



## Finite Element Analysis of Concrete Gravity Dam

Tuba Aydin

Suleyman Demirel University, Faculty of Engineering and Natural Sciences, Department of Civil Engineering, Isparta, Turkey

✉: [tubaaydin@sdu.edu.tr](mailto:tubaaydin@sdu.edu.tr)  ID: 0000-0002-7404-5020

Received: 26.09.2024, Revised: 6.11.2024, Accepted: 20.11.2024

### Abstract

Dams have been structures that have served to meet humanity's water needs for centuries. Because they are large structures, critical investigations must be made before they are built. However, these investigations and scaled or one-to-one modeling of dams have become costly today. At this point, package programs working with finite elements are used functionally. ANSYS package program is also a frequently preferred package program that provides results close to reality. In this study, a 3D dam body modeled with ANSYS was examined. While modeling, the water load acting outside the dam's own weight and the material properties of the dam body were taken into account and analyzed. The dam was modeled in accordance with its exact dimensions without any scaling. This study examines the stress and displacement of a concrete gravity dam under half-filled and fully filled conditions, highlighting significant differences in structural behavior. It was determined that the displacement and stress values found for the full state were higher than the half-full state.

**Keywords:** ANSYS, Concrete Gravity Dam, Finite Element Analysis

### 1. Introduction

Dams are structures that create reservoirs by blocking a river valley, allowing for water storage. In addition to storing water, they have fundamental benefits such as raising the water level and creating a large water surface [1]. It is very important to build dams in order to raise the water level and meet the water and irrigation water needs in the region where the river is located. In Turkey, the flow regimes of rivers are irregular, so it is necessary to store water during periods of abundance. In dry seasons, the stored water is used. For this purpose, many dams have been constructed [2]. To serve various purposes, dams and water-raising structures have been constructed since ancient times in human history. Since water is of great importance for human survival, it is important to build dams that meet the specific needs of the region.

Since dams are large and costly water structures, it is important to conduct experimental or numerical analysis before their construction to anticipate potential problems and predict the behavior of these structures. In recent years, this has been made possible through various software programs. Experimental studies for water structures can be costly, which is another reason why software programs are preferred. Additionally, these software programs use finite element methods and provide users with the ability to model the water structure in detail, allowing for realistic results. ANSYS is a software package that uses finite element methods and includes many modules. It is frequently used in solving engineering problems and is also effectively utilized in the field of civil engineering.



Ahemad et al. modeled a gravity dam in 3D using the ANSYS 2020 software package. They conducted analysis for both the filled and empty states of the dam. Stress and pressure values under the effect of water load were determined. Analysis were performed for different loading combinations, and it was concluded that the dam was safe [3].

Saxena and Patel examined the dynamic characteristics of the Shahpurkandi Dam, a concrete gravity dam with a height of 55.5 meters, using the ANSYS finite element software package. They analyzed the modal response of the dam by considering the condition when the reservoir is fully filled. The model was created taking into account the dam-foundation interaction. It was also found that the fully filled reservoir condition is more critical compared to the empty reservoir condition. It was demonstrated that ANSYS is a reliable and suitable software for investigating the dynamic behavior of a structure [4].

Sharma and Nallasivam computed the static response of a 2D model of the Bhakra concrete dam using the finite element software ANSYS. To investigate the dam-foundation interaction, the dam was modeled in two different ways: one including both the dam and foundation, and the other considering only the dam structure. It was found that the highest displacement value in the dam occurred at the crest [5].

Todorov et al., investigated the behavior of an old gravity concrete dam in Bulgaria under different load conditions. Various water loads in the reservoir, dependent on the seasonal variations in water quantity, were applied to the structure. The analysis conducted with ANSYS revealed that the reservoir was appropriately designed according to the dam's load-carrying capacity [6].

Vipparthy performed a finite element analysis of the Indirasagar Polavaram gravity dam by keeping the upstream hydrostatic pressure constant and varying the downstream tailwater pressure. The concrete gravity dam was modeled in 2D. The analysis investigated displacement and stress values, and it was observed that the maximum stress values accumulated at the heel of the dam. It was concluded that the developed finite element model accurately predicted the linear and modal responses of the dam [7].

Silveira et al., conducted a finite element analysis using a typical Brazilian dam profile and seismic data from a Brazilian earthquake. The aim was to investigate the impact of foundation-reservoir interaction on the dynamic behavior of concrete gravity dams, focusing on natural frequencies and vibration modes for free vibration analysis. The dam model was created using the APDL part of the ANSYS software. Stress and displacement parameters for the dam were determined. It was suggested that, in the dynamic analysis of dams, all relevant parameters should be analyzed consistently with the existing conditions in the dam-reservoir-foundation system [8].

