Static and free vibration analysis of gravity dam under the influence of hydrostatic pressure using ANSYS finite element models

Pandimani no family name (mani.p@gmrit.edu.in)
GMRIT
Ravindra Vipparthy
JNTUK

Research Article

Keywords: FE analysis of gravity dam, ANSYS, Mode shapes, and natural frequencies, Different reservoir water depths, Stress analysis

Posted Date: July 13th, 2022

DOI: https://doi.org/10.21203/rs.3.rs-1823407/v1

Abstract

Dams are considered to be very important structures as they play an important role in the economic and social development of the area in which it is being constructed as well as utilized for hydropower generation. The concrete gravity dam is a solid structure that retains its stability against the design loads through its self-weight alone. The dam considered in this study is a structure of the Indirasagar polavaram gravity dam which is located in the village called Ramayyapeta comes under the West Godavari District, Andhra Pradesh. In this study, the finite element (FE) method is employed to numerically analyze the 2-D concrete gravity dam. The main objective of this study is to perform a linear FE analysis of the Indirasagar Polavaram gravity dam by varying the upstream hydrostatic pressure with constant downstream tailwater pressure. In the static analysis parameters like displacements, stresses and strains are assessed at each node, and their corresponding distributions are noticed. Subsequently, in the dynamic analysis approach, only free vibration analysis (Modal analysis) is done by considering only the dam's weight, the first five-mode shapes and their respective natural frequencies are extracted using ANSYS. The linear material behavior of the dam is modeled using the quadrilateral 4 node PLANE182 elements and the hydrostatic pressure is applied as triangular distribution. From the numerical analysis, it is concluded that the maximum tensile stresses are accumulated at the heel portion of the dam for cases 1 to 4, whereas it is observed distributed along the dam base for the rest of the cases. The lateral displacement and strain are observed maximum for case 1. The First mode's natural frequency of 5.81 Hz is ascertained and the maximum frequency and displacement are produced under the fifth mode.

1. Introduction

A dam can be defined as a barricade constructed across a river or stream. The structure is generally constructed to store a large amount of water and is constructed where a relatively sound and competent rock is available for abutment and foundation. Generally, the gravity dam is designed to withstand the hydrostatic forces by its self-weight alone [1]. Typically the gravity dams consist of overflow and non-overflow sections. The purposes of dam construction are for the generation of hydroelectric power, navigation, fish and wildlife enhancement, flood damage reduction, water quality and supply, and recreation. The stresses, strains, and displacement responses under different reservoir depths and their natural frequencies of the dam are required to assess the concentrations and distributions of these parameters within the dam body. The linear analysis and the free vibration analysis of the dam presented in this study will give prior information on the dam behavior before attempting the intensive nonlinear dynamic responses.

To achieve numerical solutions to any engineering problems using the finite element analysis (FEA) method, either one needs to formulate the computer program or use the commercially available numerical software. Currently, several numerical software such as LS-DYNA, ABAQUS, ANSYS, etc. is available to model and analyze any types of complex engineering structure and determine their responses such as stresses, deformations, strains, etc. under the action of external loads. The finite element analysis using the ANSYS program is used to solve a wide variety of engineering problems which has a comprehensive graphical user interface (GUI) that provides users friendly access to develop any type of structural model or write a program using code commands. In the ANSYS family, two basic solution methods as h-method and the p-method are available to analyze any structural problems: any type of analysis (linear or nonlinear) can be done using the h-method whereas the p-method can only apply to linear static analysis. In this study, the h-method is used for simulation purposes.

The response of the dam under seismic analysis considering the dam-reservoir-foundation effect is done by numerous researchers to assess the dynamic responses [2, 3, 4, 5]. Various researchers in the past have used the

ANSYS tool to analyze the 2-D gravity dam under static and dynamic loads, few works related to finite element analysis of dams are presented from the available literature. Shang et al. developed a 2-D finite element model with the varying position of the gallery in the horizontal as well as vertical directions to investigate the stress field concentrations on the dam structure [6]. They concluded that when the gallery spacing was increased from 3 to 9 m away from u/s face, the maximum tensile stresses near the gallery reduced gradually up to 8m and then remains stable, also the maximum tension was achieved at the heel portion. Chen et al. conducted a seismic analysis of a gravity dam using the two-dimensional FE models of two different gravity dams [7]. They investigated the influence of two boundary conditions such as viscous boundary and spring viscous boundary and their seismic input models on the dam-foundation-reservoir system. They concluded that the results in terms of hydrodynamic pressure, principal stresses, and displacements under the viscous boundary and spring viscous boundary conditions are found in good agreement. Seyed and Khiavi developed a 2D finite element model of the Koyna dam and performed free vibration analysis with considering prestress and without considering prestress effect and analyzed by considering the presence and absence of dam mass [8]. They analyzed for first four made-shapes and compared the predicted frequencies with the reference frequency available in the literature. They concluded that the highest reduction in modal frequency was achieved for the case without considering the prestress effect. Barıs Sevim developed 2-D numerical models using the ANSYS package to assess the influence of geometrical dimensions on the base width to dam height ratio (L/H) subjected to the seismic response [9]. They consider five different L/H ratios and concluded that the ratio with L/H = 1 predicts better performance against the seismic excitations of gravity dams. They also extracted the first nine modes and concluded that the natural frequency increases with increasing L/H ratio but the model with a 1.25 L/H ratio show more rigid compared to other models. Asghari et al. developed 2-D FE models of the Koyna dam with considering the foundation mass and explored the effect of dam-foundation- reservoir interaction on the seismic excitations [10]. The numerical results are compared with the already published works and they suggested that the incorporation of foundation mass had a significant effect on the nonlinear seismic response of the gravity dam. Ghaedi et al. developed 2-D FE models of a roller-compacted concrete dam by considering the flexible foundation-dam reservoir interaction subjected to earthquake excitations [11]. They used the horizontal and vertical time history data of the Koyna dam and assessed the responses such as displacements, stresses, accelerations, and crack evolution of the developed dam model. They stated that the response of a dam considering a flexible foundation had a great influence on the seismic behavior compared to a rigid foundation. Kartal et al. conducted a finite element analysis of the RCC dam to assess the seismic responses under six different reservoir lengths [12]. They concluded that as the reservoir length increases the displacement increases up to 3h length and beyond this, the displacement remains ineffective, hence they suggested that 3h length of the reservoir under nonlinear analysis is adequate to assess the seismic response of the RCC dam. Li et al. did a stress and seepage analysis of roller-compacted concrete (RCC) gravity dam by constructing an impervious layer with four different types of concrete material on the upstream (u/s) side using the ANSYS software [13]. The dam was subjected to two types of loadings that are normal and flood check water levels. They concluded that the tensile stresses and compressive stresses are maximum at the heel and toe region respectively. They suggested that the poor control of seepage was observed for the impervious layer constructed with three-graded RCC concrete. Khosravi and Heydari performed a 2D finite element analysis of a gravity dam with full and empty reservoir conditions under rigid and flexible type foundations and assessed the mode shape and natural frequencies [14]. They suggested that the frequency becomes maximum when the foundation was rigid and under empty reservoir conditions. Khosravi and Heydari did shape optimization analysis on the 2-D gravity dam of Pine-Flat, Koyna, and idealized triangular type under different foundation reservoir conditions using FE analysis [15]. They extracted the mode-shape and natural frequencies of these dam types under four different cases and compared their results with the reference results available in the literature. They concluded that the dams under empty conditions with rigid foundations give maximum natural frequency. In this study, FE analysis of a 2-D concrete gravity dam is done using the ANSYS finite element analysis package to assess the responses of the dam under different reservoir water depths. Subsequently, the mode shapes and natural frequencies of dams are predicted under the free vibration method.

