition. Six most significant result are selected from the modal analysis including the FRF for all three axes of the intake section. From the FRFs, the highest natural frequency value occurred at 11.024 Hz with the maximum deflection of 2.3203 m in the y-direction direction. This value must be taken into consideration for any external induced vibration to avoid the failure of the dam structure.

Keywords Modal and harmonic analyses · Natural frequency · Mode shape · Frequency response function · Finite element · Intake section

---

A. Arbain · A. Z. A. Mazlan (✉)
School of Mechanical Engineering, Universiti Sains Malaysia, Engineering Campus, 14300 Nibong Tebal, Penang, Malaysia
e-mail: zhafran@usm.my

M. H. Zawawi
Department of Civil Engineering, College of Engineering, Universiti Tenaga Nasional, 43000 Kajang, Selangor, Malaysia

M. R. M. Radzi
TNB Generation Division, Tenaga Nasional Berhad, 59200 Kuala Lumpur, Malaysia

F. Mohamed Nazri (ed.), *Proceedings of AICCE'19*, Lecture Notes in Civil Engineering 53, https://doi.org/10.1007/978-3-030-32816-0_79

1 Introduction

Intake section is one of the most important part of the dam where the water is drawn from upstream reservoir to downstream river through the pipeline or tunnel. This section is built using varies kind of materials including concrete and steel which function to discharge the water from reservoir directly to the river downstream [7]. Therefore, it is necessary to study the dynamic characteristics of this section including the modal and harmonic response analyses. Mohanty and Rixen [4] also stated that, modal parameters such as boundary conditions and coupling forces should be considered as a part of the system during the analysis. In order to obtain the reliable static and seismic responses of the dam structure, the numerical analysis need to consider parameters such as the variation of foundation mass, foundation stiffness and exact geometry of the dam [2]. One of the software that can be used to perform the simulation is ANSYS, which has a powerful processing module and solving functions. The intake section of the dam can be investigated through the modal analysis using finite element method (FEM) which has the ability to solve complex geometries and various case studies [6]. Modal analysis is performed to find the dynamic parameters of the dam structure and these parameters are important for the further analysis of the dam such as stress, pressure, strain and deflection [1]. This study is supported by Khan and Awari [3] which stated that, from modal analysis, a lot of vibration problem studies can been investigated. Meanwhile, through the harmonic analysis, the frequency response function (FRF) can be determined. FRF is a curve that indicates the overall frequency response of the structure in certain frequency ranges based on the input force and output deflection. From this, the detail natural frequencies and the mode shapes of the structure that corresponded to the peak frequencies of the FRF curve can be determined [5].

2 Methodology

2.1 3D Modelling of the Intake Section

The intake section is one of the important parts of the whole Kenyir Dam structure. A t this structure, water is released from the upstream to downstream and is controlled by four gates; one gate for each pressure tunnel. The intake section is mainly built of concrete and reinforcement steel materials. For this simulation study, a three-dimensional (3D) model of the intake is designed using SolidWorks software. Figure 1 shows the 3D model of the intake section.

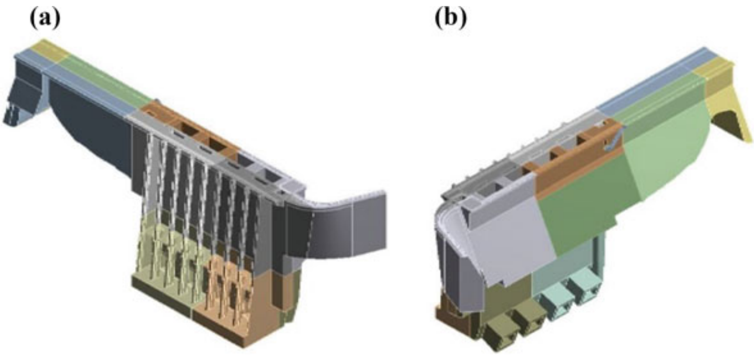
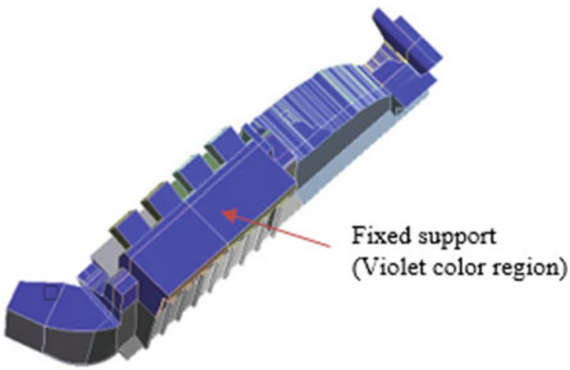