Messaad et al., investigated the dynamic behavior of the dam-reservoir-foundation system under seismic loading using the ANSYS software. The Oued Fodda concrete dam, located in Chlef in northwestern Algeria, was selected as a case study. Variations in the foundation stiffness can lead to changes in the stress and displacement values of the dam. It was suggested that considering the dam-foundation interaction would be beneficial for obtaining more reliable results [9].

Gao et al., investigated the phenomenon of stress concentration occurring in gallery voids within concrete gravity dams. They attempted to create stress distribution maps around the gallery voids and the gallery design using the ANSYS software. It was observed that variations

in the shape and size of the galleries could lead to changes in the stress distributions of the dam [10].

Wei et al., studied a hydroelectric power plant dam. The concrete gravity dam in question was modeled using ANSYS, and stress and displacement values were investigated. The analysis for the reservoir being fully filled or at normal water levels showed that the stress and displacement parameters of the gravity dam can vary [11].

Habib et al., summarized and compared the general findings of recent studies on the structural performance of roller-compacted concrete (RCC) dams. It was found that commercial software packages using the finite element method can predict the behavior of RCC dams. The condition of cracks in RCC dams serves as an important indicator of damage formation [12].

Taylan and Aydin examined the behavior of the Darıderesi II Dam, located in Isparta, under the influence of three different earthquakes. The dam was modeled in detail, and the analysis results were obtained using ANSYS. Displacement and stress values were determined as a result of the analyses conducted for the Kocaeli, Düzce, and Dinar earthquakes. According to the analysis results, it was observed that these earthquakes would not cause any failure in the dam body [13].

In this study, a concrete gravity dam was modeled in 3D using the ANSYS software. The stress and displacement values were examined for both the half filled and fully filled conditions of the concrete gravity dam downstream, in addition to the dam's own weight. It was found that the displacement and stress values obtained for the half filled condition were lower compared to those for the fully filled condition.

In previous studies, the behavior of rock-fill dams under the effect of water load has been examined. However, in this study, the displacement and stress values of a concrete gravity dam were investigated.

This paper is divided into sections under the headings of Materials and Methods, Finite Element Method and ANSYS Software, and Numerical Analysis-Results. Under the Materials and Methods section, the working principle of the finite element method and the details of the modules used within the ANSYS software are described in detail. The Numerical Analysis-Results section provides visual representations of the stress and displacement results for the 3D dam model.

## **2. Materials and Methods**

### **2.1. Finite Element Method**

The finite element method has become a widely preferred technique in solving engineering problems thanks to the development of sophisticated software packages. Particularly in complex and large geometries, various analysis are conducted using the finite element method. Large geometries are divided into manageable smaller elements and modeled in either two or three dimensions. The behavior of the structure under various static and dynamic loads applied to these models is then studied.

Software packages that use the finite element method are preferred because they provide realistic results and produce outcomes in a shorter time compared to experimental studies. Additionally, creating scaled models of large and complex geometries and conducting various

analysis can be very costly. Therefore, software packages play a crucial role in solving engineering problems by saving time and reducing costs.

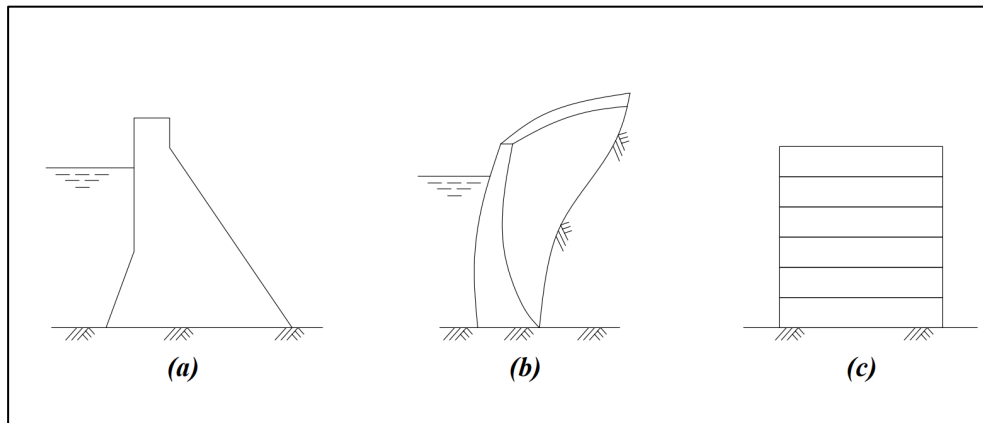


Fig. 1. Engineering structures

The behavior of various civil engineering structures, such as the concrete gravity dam body, arch dam body, and massive concrete blocks shown in Fig. 1, under different loads can only be analyzed using software packages. In Fig. 1. a, the concrete gravity dam body is nearly triangular in shape, although there is a slope in the upstream and downstream sections. Fig. 1. b shows an arch dam, which represents a dam body that continues in a parabolic shell shape. The material properties of the concrete gravity dam and the arch dam are also different from each other. Additionally, while the dam bodies are divided into meaningful geometries, both will be meshed with different elements. Fig. 1. c shows a massive concrete block used in the construction of a concrete gravity dam. Typically, a concrete layer 1.5-3.0 meters thick is poured every 6-10 days, leading to variations in the modulus of elasticity within each layer. Considering the influencing parameters within each structure and elasticity theory, it is not possible for theoretical methods to reflect realistic results. Given these considerations, numerical methods emerge as the sole solution for calculating stress values [14].