1.1 Static and modal analysis

In the case of static analysis, the structures are analyzed under the steady-state loading conditions where the induced responses like displacements, stresses, strains, and internal forces in structural members that are caused by the external static loads are evaluated. The vibration characteristics such as natural frequencies and mode shapes of a structure can be evaluated using modal analysis. The modal analysis can also provide former information needed for doing other dynamic analyses such as harmonic or transient analysis. In ANSYS the modal analysis can be considered a linear analysis by only considering the linear material properties without any external loads. The ANSYS provides various mode extraction methods such as Unsymmetric, Block Lanczos, damped, QR damped, PCG Lanczos, and supernode methods, among these methods the Block Lanczos method, is used to extract the mode shape of the 2-D dam in this study.

1.2 Indirasagar polavaram dam

The Polavaram dam which is located in the state of Andhra Pradesh on the Godavari River is a multi-purpose project, about 34 km u/s of Kovvur near Polavaram village of Rajahmundry district. Under the left canal for East Godavari and Visakhapatnam districts and from the right canal for West Godavari and Krishna districts, the Polavaram project envisages irrigation benefits up to 7.20 Lakhs Acres of land, and also it generates 960 MW hydro-electric power. Besides, under its left canal, this project envisages 23.44 TMC of water supply for industries and steel plants in Visakhapatnam. In addition, it diverges 80 TMCft. of water to the Krishna river through the right canal. The existing project work is a modified version of the actual Rampadasagar project which is now selected about 2.0 km u/s from the old Rampadasagar project. The Polavaram project is essentially diversion work with 2130 million cusecs live storage capacity.

The reservoir formed by the dam submerges an area of 677 km^2, out of which 601 km^2 is in Andhra Pradesh, 24.00 km^2 is in Madhya Pradesh and 12 km^2 is in Orissa. The cross-section view of the gravity dam modeled in this study is shown in Fig. 1. The non-overflow section has a 44m height from its base, a top width of 9.675m, and a base width of 37.51m. The upstream face of the dam has a slope of 10:1 up to 35.72m and then straight up to the top level of the dam. The dam has a downstream slope of 1:0.85 up to 28.54m and then straight up to the top level. The upstream water level at F.R.L is 35.72m elevation from the base of the dam and the maximum tailwater level is kept constant at 21.66m.

1.3 Objectives of the present study

The main object of this study is to develop a 2-D FE model and perform a linear numerical analysis of the Indirasagar Polavaram concrete gravity dam for static and dynamic behavior by using the sophisticated numerical analysis package ANSYS. The dam is analyzed for different reservoir water depths with a constant tailwater level and parameters like displacements, stresses, and strains, and their distributions throughout the body are estimated through the contour diagrams for better understanding purpose. Later, it is analyzed for free vibration response under the dam self-weight alone to extract the first five-mode shapes and natural frequencies of the dam.

2. Fe Modeling Of The Dam Section

The modeling of a concrete gravity dam is done by one of the Finite Element Method (FEM) commercial package software ANSYS 14. The dam is modeled in two-dimensional (2D) using the Quadrilateral 4node PLANE182 element [16]. The behavior of the dam element considered here is the plane stress condition and the mesh size length is taken as 5. The material properties in terms of Young modulus, Poisson's ratio, and density of concrete are assigned for the PLANE 182 element to simulate the linear properties of the dam. The dam base is assigned with a zero displacement in x and y-directions such that it is fully constrained at the base. The FE development approach through key points, lines, areas, and discretization of the 2-D dam is depicted in Fig. 2.

2.1 Element description and material properties

The 2-D solid structures are modeled by using the PLANE182 element available in the ANSYS library, which can act as plane stress, plane strain, or an axisymmetric element. It has four nodes and each node has two translational degrees of freedom i.e., in the x and y-direction. It supports plasticity, stress stiffening, hyperelasticity, large deflection, and large strain capabilities [16]. It also supports mixed formulation capability for simulating the deformations of incompressible elastoplastic and hyperelastic materials. The concrete behavior is modeled by assigning the linear elastic properties such as Young's modulus = 27386.12 MPa, Poisson's ratio = 0.15, and the density = 2.4 E-5 N/mm^3 to the PLANE182 elements. The geometry of the element used to develop the 2-D dam is shown in Fig. 3.

