# Analysis of Stress and Displacement of Taham Earth Dam

#### Shayan Shahba and Fazlollah Soltani\*

Civil Engineering Department, Graduate University of Advanced Technology, Kerman, Iran; shahba860@yahoo.com, soltani.fazlollah@yahoo.com

### Abstract

This paper explores the stability and displacements of earth dam during construction and impounding using experimental instrument and simulated data in Taham earth dam, Iran. Data were collected using experiment as instrument and then these data were simulated applying FLAC, PLAXIS and GeoStudio software. The simulation results were compared to experimental instrument data as reference data, to find out the accuracy of each software output. The comparisons results indicated that the output of FLAC were similar to experimental instrument data, thus FLAC can be applied for evaluating stability and settlement analysis of earth dam in that it can provide reliable results.

Keywords: Taham dam, Exact Instruments, Modeling with Software,

# 1. Introduction

Dam failure disasters usually cause huge loss of lives and destructions of properties and environment. For example, the 1963 failure of Vajont dam in Italy caused 2600 deaths, the 1976 failure of Teton dam in America caused hundred deaths and economic loss about 1 billion dollars, and the 1993 failure of Gouhou dam in China caused 300 deaths. The statistical analysis of 534 dam failures from 43 countries before1976indicated that earth-rock dam failures accounted for the largest proportion of all failures and included 49% caused by overtopping, 28% seepage in dam body and 29% seepage in foundation<sup>1</sup>.Dam engineering is so specialized activity that consists of many scientific branches and with using much engineering judgments, it provides the connection and fairness between them.

Many dams are located at the upstream of villages or crowded cities that any defect in their safety will cause

to big hazards and irreparable casualties. With respect to severe dependence of society and industry on water, guaranty of exploiting possibility has great importance, and controlling their stability is an inevitable affair. So in this article the effort is to determine the produced displacements and stresses in a non homogeneous earth dam with numerical methods then compare the software results with exact instruments results.

The most important usage in stage of design is to evaluate the quantity of pore water pressure at the end of the construction. Geotechnical parameters in design stage are mainly concluded from experiments on depth resources. It is necessary to control the stability of dam during its construction<sup>6</sup>. One of the parameters is its settlements. Measuring the settlements in different operation stages and in special points in an earth dam to determine the required height and volume of materials is urgent. The parameters of the chosen behavioral model for fine materials are calibrated with the results of triaxial test.

\*Author for correspondence

### 2. Introduction of Studied Project

Taham dam is located in 15 kilometers north-east of Zanjan and at 8 kilometers downstream of Taham village. It has a clayey core with 123m height, 458m crest height, and 12m crest width and 538m heel width.



Figure 1. Plan of location and selected section.

### 3. Introduction of Software's

Numerical analysis has done with geotechnical software FLAC V4. FLAC is capable of water flow analysis and soil transformation with linear and nonlinear models for soil and rock. The numerical solution method in this software is based on explicit finite difference that has appropriate compatibility with nonlinear models and time-dependent analysis.

The second applied software which is used for analysis and comparison is GeoStudio. It is finite element software and it can do analysis such as stress-strain, water flow, permeance, slop stability, dynamic analysis and also conditions such as instant drop of water in reservoir of an earth dam. The section SIGMA/W of GeoStudiois used in this article. Creation of mesh area, applying boundary conditions and initial conditions, choosing the right model for materials, determining material parameters, assigning materials to different regions of model and defining fluid properties are main parts of these two softwares<sup>3</sup>.

The third software which its results are compared with the results of exact instruments is PLAXIS. PLAXIS is a finite element package that has been developed specifically for the analysis of deformation and stability in geotechnical engineering projects. With this software gradual cut and fill with different loading conditions and boundary conditions can be modeled by 6-node or 15-node elements. Mohr-Coulomb constitutive soil model, hardening-soil model, soft soil creep model and soft soil model can be used in this program<sup>4</sup>.

# 4. Failure Mode Applied in this Article

For attribution of materials to elements it is necessary to define the materials properties based on the chosen model. The selected model in this article is Mohr-Coulomb which