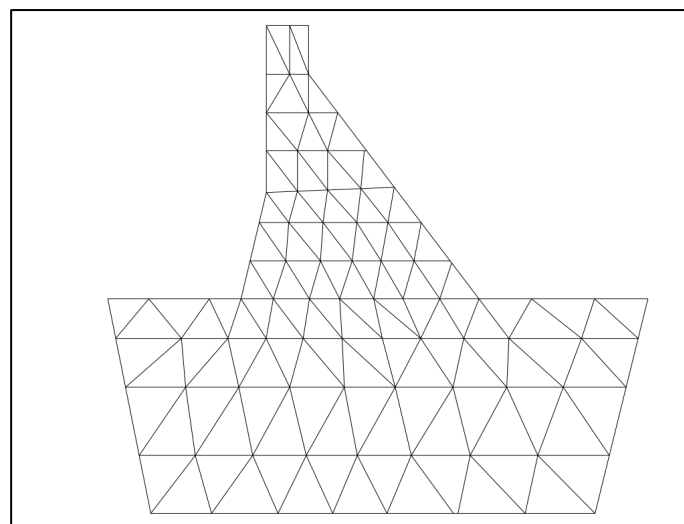


Fig. 2. Cross-section of the gravity dam divided into triangular elements

When using the finite element method, the structure to be analyzed is divided into finite elements. Fig. 2. shows the finite element representation of a concrete gravity dam body, divided into elements of a specific type. As seen in the figure, it consists of triangles of different

sizes. This way, the mesh structure conforms to the boundary conditions of the structure's shape. Additionally, it is worth noting that different types of elements can represent different material properties. The finite element method is employed to analyze complex structural problems in computer-based environments. Nowadays, this method is also preferred in construction, hydraulic, mechanical, and aerospace fields.

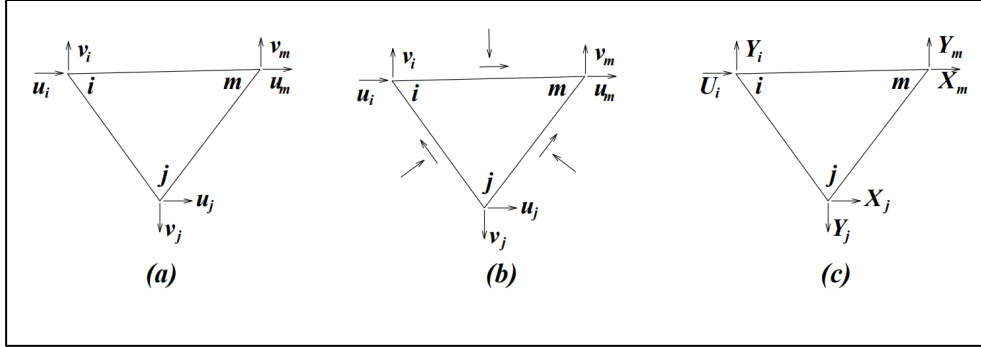


Fig. 3. Node displacements, equivalent node forces, and node loads of an element

The concrete gravity dam body in Fig. 2. is modeled as a 2-dimensional plate. The concrete gravity dam body can be divided into a specific number of triangular elements, as shown in Fig. 3. Each element has three nodes, and each node has two displacement components. Therefore, at each node of the element, there are a total of six displacement components [14].

The advantages of the finite element method include:

- It can be used for the analysis of structures with various shapes,
- Different material properties can be assigned to different elements used in the mesh structure,
- Due to the large number of unknowns, it is feasible to perform these computations on a computer [14].

The finite element method is a powerful numerical analysis technique that is widely preferred today. It is used not only for structural problems but also across many different engineering fields.