2.2 Loading and boundary conditions

The hydrostatic pressure at the upstream and downstream face of the dam, and the uplift pressure considered in the static analysis for case 1 along with the boundary condition are shown in Fig. 4. The dam is analyzed for different reservoir water levels at the upstream face with a constant tailwater level at the downstream face up to 21.66 m height, to investigate the critical points of stresses, strains, displacements, and their distribution over the body of the dam. Different cases adopted in this study for the analysis of the dam are shown in Table 1.

Table 1: Hydrostatic pressure acting on the dam for different reservoir water levels

Loading cases	u/s water levels	d/s water levels	Hydrostatic pressure (MPa)			
			u/s face	bottom face		d/s face
				at heel	at toe	
1	35.72m	21.66m	0.36	0.36	0.22	0.22
2	30m	21.66m	0.30	0.30	0.22	0.22
3	25m	21.66m	0.25	0.25	0.22	0.22
4	20m	21.66m	0.20	0.20	0.22	0.22
5	15m	21.66m	0.15	0.15	0.22	0.22
6	10m	21.66m	0.12	0.12	0.22	0.22
7	5m	21.66m	0.05	0.05	0.22	0.22

3. Fe Analysis Results And Discussions

The FE analysis predicted responses of dams in terms of stresses, strains, and displacements under different reservoir depths, and the first five-mode shape diagrams under the free vibration criteria and the result discussions are illustrated in this section.

3.1 Predicted displacements

The horizontal displacement of the dam is found maximum for case 1 in which the u/s water level is up to 35.72m in height with a constant tailwater level of 21.66m. Since the depth of the reservoir on the u/s side in case 1 is more than that of other cases, hence the highest displacement is achieved for case 1, and the dam is displaced towards the d/s side. As the reservoir water level on u/s decreases, the horizontal displacement towards the d/s side also decreases for cases 2, 3, and 4 as illustrated in Table 2. For cases 5, 6, and 7 negative displacements have occurred because the u/s water level is less than the tailwater level as a consequence the dam displaces towards the u/s side, hence negative displacement is obvious for these cases. The node Vs displacement plot for u/s face and d/s face is shown in Figs. 6 and 7 respectively, which suggests that the node at the crest level at u/s and d/s face experiences the highest positive and negative horizontal displacements under case 1 and case 7 respectively. Figure 7(a) depicts the predicted maximum displacements for all the cases. It is evident from Fig. 7(a) that the maximum positive and negative horizontal displacements have occurred for case 1 and case 7 respectively. Since the height of the u/s water level is reduced and the tailwater level which is having more depth than the u/s water levels for cases 5, 6, and 7, consequently negative displacement towards the d/s is more for case 7 as compared to the cases 5 and 6 as shown in Fig. 7(a).

Table 2
Predicted responses of FE model under varying reservoir water depths

Loading case	Stresses (MPa)				Strains				Max. disp. (mm)	
	SX		SY		EPELX		EPELY			
	Max.	Min.	Max.	Min.	Max.	Min.	Max.	Min.	UX	UY
1	8.25E-02	-0.24429	0.48061	6.40E-02	1.35E-07	-1.08E-05	1.70E-05	-8.38E-06	6.99E-04	1.49E-04
2	4.84E-02	-0.20334	0.27553	-0.17801	5.74E-08	-8.33E-06	9.76E-06	-5.73E-06	3.00E-04	-6.07E-05
3	2.49E-02	-0.16734	0.14507	-0.13474	9.26E-08	-6.44E-06	5.14E-06	-4.59E-07	1.26E-04	-4.23E-05
4	8.98E-03	-0.13158	5.36E-02	-0.11493	9.59E-08	-4.76E-06	1.90E-06	-4.09E-06	4.97E-05	-3.26E-05
5	4.71E-03	-0.10867	1.37E-02	-0.10179	1.71E-07	-3.39E-06	5.19E-07	-3.64E-06	-4.50E-05	-2.66E-05
6	4.26E-03	-0.10472	1.12E-02	-9.61E-02	1.67E-07	-3.28E-06	4.13E-07	-3.44E-06	-4.79E-05	-2.49E-05
7	3.40E-03	-0.10144	7.96E-03	-9.11E-02	1.63E-07	-3.19E-06	3.42E-07	-3.26E-06	-5.03E-05	-2.37E-05

3.2 Stresses and strains

The maximum and minimum normal stresses have occurred for case 1 and case 7 respectively as can be evident from Table 2. The stresses are directly proportional to the hydrostatic pressure acting on the u/s side and hence the stresses are reduced with the reduced water levels as shown in Fig. 7(b). The maximum compressive stresses are concentrated just above the heel at the u/s face for cases 1 to case 4, whereas for the remaining cases the maximum compressive stresses are concentrated just above the toe at the d/s face of the dam since the height of tailwater is more than the u/s water level in these cases. The maximum tensile stresses are concentrated at the heel of the dam for the first four cases whereas for the remaining cases it is found distributed throughout the base of the dam. The maximum and minimum compressive strains are experienced for case 5 and case 2 respectively as depicted in Fig. 7(c). The maximum compressive stresses are concentrated above the heel at the u/s face of the dam for the first three cases whereas these stresses are concentrated just above the toe of the dam at the d/s face. The tensile strains for all the cases are found distributed throughout the base of the dam as well as the upper portion of the dam section. The output results captured by the FE models in terms of deformation, stresses, strains, and displacement contour diagrams are shown in Fig. 8. The contour plot shown in Fig. 8 represents the maximum concentration and their distribution over the body of the dam section which enables a better understanding of the dam behavior.

3.3 Modal analysis

To obtain the first five-mode shapes and the natural frequency, five 2-D FE models are developed and considering only the self-weight of the dam for the model analysis. The base of the dam is constrained in x and y-directions by assigning a zero displacement. Then the dam is analyzed for free vibration to achieve the first five-mode shapes and their corresponding natural frequencies. The natural frequency of the FE model increases from 5.81 Hz to 17.28 Hz from mode shapes 1 to 5 and the respective peak displacements are illustrated in Table 3. From Fig. 9 (a), it is evident that mode 1 achieved the lowest frequency and the highest frequency was obtained for the last mode. The maximum and minimum peak displacements under the free vibration were experienced for mode 5 and mode 2 respectively as shown in Fig. 9(b). The mode shapes predicted through FE analysis of the dam are shown in Fig. 10.