| ψ<br>(°)<br>Dilation<br>angle | n<br>Porosity | K<br>(cm/s)<br>Permeability<br>coefficient | Φ'<br>(°)<br>Angle of<br>internal<br>friction | C'<br>(MPa)<br>Cohesion | E<br>(MPa)<br>Modulus of<br>elasticity | v<br>Poisson's<br>ratio | γ<br>(kN/m³)<br>Unit<br>weight | Parameter<br>Region               |
|-------------------------------|---------------|--|---|-------------------------|--|-------------------------|--------------------------------|-----------------------------------|
| 2                             | 0.35          | 1.5*10-7                                   | 26.7  | 0.05                    | 50 ~ 19                                | 0.35                    | 17.5                           | Clayey core                       |
| 8                             | 0.25          | 2.66*10-6                                  | 40  | 0                       | 80 ~ 60                                | 0.3                     | 23.5                           | Shell                             |
| 6                             | 0.45          | 5*10-1                                     | 36  | 0                       | 65 ~ 45                                | 0.3                     | 21.1                           | Upstream filter                   |
| 6                             | 0.5           | 10-2                                       | 36  | 0                       | 60 ~ 40                                | 0.3                     | 20.3                           | Downstream filter                 |
| 5                             | 0.5           | 1.33*10-7                                  | 37  | 0                       | 50 ~ 70                                | 0.3                     | 22                             | Drainage                          |
| 8                             | 0.25          | 10-10                                      | 41  | 2                       | 900                                    | 0.2                     | 26                             | Rock foundation<br>(Andesitetuff) |

 Table 1.
 Geotechnical parameters in numerical analysis<sup>2</sup>

is an elastic-plastic model. In the mentioned model, failure envelope will result from shear failure criterion ( $f_s$ ) with respect to tensile failure criterion ( $f_t$ ). Surface failure is defined from functions  $f_s$  (line AB in Figure 2) and  $f_t$ (line BC in Figure 2) and will be described by formulas below. In this model surface behavior is independent from mean stress ( $\sigma_2$ ). After arriving the stress value to the failure limit ( $\sigma$ y), the stress conditions will not change and plastic deformations will begin.



Figure 2. Failure criterion in Mohr-coulomb model.

### 5. Investigation and Comparison between Test Results and FLAC Results for Clayey Core

To evaluate the results of numerical analysis, we should ensure the acceptable precision of model parameters. In this regard, triaxial tests have done on the materials of Taham earth dam and then compared with FLAC results. These tests are limited only to fine materials. The sample which is made by FLAC is both two-dimensional and axil symmetric.

To adapt the resulted numerical analysis curves with test results curves, it is tried to coincide the elastic part of numerical curve with elastic part of tests curve by changing the values of bulk modulus (B) and shear modulus (S). The cohesion value and friction values are obtained from test, so the failure stress ( $\sigma$ y) won't change during triaxial and error procedure, and only the elastic gradient and consequently the strain value in failure limit will change<sup>4</sup>. Laboratory curves and numerical adapted curves to them for fine materials are shown in Figure 3. In static analysis of dam, the analysis of effective stress with respect to pore water pressure is considered. Material's effective parameters which were obtained from CU tests on standard samples are used in numerical modeling. The calibrated parameters of clayey materials are presented in Table 2.

Table 2.Properties of clayey materials in numericalanalysis2

| φ     | C     | S        | B           | γ                    |                |
|-------|-------|----------|-------------|----------------------|----------------|
| (°)   | (MPa) | (MPa)    | (MPa)       | (KG/m <sup>3</sup> ) |                |
| 22.70 | 0.06  | 7 ~ 18.5 | 21.1 ~ 55.5 | 1750                 | Clayey<br>core |



**Figure 3.** Calibration of numerical stress-strain curves with the reults of laboratory model<sup>2</sup>.

To interfere the effects of confining pressure changes in coefficients rate, lateral pressure in different levels are extracted from exact instruments at first, then based on the calibration results, gradual changes in modulus of elasticity in dams height is determined. Hence the modulus of elasticity will be assigned to dam layers in approximate form (Table 1). The stress-strain parameters for coarse materials are mainly obtained from direct shear test.

# 6. Finite Element Netting and Location of Model's Section

To perform a two-dimensional analysis, the highest section of dam (section 10-10) has been selected (figure 1). This section is modeled, in Flac with 8205 elements (figure 4), in GeoStudio with 7382 elements (figure 5) and in plaxis with 6168 elements (figure 6).



Figure 4. Finite element model of dam in Flac<sup>2</sup>.



Figure 5. Finite elements model of dam in geostudio.



Figure 6. Finite element model of dam in plaxis.

### 7. Results of Numerical Analysis by FLAC, GeoStudio and PLAXIS

To analyze the dam with FLAC, at first the initial stresses are (before construction) are calculated. After assigning boundary conditions and initial conditions, the dam foundation has been analyzed and its deformations are equal to zero. Therefore, the model is ready for creation new conditions which are the soil layers. With developing any layer, the analysis will continue until achieving the equilibrium state. By defining the last layer, end of construction will be modeled.