A simple model consisting of a dual-spring system is shown in the Fig. 4. The stiffness of the springs is defined as  $K_1$  and  $K_2$ , and the movement of the springs is restricted to the x-direction. The forces acting on nodes 1 and 2 are denoted as  $F_1$  and  $F_2$ , respectively. To determine the displacement parameter occurring in the springs;

$$F = K.X \quad (1)$$

**F**: Applied Force, **K**: Spring constant, **X**: Displacement

When an equation is formulated considering the parameters of the springs in the figure,

$$F_2 - (X_2 - X_1)K_2 = 0 \quad (2)$$

$$F_1 - XK_1 + (X_2 - X_1)K_2 = 0 \quad (3)$$

If this equation is written in matrix form, it will take the following form;

$$\begin{bmatrix} F_1 \\ F_2 \end{bmatrix} = \begin{bmatrix} K_1 & -K_2 & X_1 \\ -K_2 & K_2 & X_2 \end{bmatrix} \quad (4)$$

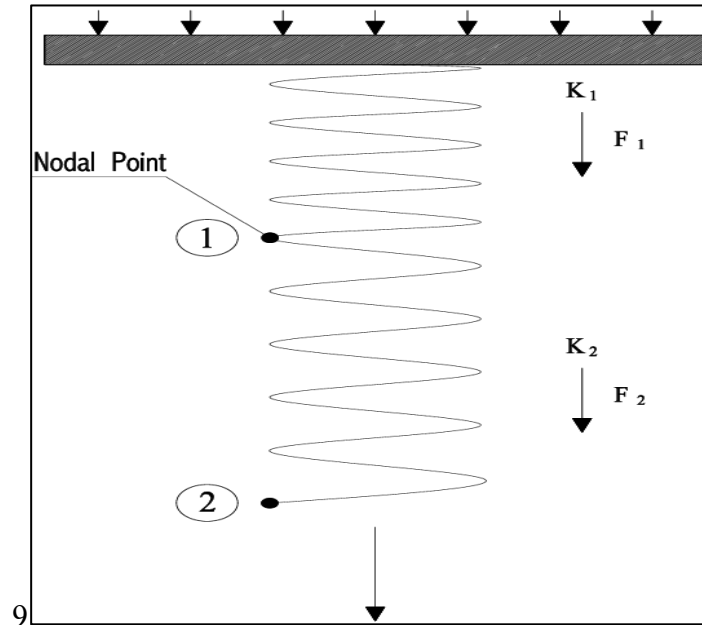


Fig. 4. Spring system

In this mathematical model, if the forces and spring constants are known, the displacement value occurring in the system can be calculated [15].

## 2. ANSYS Software

ANSYS is a package program frequently preferred in recent years for solving various engineering problems. It performs analysis using the finite element method and has two different modules: APDL and WORKBENCH. The APDL module assigns materials to models and analyzes their behavior under loads according to commands, while the WORKBENCH module performs analysis using visual commands on the screen without the need for command writing. The fundamental difference between the two modules, which operate on the same solution principle, is that the WORKBENCH module offers a more visual interface to the user. In this study, analysis were performed using the WORKBENCH module.

The Static Structural parameter within the WORKBENCH module was chosen for the analysis of the concrete gravity dam. The Static Structural parameter was modeled directly using the Space Claim drawing program available in the Geometry Fig. 5. shows the dimensions of the concrete gravity dam. As seen in the Fig. 5, the crest width of the concrete gravity dam is 7 meters, and the crest length is 257 meters. The concrete gravity dam model was created taking into account the crest length of 257 meters as shown in Fig. 4.