Fig. 1 Upstream (**a**) and downstream (**b**) isometric view of 3D model of intake section

2.2 FE Model and Sensitivity Analysis of the Intake Section

In . The analysis compared three types of meshing which are automatic this study, the finite element (FE) model of the intake section is developed using ANSYS software. Before performing the simulation, a mesh sensitivity analysis is conducted to show how the mesh quality influence the result of the simulation. Through this analysis, an optimum type of meshing can be obtained, hex dominant and tetrahedral and the selection of the optimum meshing is when the most accurate result is obtained with appropriate elements and shorter computation time. The similar geometry shapes have been used to compare the performance of these meshing. A boundary condition has been set according to the real behavior of the intake section of the real dam. The base of the intake section is set as fixed support to demonstrate the foundation of the real structure that attached to the earth topography. Forces are applied to the structure for all three coordinate axes. Figure 2 shows the FE model with the boundary condition for the intake section of the dam.

Fig. 2 FE model with boundary condition for the intake section



2.3 Modal Analysis of the Intake Section

Modal analysis (MA) of the intake section is conducted using ANSYS Workbench (Modal) software to study the dynamic properties of the structure. The CAD model is firstly designed in SolidWorks before it is imported to the ANSYS software. The whole intake section is then set as one part and shared the same topology. The maximum mode shapes to be find is set to 30 with no frequency limit search range and six most significant mode shapes are chosen for the detail discussion.

3 Results and Discussion

3.1 Meshing Sensitivity Results of the Intake Section

The results of three different types of meshing which are automatic, hex dominant and tetrahedral is shown in Table 1. From the table, the lowest 1st natural frequency that has been determined is the meshed body with hex-dominant of 1.3306 Hz followed by tetrahedral and automatic meshed with 1.3444 and 1.3619 Hz, respectively. Figure 3 shows the graph of meshing sensitivity analysis of the intake section. From the figure, it is proved that the hex dominant has the lowest natural frequency value which indicates this type of meshing is the most optimized meshing for the simulation of the structure.

3.2 Modal and Harmonic Reponses of the Intake Section

Figure 4 shows the meshed FE model of the real size intake section that has been developed using ANSYS software. The meshing type is set as hex dominant while element size used is 2.0 m per element. This resulting an amount of 108 083 hex dominant elements for the whole intake section of the dam.

Table 2 shows the six most significant mode shapes of the intake section. It is clearly shown that, the 24th mode shape affected the largest area of the intake section with natural frequency of 11.024 Hz and overall deflection value of 8.8028×10^{-4} m. This mode shape caused the most major deflection at the upper part of the intake section which includes the trash rack and head gate areas. The 1st, 13th. 20th, 26th and 29th mode occurred at 3.2621 Hz, 4.6729 Hz, 8.8442 Hz, 12.47 Hz and 12.525 Hz, respectively where only few masonries that hold the trash rack is deflected.

Table 3 shows the FRF results of the intake section in x, y and z directions. From the graphs shown in Table 3, the highest natural frequency occurred at 11.024 Hz (24th mode shape) for all three directions. Whilst for maximum deflection values, the highest deflection occurred in y-direction with 2.3202 m