Table 3
Mode shape and natural frequency under free vibration

Mode shape	Time/Frequency (Hz)	DMX
1	5.81	$6.75\text{x}10^{-4}$
2	6.97	$4.78\text{x}10^{-4}$
3	11.26	$6.09\text{x}10^{-4}$
4	15.70	$5.27\text{x}10^{-4}$
5	17.28	$6.80\text{x}10^{-4}$

4. Conclusions

In this study, static linear analysis of the 2-D gravity dam is performed using FE models under seven different reservoir water levels with constant tailwater level conditions. Free vibration analysis is done to extract the natural frequencies and first five-mode shapes of dams. From this study, the following conclusions are achieved. The maximum deflection of $6.99\text{x}10^{-4}$ mm and $1.49\text{x}10^{-4}$ mm in the x and y-direction for 35.72m reservoir depth is obtained at the top crest level and near the FRL level respectively. The maximum normal stress of $8.25\text{x}10^{-2}$ MPa and 0.482 MPa in the x and y-direction are developed just above the heel portion of the dam. The maximum elastic strain of $1.71\text{x}10^{-7}$ in the x-direction is distributed at the base and top portion of the dam under case 5. The maximum elastic strain of

$1.70x10^{-5}$ in the y-direction is found at the heel portion under case 1. The maximum compressive stresses are concentrated near the heel portion for cases 1 to 4, whereas from cases 5 to 7 it is accumulated near the toe portion of the dam. The maximum tensile stresses are concentrated at the heel portion of the dam up to case 4, whereas for the remaining cases it is observed distributed along the dam base. The natural frequencies for the first five-mode shapes are predicted and the natural frequency of 5.81 Hz is achieved for the first mode of the dam under free vibration. The developed FE model well predicted the linear and modal responses of the dam.

5. Scope For Further Study

- The dam can be analyzed for linear and non-linear responses for dam-foundation-reservoir systems.
- The dam can be analyzed for forced vibration analysis under earthquake excitations.
- The dam can be analyzed for Flexible and rigid foundation conditions under hydrodynamic forces.
- A crack analysis of the dam can be included.

Declarations

Compliance with Ethical Standards

Funding: No external funding was received for this work

Conflict of interest: The authors report no conflict of interest for the material explained in this article.

Informed consent: All the authors are aware and agreed to the submission of the manuscript.

References

1. Md H, Ali MdR, Alam MdN, Haque MJ Alam (2010) Comparison of Design and Analysis of Concrete Gravity. Dam" 3:18–28. DOI: 10.4236/nr.2012.31004
2. Chen Wang H, Zhang Y, Zhang L, Guo Y, Wang (2021) Influences on the Seismic Response of a Gravity Dam with Different Foundation and Reservoir Modeling Assumptions", Water. 13:1–22. https://doi.org/10.3390/w13213072. Thiri Thon Thira Htun
3. Mohsen Ghaemian A, Noorzad, and Hamid Mohammadnezhad (2018) Assessment of Foundation Mass and Earthquake Input Mechanism Effect on Dam–Reservoir–Foundation System Response. Int J Civil Eng. https://doi.org/10.1007/s40999-018-0325-9
4. Tan H, Chopra AK (1995) Earthquake analysis of arch dams including dam-water foundation rock interaction. Earthq Eng Struct Dyn 24:1453–1474
5. Mohammadnezhad H, Ghaemian M, Noorzad A (2019) Seismic analysis of dam-foundation-reservoir system including the effects of foundation mass and radiation damping. Earthq Eng Eng Vib 18(1):203–218
6. Gao S, Chen Y, and Junhong Xiang (2021) Finite element analysis of stress in the gallery of gravity dam on inclined bedrock based on ANSYS model, IOP Conference Series: EarthEnvironmental Science, pp:1–6. doi:10.1088/1755-1315/826/1/012067
7. Chen D, Hou C, Feng, Wang (2021) "Influences on the Seismic Response of the Gravity Dam-Foundation-Reservoir System with Different Boundary and Input Models", Shock and Vibration, pp:1–15. https://doi.org/10.1155/2021/6660145

8. Seyed Reza Jafari and Majid Pasbani Khiavi (2019) Parametric Study of the Modal Behavior of Concrete Gravity Dam by Using the Finite Element Method. Civil Eng J 5:12. http://dx.doi.org/10.28991/cej-2019-03091437
9. Barıs S (2018) "Geometrical dimensions effects on the seismic response of concrete gravity dams", Advances in Concrete Construction, Vol.6,No.3,pp:269–283 DOI : https://doi.org/10.12989/acc.2018.6.3.269
10. Erfan Asghari R, Taghipour, Mohsen Bozorgnasab and Mojtaba Moosavi (2018) Seismic Analysis of Concrete Gravity Dams Considering Foundation Mass Effect. KSCE J Civ Eng 22:4988–4996. DOI 10.1007/s12205-017-0278-6
11. Ghaedi K, Hejazi F, Ibrahim Z, Khanjai P (2017) Flexible Foundation Effect on Seismic Analysis of Roller Compacted Concrete (RCC) Dams Using Finite Element Method. KSCE J Civ Eng 22:1275–1287. https://doi.org/10.1007/s12205-017-1088-6
12. Kartal ME, Ayse BS (2017) Assessing seismic response of a 2D roller-compacted concrete dam under variable reservoir lengths. Arab J Geosci 118. https://doi.org/10.1007/s12517-017-3271-y
13. Li Ming-chao, Guo Xin-yu, Shi J, Ze-biao Z (2015) Seepage and stress analysis of anti-seepage structures constructed with different concrete materials in an RCC gravity dam. Water Sci Eng 8(4):326–334. http://dx.doi.org/10.1016/j.wse.2015.10.001
14. Shiva Khosravi and Mohammad mehdi Heydari (2013) World Appl Sci J 22(4):538–546. DOI: 10.5829/idosi.wasj.2013.22.04.551. "Modelling of Concrete Gravity Dam Including Dam-Water-Foundation Rock Interaction"
15. Shiva Khosravi and Mohammad mehdi Heydari (2013) Design and Modal Analysis of Gravity Dams by Ansys Parametric Design Language. Walailak J Sci & Tech 12(2):167–180
16. ANSYS (2007) "A Finite Element Computer Software and User Manual for Nonlinear Structural Analysis". ANSYS, Inc., Canonsburg, PA