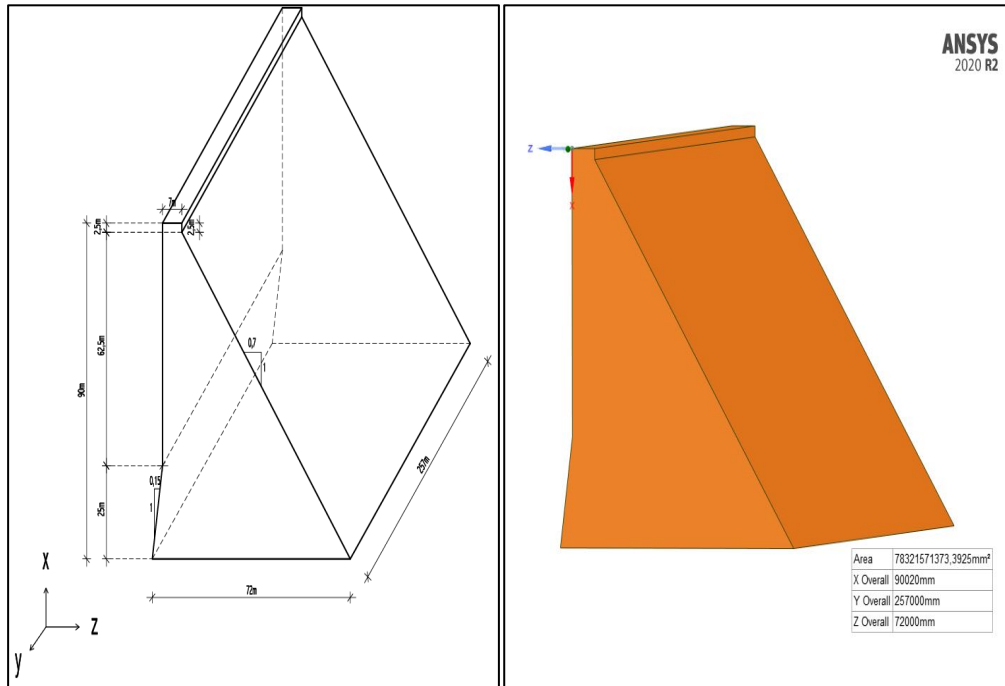


Fig. 5. Dimension of concrete gravity dam

Table 1. Material properties of concrete gravity dam

Material	Density $\gamma$ (kg/m <sup>3</sup> )	Young's Modulus E (Pa)	Poisson's Ratio $\nu$
Concrete	2300	3e+10	0,18

The behavior of the concrete gravity dam under water load effects was investigated. When defining the boundary conditions in the model, the material characteristics listed in Table 1. were specified first. After assigning the material, a mesh structure suitable for the dam's shape was applied. Then, since the dam is anchored to the slopes, a fixed support constraint was applied in these areas. Following the assignment of all boundary conditions, water load was defined for the model under both fully filled and half-filled condition. While examining the effect of water load, the effect of temperature was neglected and was not included in the boundary conditions.

Two different load cases were applied to the model equipped with the mesh structure: one for partially filled and one for fully filled conditions. Displacement and stress values were obtained for both the partially filled and fully filled conditions.

### 3. Numerical Analysis And Results

The stress and displacement values of the dam, modeled in three dimensions, were examined for the case of the reservoir being half full. Fig. 6. shows the water load for the half-filled condition of the dam. The water load is expected to create the highest stress at the base and the lowest stress at the dam surface. According to fluid mechanics principles, the stress due to the water load is represented in Fig. 6.

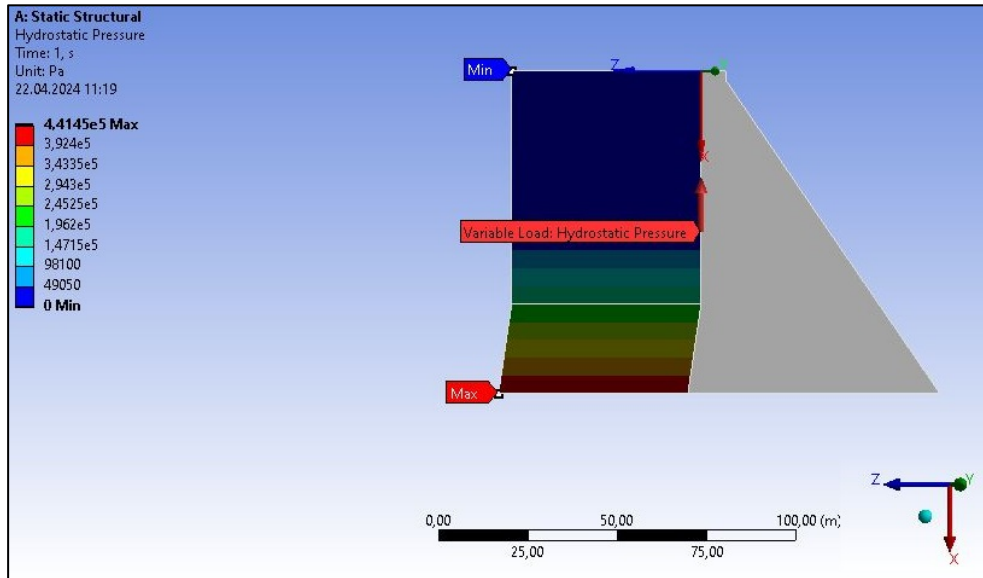


Fig. 6. Water load for the reservoir in the half-filled condition

For the half-filled condition of the dam, the stress and displacement values at the downstream section were examined. Fig. 7. shows the highest displacement value, which was found to be 0.00037326 meters. As expected, the highest displacement value is located in the middle part of the crest region.