Table 1 Comparison of different type of meshing and natural frequencies

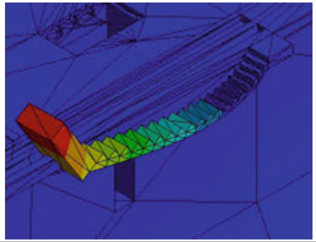
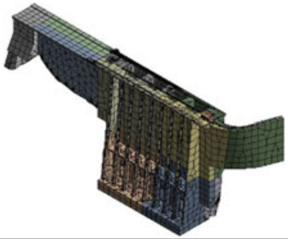
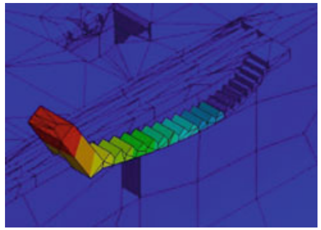
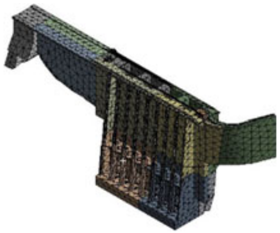
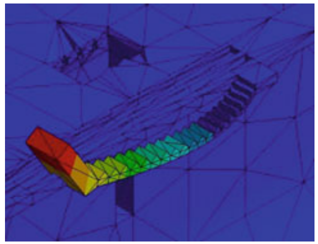
Meshing type	Meshed body	1st natural frequency (Hz)	Mode shapes
Automatic		1.3619	
Hex dominant		1.3306	
Tetrahedral		1.3444	

Fig. 3 Meshing sensitivity analysis of the intake section structure

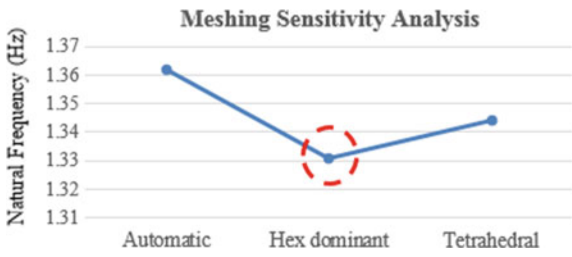
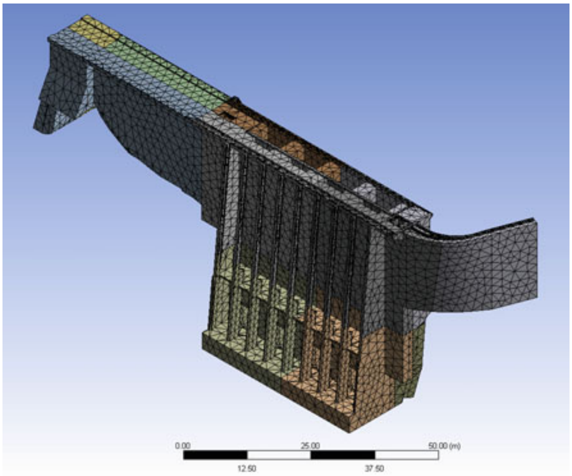


Fig. 4 Meshing FE model of the intake section



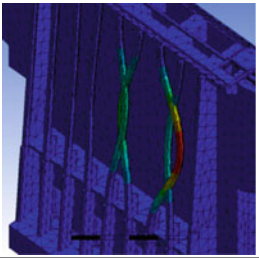
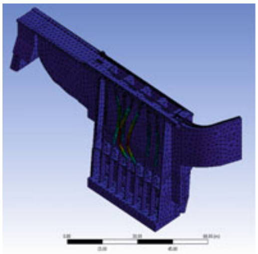
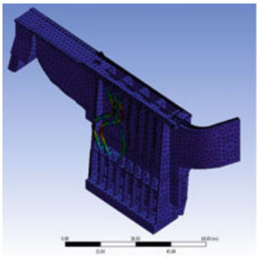
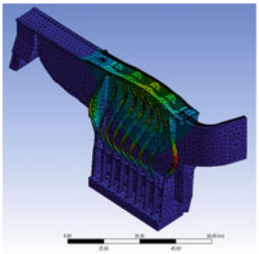
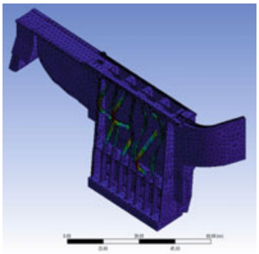
followed by x-direction and z-direction with 0.4323 m and 6.3633 × 10-2 m, respectively. The 2.3202 m deflection is referred to the highly bended mode shapes of the upper part of the intake section as shown previously in Table 2. Any external induced vibration that coincides with this frequency value may destruct the structure of the intake section due to resonance.

4 Conclusion

The modal and harmonic response analyses of the intake section of the Kenyir Dam structure have been successfully carried out. The results are summarized as follows:

- The real scale three-dimensional (3D) model of the intake section of the Kenyir Dam has been developed in detail accordingly to the drawing of the dam.
- The optimum type of meshing has been found through meshing sensitivity analysis which is the hex dominant.