After adding each layer, we have two calculation steps. Step one is mechanical calculation that the materials are in undrained situation, and because of that the pore water pressure will appear in the dam's core. In step two due to construction period of each layer; the embankment will have enough time for consolidation depreciation of pore water pressure. Several results of numerical analysis with FLAC by other authors are shown in Figure 7, Figure 8, Figure 9, Figure 10, Figure 11, Figure 12 and Figures 13<sup>2</sup>.



Figure 7. Contours of pore water pressure at first impounding.



**Figure 8.** Contours of pore water pressure at end of the construction.

Figure 7 shows Contours of pore water pressure at first impounding, Figure 8 shows Contours of pore water pressure at end of the construction.



**Figure 9.** Contours of total vertical stress at end of the construction.



**Figure 10**. Effective vertical stress changes at base level at the end of construction.



**Figure 11.** Effective total stress changes at base level at the end of construction.

Figure 10 shows effective vertical stress changes at base level at the end of construction, Figure 11 shows effective total stress changes at base level at the end of construction.



**Figure 12.** Contours of horizontal displacements at the end of construction.



Figure 13. Contours of settlement at the end of construction.

Figure 12 shows contours of horizontal displacements at the end of construction, Figure 13 shows Contours of settlement at the end of construction.

Maximum pore water pressure at the end of construction is 400 kPa and will occur at 43 meters above the bottom of this dam (Figure 8, in blue zone). In this place the proportion of pore water pressure to surcharge pressure is 0.26. Center of clayey core because of low permeability and the effect of impounding have 100 kPa increases in pore water pressure, but in upstream of core the pore water pressure has reached to 900 kPa (Figure 7). Low quantity of pore water pressure at the end of construction has caused insignificant difference between total stress and effective stress (Figure 10 and Figure 11).

It is worth mentioning that there is a decrease in boundaries of core and filters, and a decrease of stress in the boundaries of downstream filter and drainage can be noticed. Maximum calculated settlement by FLAC is 90 cm and occurs at 70 m above the bottom of dam (Figure 13). Horizontal displacement at upstream is 10 cm and at downstream is 15 cm (Figure 12). Total settlement after impounding is 120cm which is 30 cm increased than the end of construction stage. In upstream shell due to a decrease in effective stress of materials, the displacements at the end of construction have increased up to 15 cm<sup>2</sup>.

GeoStudio will calculate just instantaneous settlement. Based on this software's results maximum instant settlement will occur one day after impounding and its value is 1.7 m, and the value of horizontal displacement is 17 cm. The maximum measured stress is 3500 KPa and occurs in foundation under dam's core. In Figure 14, Figure 15, Figure 16 and Figure 17 several analyzed results of GeoStudio are illustrated.

The settlement and stresses that applied to dam before and after impounding are also calculated in PLAXIS (the dam is modeled with 15-node elements). Maximum settlement is 1.15 m, horizontal displacement is 13 cm and maximum stress is 2600 N/m2. In Figure 18, Figure 19, Figure 20 and Figure 21 the results of dam analysis by PLAXIS are illustrated.



Figure 14. Maximum stress contours (KPa).



Figure 15. Dam's displacement after impounding.



Figure 16. Contours of horizontal displacement (m).



Figure 17. Contours of settlement (m).



Figure 18. Contours of horizontal displacement by plaxis.



Figure 19. Contours of settlement by plaxis.



Figure 20. Contours of total dam's stresses after impounding.



Figure 21. Contours of total dam's stresses before impounding.

### 8. Comparison between the Results of Numerical Analysis with the Results of Exact Instruments

8.1 Comparison between the Results of FLAC Analysis with the Results of Exact Instruments



**Figure 22.** Comparison between the results of FLAC analysis with the results of exact instruments.



**Figure 23.** Comparison between the results of FLAC analysis with the results of exact instruments.



**Figures 24.** Total stress contours of end of construction and at first impounding.



**Figures 25.** Total stress contours of end of construction and at first impounding.

Figure 22 shows comparison between the results of FLAC analysis with the results of exact instruments for pore water pressure at the end of the construction and Figure 23 shows the same comparison at the first impounding. Figures 24 and Figure 25 are respectively showing the total stress contours of end of construction and at first impounding.

### 8.2 Comparison between the Results of GeoStudio with the Results of Exact Instruments for Total Stress



**Figure 26.** Comparison between the results of GeoStudio with the results of exact instruments for total stress.



**Figure 27.** Comparison between the results of GeoStudio with the results of exact instruments for total stress.

The comparison between results of GeoStudio analysis with exact instruments for total stress before impounding and after impounding are shown in Figures 26 and Figures 27 respectively. It is observed that the value of stress will be variable after impounding.