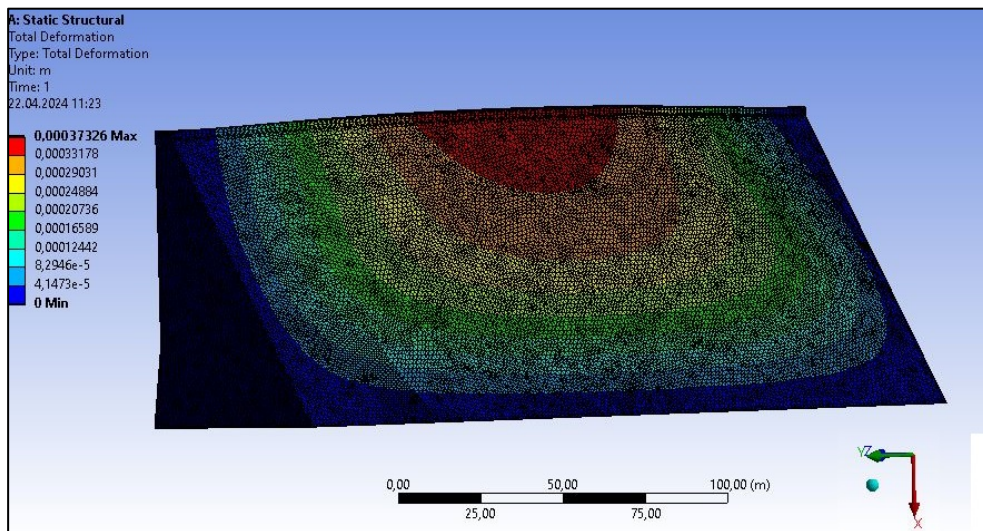


Fig. 7. Displacement at the downstream section of the dam in the half-filled condition

For the half-filled condition of the dam, a stress value was observed at the downstream section. As shown in Fig. 8, the highest stress value was found to be  $1.813 \times 10^6$  Pa. The lowest stress value observed was  $2.2585 \times 10^{-8}$  Pa.



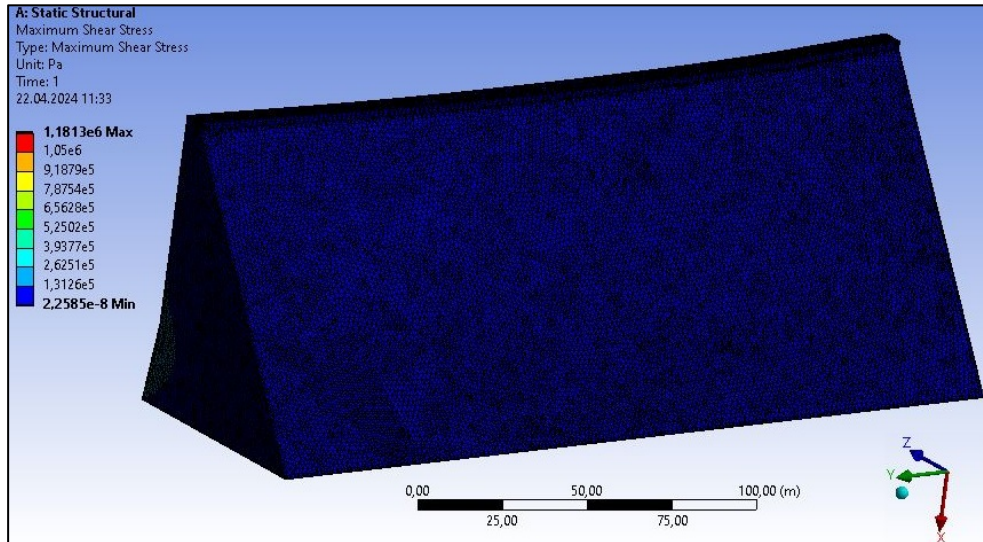


Fig. 8. Stress at the downstream section of the dam in the half-filled condition

Fig. 9. shows the water load for the dam in the fully filled condition. The water load is defined as lowest at the crest and highest at the base of the dam.

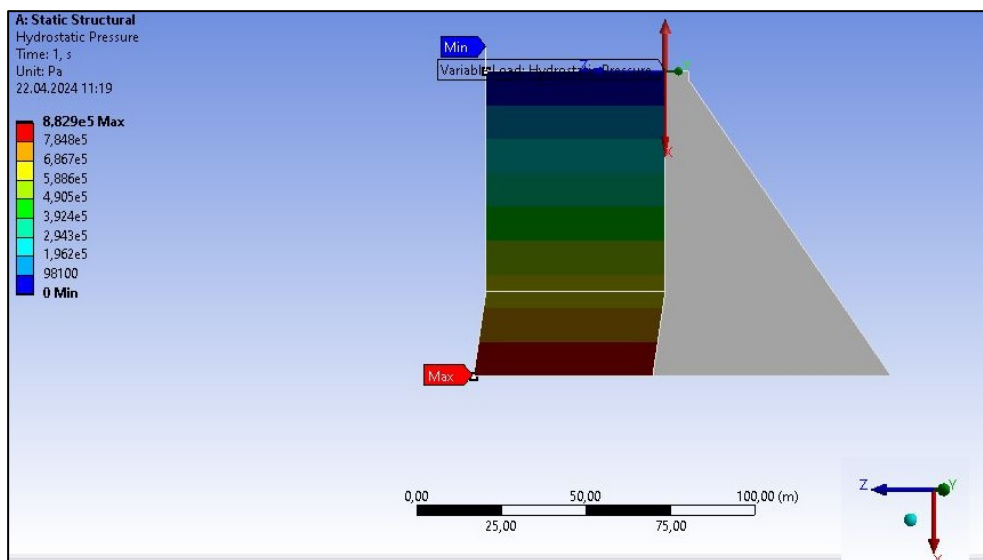


Fig. 9. Water load for the dam in the fully filled condition

Fig. 10. shows the displacement parameter at the downstream section of the dam in the fully filled condition. The displacement value was found to be 0.0075763 meters. As observed in the half-filled condition, the highest displacement value also occurs in the middle part of the crest region when the dam is fully filled.