Figures

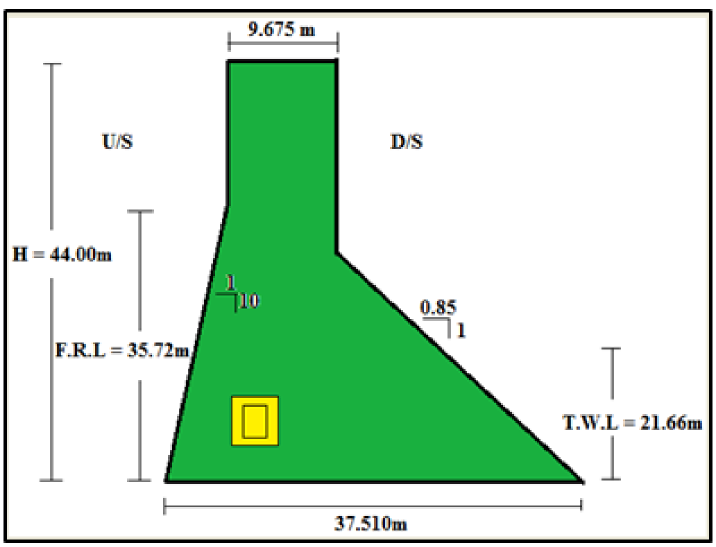

Figure 1

Idealised sectional view of the Indirasagar gravity dam.

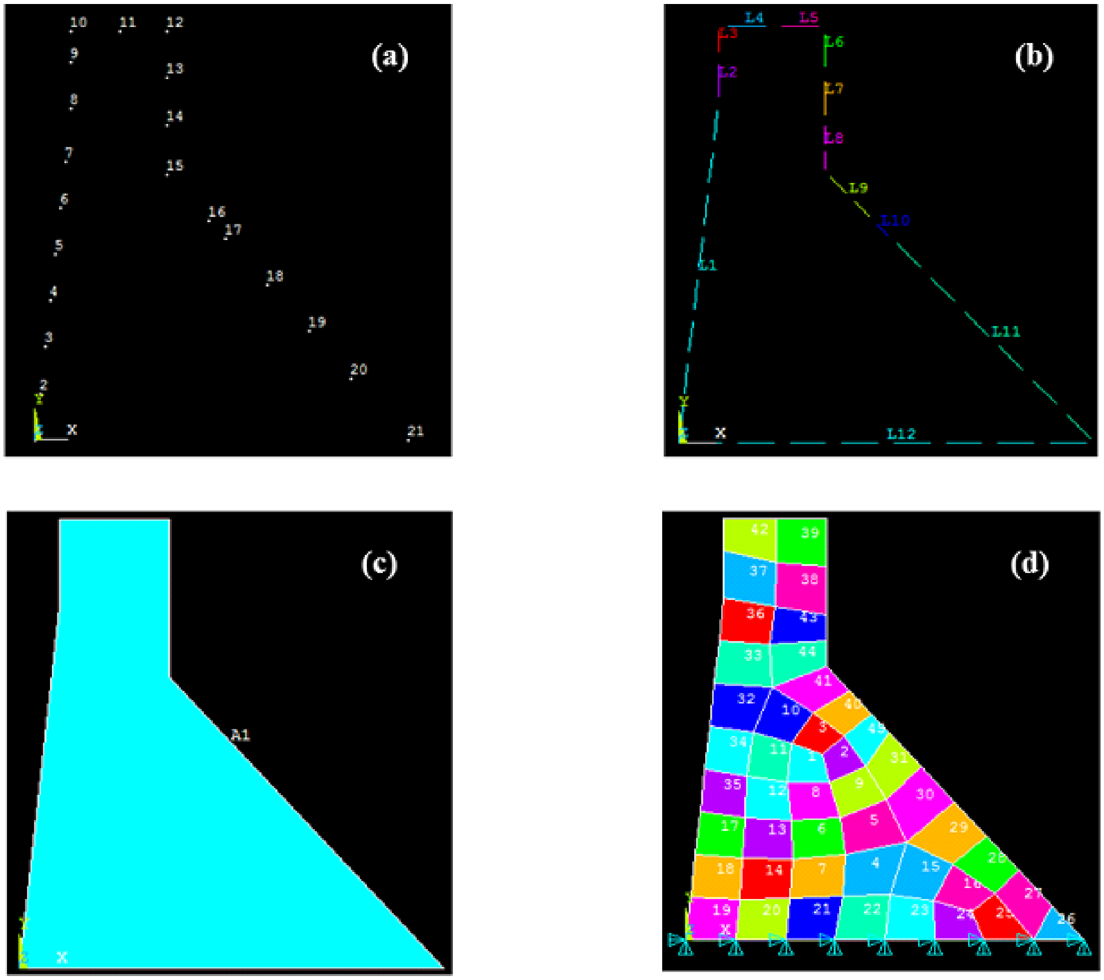

Figure 2

Development of 2-D dam FE models: (a) key points, (b) line diagram, (c) area diagram, and (d) discretized elements