Table 2 Six most significant mode shapes and natural frequencies of the intake section structure

Modes no.	Mode shapes	Natural frequency (Hz)	Deflection values (m)
1		3.2621	6.2796×10^{-3}
13		4.6729	5.2541×10^{-3}
20		8.8442	5.1748×10^{-3}
24		11.024	8.8028×10^{-4}
26		12.47	4.4552×10^{-3}

(continued)

Table 2 (continued)

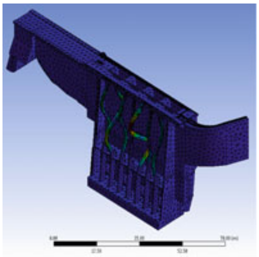
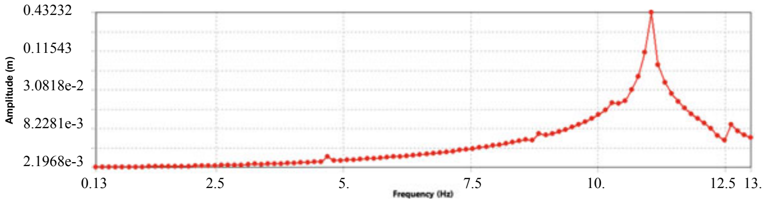
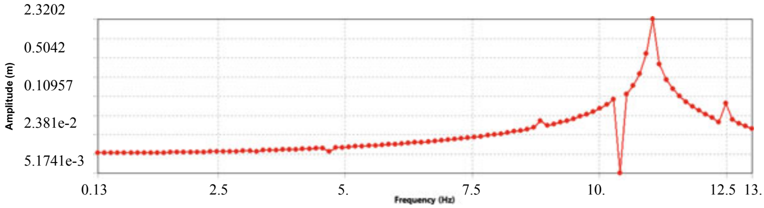
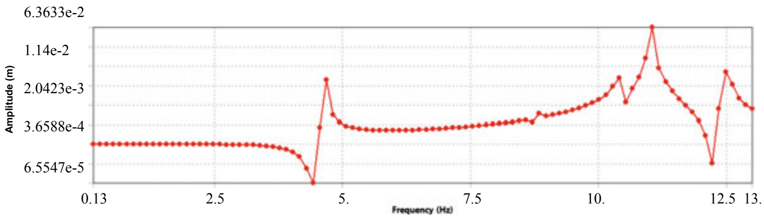
Modes no.	Mode shapes	Natural frequency (Hz)	Deflection values (m)
29		12.525	5.4465×10^{-3}

Table 3 FRF graphs of the intake section in x, y and z directions

Direction	FRF graph
x	
y	
z	

- The modal analysis of the intake section has been carried out where the 24th mode shape shows the largest deformation area to the structure at 11.024 Hz of natural frequency.
- The harmonic analysis has been successfully accomplished and the result shows that, the maximum deflection occurred in *y*-direction with 2.3202 m of amplitude.

Acknowledgements The authors would like to acknowledge Universiti Sains Malaysia for providing financial assistance through grant (203.PMEKANIK.6071370).

References

1. Chaphalkar SP, Khetre SN, Meshram AM (2015) Modal analysis of cantilever beam structure using finite element analysis and experimental analysis. Ajer 4(10):178–185
2. Elmenshawy MRE (2015) Static and dynamic analysis of concrete gravity dams. Tanta University
3. Khan IA, Awari GK (2014) Analysis of natural frequency and mode shape of all edge fixed condition plate with uncertain parameters 3(2):9277–9284
4. Mohanty P, Rixen DJ (2005) Identifying mode shapes and modal frequencies by operational modal analysis in the presence of harmonic excitation
5. Ramesha CM, Abhijith KG, Singh A, Raj A, Naik CS (2015) Modal analysis and harmonic response analysis of a crankshaft. Int J Emerg Technol Adv Eng 5(6):323–327
6. Silveira IV, Pedroso LJ (2018) Analysis of natural frequencies and modes of vibration involving interaction dam-reservoir foundation for concrete gravity dams. In: Third international dam world conference
7. Takasu S, Hirose T (2009) Water intake structures for surface and subsurface waters. Water Storage, Transp Distrib