For the case when the dam is completely full, a stress value has been generated at the downstream section. As shown in Fig. 11, the maximum stress value has been identified as  $3.5111 \times 10^6$  Pa. The minimum stress value is observed to be  $5.5282 \times 10^{-8}$  Pa.

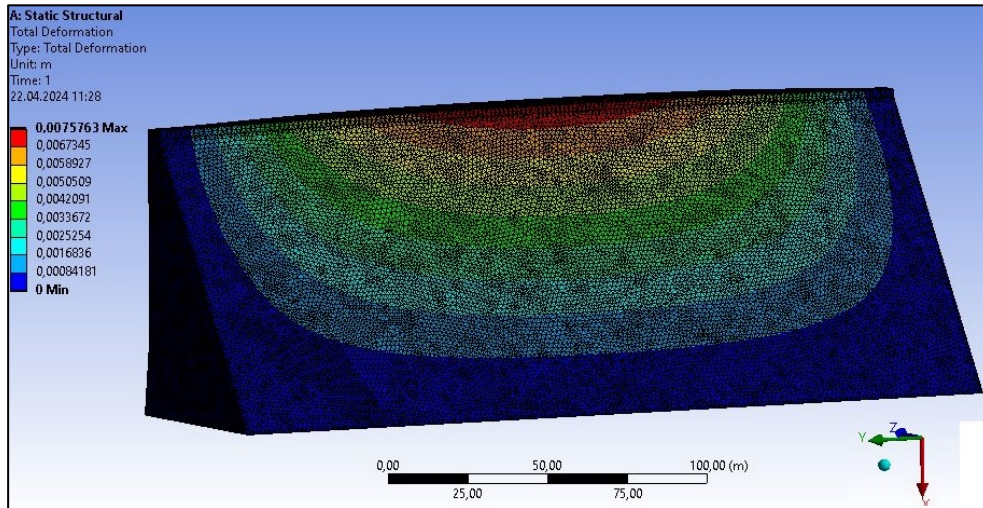


Fig. 10. Displacement at the downstream section of the dam in the fully filled condition

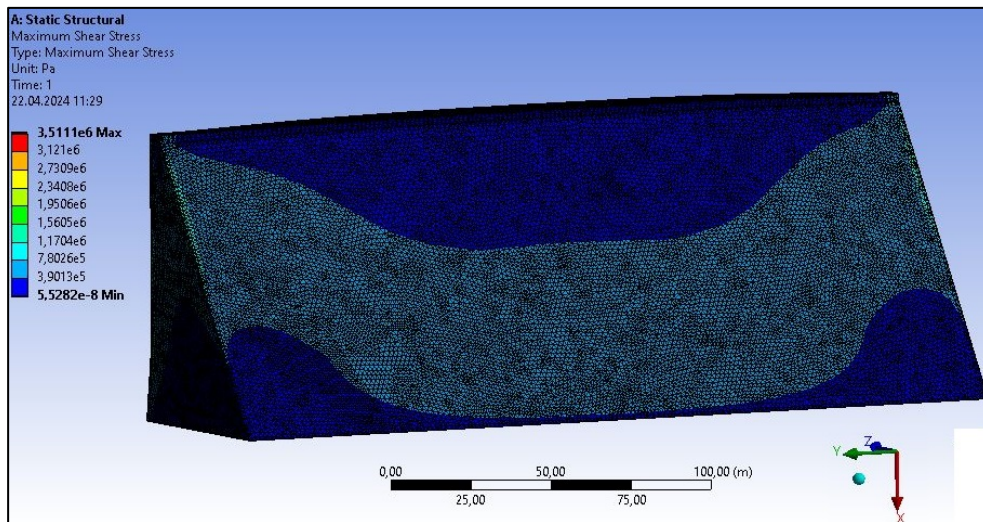


Fig. 11. Stress at the downstream section of the dam in the fully filled condition

#### 4. Conclusions

In this study, a concrete gravity dam has been examined, and the model of the dam was created in a 1:1 scale without any scaling. The lack of scaling is crucial for providing results that are close to reality regarding the stress and displacement values occurring in the dam. Analysis have been performed for the concrete gravity dam in both the half-filled and fully filled conditions. For these two cases, stress and displacement parameters have been determined and analyzed. During the analysis, attention was paid to equipping the structure with an appropriate mesh configuration, and the material properties were defined to be suitable for concrete gravity dams. Additionally, the water load was defined based on fluid mechanics principles and the height of the dam. The analyses were carried out with careful consideration of all these steps.