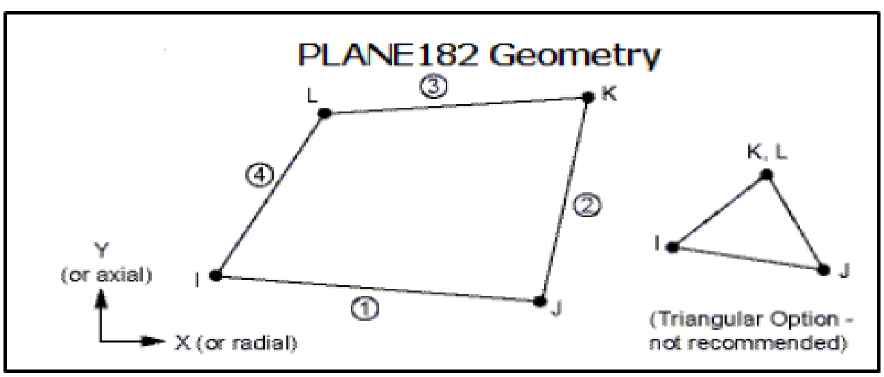

Figure 3

Geometry of 2-D 4node plane 182 elements

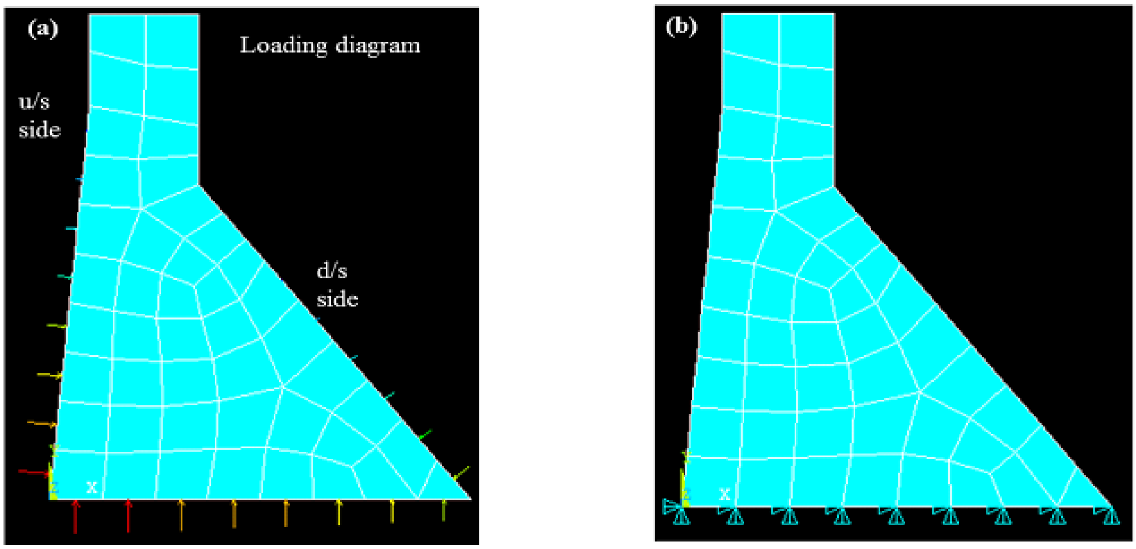

Figure 4

Developed FE model: (a) hydrostatic pressure loading and (b) boundary condition

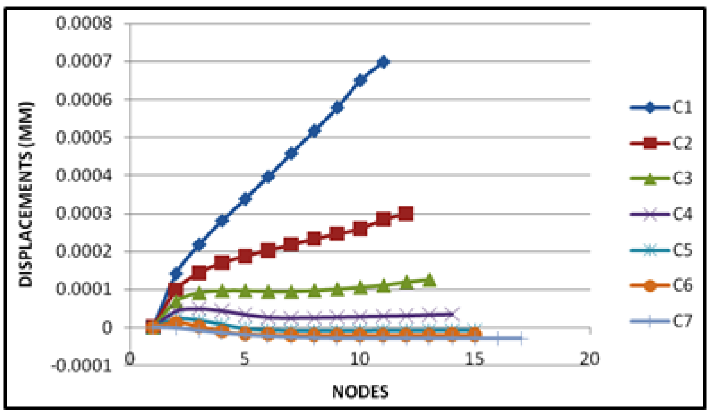

Figure 5

shows displacements in upstream face nodes for different cases

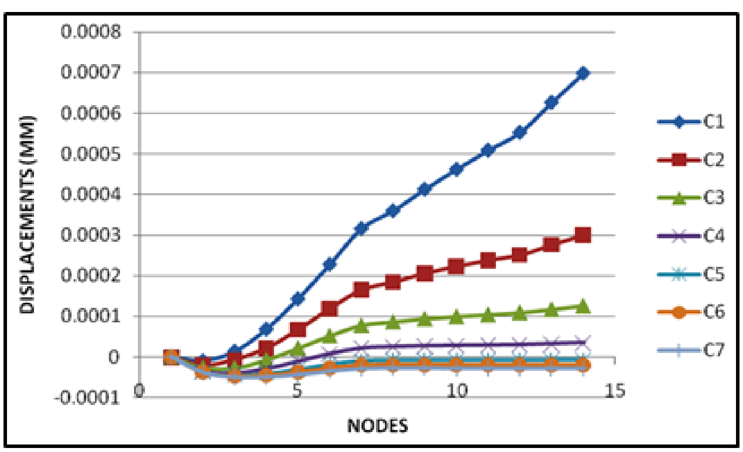

Figure 6