### 8.3 Comparison between the Results of PLAXIS with the Results of Exact Instruments for Total Stress



**Figure 28.** Comparison between the results of PLAXIS with the results of exact instruments for total stress.



**Figure 29.** Comparison between the results of PLAXIS with the results of exact instruments for total stress.

Comparison between results of Plaxis analysis with exact instruments for total stress before impounding and after impounding are shown in Figure 28 and Figure 29 respectively. Instrumental measurement in each part shows that the value of stress will decrease after impounding, but the value of numerical measurement is variable. Table 3 shows the comparison of maximum vertical and horizontal displacements. Table 4 shows the comparison of maximum measured stress.

**Table 3.** Comparison of maximum vertical and horizontaldisplacements

| FLAC | GeoStudio | PLAXIS |                             |
|------|-----------|--------|-----------------------------|
| 1.2  | 1.7       | 1.15   | Vertical displacement (m)   |
| 0.1  | 0.17      | 0.13   | Horizontal displacement (m) |

| Table 4. | Comparison | of maximum | measured | stress |
|----------|------------|------------|----------|--------|
|----------|------------|------------|----------|--------|

| FLAC | GeoStudio | PLAXIS |              |
|------|-----------|--------|--------------|
| 3000 | 3500      | 2580   | Stress (KPa) |

### 9. Instantaneous Settlement Curves (from Crest to Bottom of Foundation)



**Figure 30.** Settlement curves of FLAC and exact instruments after construction at x = 174m, 214m & 275m<sup>2</sup>.



**Figure 31.** Settlement curves in Plaxis after impounding at x = 174m, 214m & 275m<sup>2</sup>.



**Figure 32.** Settlement curves in GeoStudio after impounding at x = 174m, 214m & 275m.

In the above Figure 30, Figure 31 and Figure 32 we see the comparison between settlement curves resulted from three software and exact instrumental tools at three sections of the earth dam. It is conducted that FLAC gives us more accurate settlement curves than the other two software.

### 10. Conclusion

Performed numerical analysis with calibrated materials, has given nearly exact results of stress and displacements in body of dam. The stress and pour water pressure values are in good coincidence with the installed exact instruments except in upper level.

Basically the hardening-soil model due to including more soil parameters, can analyze these materials better than the other models. This model has a higher yielding level because of the existing plastic yielding strains (unlike the Mohr-Coulomb model with constant yielding level). And also dilation of the soil in shear is considered in this model. This subject gradually will become important in lower layers as the embankment develops and consolidation occurs. On the other words the elasto-plastic model, will model the soil behavior in presence of normal stresses. But the hardening model will consider the elasto-plastic behavior for soil from first steps. Non-compliance of charts relates to the resistance of parameters of soil that basically their quantity will change as time changes and load increases (these parameters cannot be inserted in numerical model).

Since soil material properties, most of which do not have a physical basis, have a great influence on dam-break process, more work should be done to identify the variations of dam material properties and corresponding effect in dam-break process. Besides, scale effect in dam-break experiments should be considered and researches on similarity criteria of dam-break process should be enhanced<sup>5</sup>.

At the end, it can be noted that to achieve the best numerical model in earth dams it is better to pay attention to the following recommendation:

- Considering the heterogeneity materials of the core and shell in the dam's body while creating a numerical model of layered materials in terms of properties and mechanical properties at different levels according to the slope of layers, depending on the circumstances of each project.
- 2. Using the 3-D analysis and comparing its results of with 2-D analysis to understand the behavior of dam better than before.
- 3. Studying the effects of earthquake and dynamic loads one the core's pore water pressure is one of the important subjects in investigation of dam's stability.
- 4. Considering the settlement and stresses exerted on the foundation of the dam before construction.

# 11. References

- 1. ICOLD Bulletin 111, Dam-break flood analysis; 1998.
- 2. Karimi F, Soroosh A. Numerical analysis of static behavior of Taham dam. Third National Congress of Civil Engineering, Iran; 2007.
- 3. Krahn J. GeoStudio SIGMA\W. Workshop presentation; 2007.
- 4. PLAXIS V8.5 Tutorial Manual; 2007.

- 5. You L, Li C, Min X, Xiaolei T. Review of dam-break research of earth-rock dam combining with dam safety management. Procedia Engineering; 2012. p. 382–8.
- 6. Toori RA, Pourbakhshian S, Pouraminian M. Admixing PET for the Safety of Earth Dams. Indian Journal of Science and Technology. 2014 December; 7 (12):1950–5.