As a result of the analyses performed for the concrete gravity dam:

1. The displacement value obtained for the dam when fully filled is 0.0075763 m, whereas the displacement value when half-filled is 0.00037326 m. The larger displacement value in the fully filled condition is due to the effect of the water load.
2. The stress value obtained for the dam when fully filled is  $3.5111 \times 10^6$  PA, while for the half-filled condition it is  $1.813 \times 10^6$  PA. Similar to the displacement values, the higher stress value in the fully filled condition is the result of the water load effect.
3. In the study, the displacement and stress values at the downstream side of the concrete gravity dam were considered. The displacement and stress values at the upstream side of the dam will be examined in a subsequent study
4. The displacement value reaches its highest point in the middle of the crest area, as shown in the visuals. This is due to the fact that the concrete gravity dam is anchored to the slopes.
5. Large and costly structures such as concrete gravity dams can be modeled in detail using ANSYS, allowing critical parameters such as stress and displacement to be determined.
6. Unlike experimental studies conducted for dams, this approach is more cost-effective and provides results in a shorter amount of time.

## References

- [1] Ađıraliođlu, N., Baraj Planlama ve Tasarımı: cilt 1. Su Vakfı Yayınları, 2<sup>nd</sup> edition, 2007.
- [2] Erkek, C., Ađıraliođlu, N., Su Kaynakları Mühendisliđi. Beta Basım Yayıncılık Distribution, 6<sup>th</sup> Ed., 2010.
- [3] Ahemad, F., Awar, U., Hanumanthappa, M. S., Stability analysis of gravity dam with is-code method and ANSYS. SAMRIDDHI: A Journal of Physical Sciences, Engineering and Technology, 14(3), 435-441, 2022.
- [4] Saxena, S., Patel, M., Evaluating dynamic behaviour of a concrete dam using modal analysis. Materials Today: Proceedings, 93, 296-301, 2023.
- [5] Sharma, A., & Nallasivam, K., Static analysis of a concrete gravity dam using the finite element technique. Asian Journal of Civil Engineering, 24(8), 2939-2957, 2023.
- [6] Todorov, O. K., Petkova, S. S., Kisliakov, D. S., Formulation of parameter limit values for the operation of a large concrete gravity dam, 12th International Scientific Conference Civil Engineering Design and Construction (DCB 2022), New York, USA, 7-10 September, 2022.
- [7] Vipparthy, R., Static and free vibration analysis of gravity dam under the influence of hydrostatic pressure using ANSYS finite element models, 2022. <https://doi.org/10.21203/rs.3.rs-1823407/v1>.

- [8] Silveira, I. V. D., Pedroso, L. J., Marotta, G. S., Study of the influence of the foundation and the reservoir on the dynamic response in a concrete gravity dam profile. *Revista IBRACON de Estruturas e Materiais*, 14(4), e14403, 2021. <https://doi.org/10.1590/S1983-41952021000400003>.
- [9] Messaad, M., Bourezane, M., Latrache, M., Berrabah, A. T., Ouzendja, D., Three-dimensional seismic analysis of concrete gravity dams considering foundation flexibility. *Mechanics and Mechanical Engineering*, 25(1), 88-98, 2021.
- [10] Gao, S., Chen, Y., Xiang, J., Finite element analysis of stress in the gallery of gravity dam on inclined bedrock based on ANSYS model. 2021 International Conference on Water Conservancy, Energy and Environmental Engineering, Nanjing, China, 25-27 June 2021.
- [11] Wei, L., Wei, P., & Yang, Y., Static calculation and analysis of certain gravity dam. In *Journal of Physics: Conference Series*, 1550 (3), 032006, IOP Publishing, 2020. doi:10.1088/1742-6596/1550/3/032006.
- [12] Habib, A., Hourri, A. A., Habib, M., Elzokra, A., Yildirim, U., Structural performance and Finite Element modeling of roller compacted concrete dams: A review. *Latin American Journal of Solids and Structures*, 18, e376, 2021.
- [13] Taylan, D., Aydın, T., Analysis of dynamic behavior of Darideresi-II Dam by ANSYS. *Natural Hazards*, 90, 1223-1235, 2018.
- [14] Zhu, B., *The Finite Element Method: Fundamentals and Applications in Civil, Hydraulic, Mechanical and Aeronautical Engineering*, First Edition, 2018.
- [15] Ovalı, İ., *Ansys Workbench*. Kodlab Yayın Dağıtım Yazılım Ltd. Şti, 2018.