shows displacements in downstream face nodes for different cases

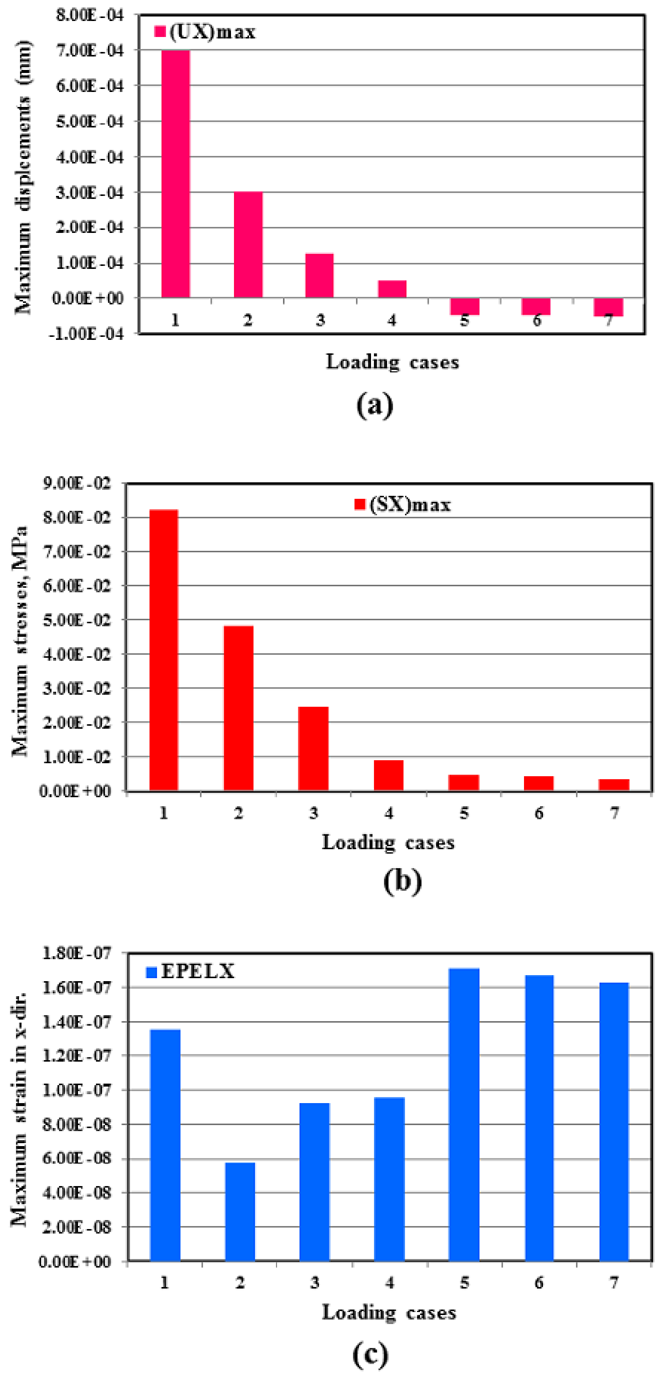

Figure 7

FE results under different loading cases in x-directions: (a) maximum displacements, (b) maximum stresses, and maximum strains.

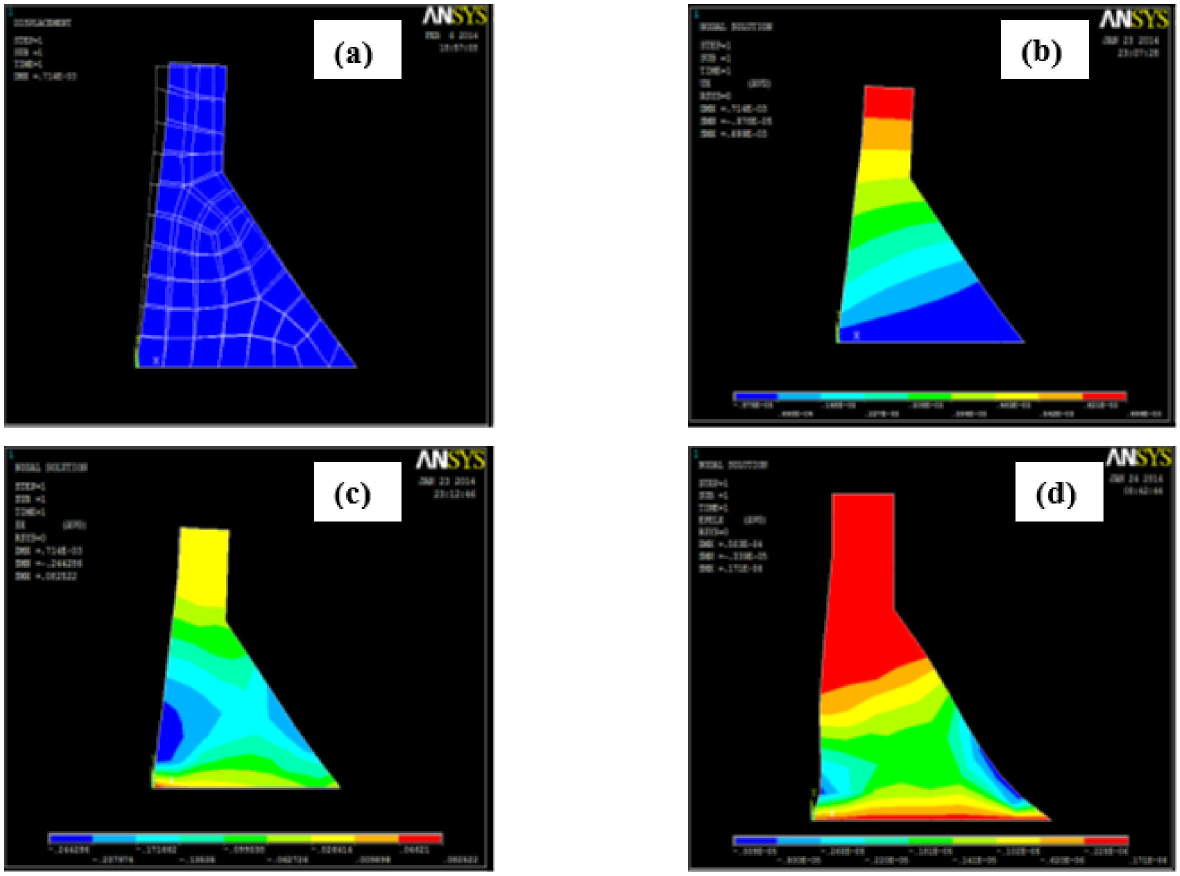

Figure 8

The FE analysis output results: (a) deformation diagram, (b) maximum displacement, (c) maximum stresses, and (d) maximum strains.

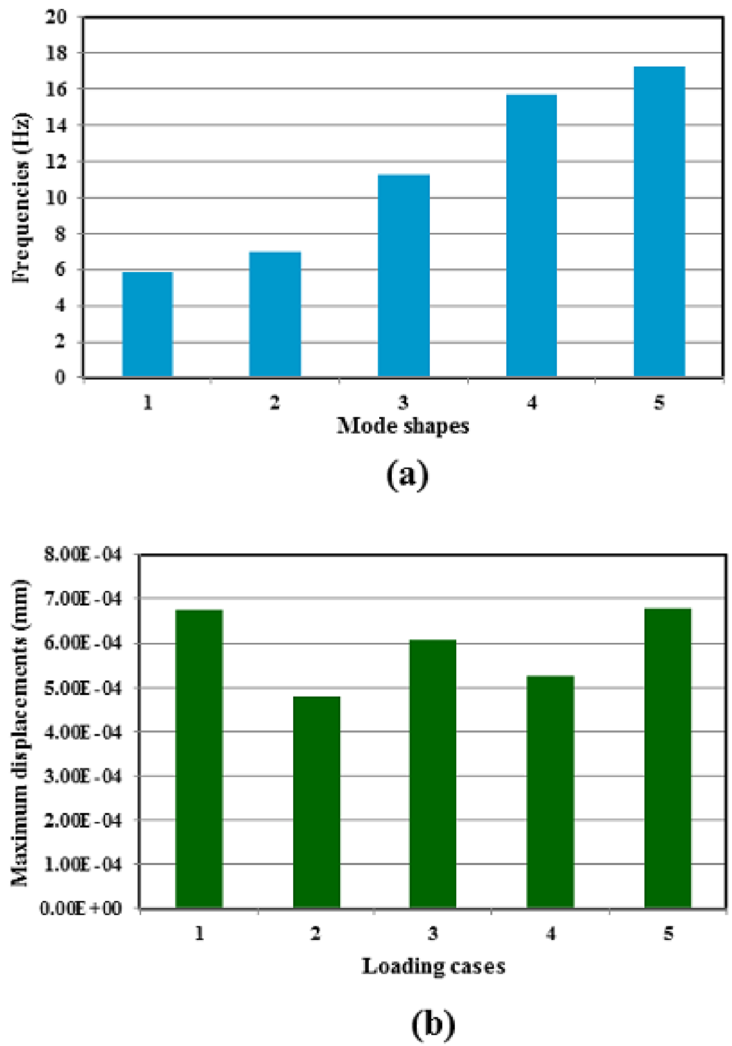

Figure 9

Free vibration analysis results: (a) natural frequencies for the first five modes and (b) peak deflections at each mode shape.

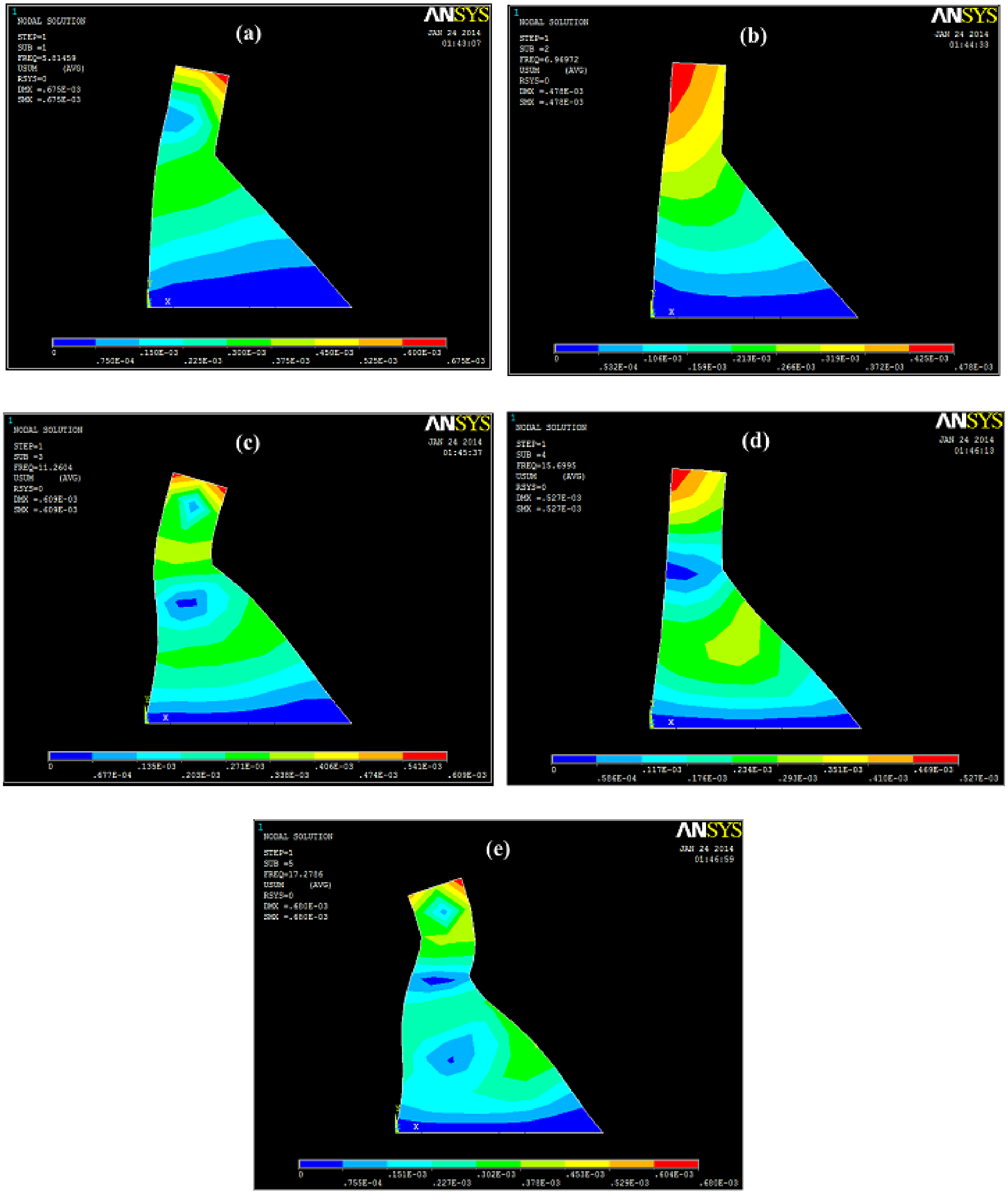

Figure 10

Mode shapes of the dam under free vibration