International Journal of Science and Research (IJSR)
ISSN (Online): 2319-7064
Index Copernicus Value (2015): 78.96 | Impact Factor (2015): 6.391

The Methodology of Representing and Analyzing the Three-Dimensional Tunnel Case

Mohannad H. Al-Sherrawi[1], Adnan F. Ali[2], Mohammed Najm[3]

[1, 2, 3]University of Baghdad, College of Engineering, Civil Eng. Department, Baghdad, Iraq

Abstract: ***This study is conducted to demonstrate the ability of the 3-D representation of the underground reinforced concrete tunnels with the action of the method of construction. Three distinct objectives are considered; the methodology of idealization based on finite element representation, the nonlinear behavior of the system, and the effect of using interface element to represent the soil structure interaction. This study contains models of the materials, some notes about the used elements, and finally the simulation of two case studies with some results. Some of the conclusions that extract from this study are, negligible differences in the results were appearing from analyzing different F.E. meshes of the tunnel, and more conservative simulation for the model can be made by using interface element to simulate the soil structure interaction because it increases the response (displacement and stresses) in tunnel lining.***

Keywords: tunnel, soil, reinforced concrete, nonlinear, interface element.

1. Introduction

Tunnels are structures of a relatively uniform cross-section and of a significant length can be used as: Transportation routes for rapid transit (like metro lines), railroad and vehicle traffic, convey both fresh and wastewater, and conduit water for hydroelectric power generation, and may be used for other purposes. Tunnels may be susceptible to different types of loads, thus the conventional representation of 2-D plane strain condition to represent the tunnel case is not admissible for the analysis. For this reason this study is made up to show the methodology for the 3-D representation of the tunnel case to the analysis. By using the facilities of ANSYS package [1] this study had been implemented.

2. Material Modeling

The non-linear behavior of the materials is considered in the present work, because the plastic deformations of the structure as well as of the surrounding media are expected. Nonlinearity in structures can be classified mainly as, geometric nonlinearity (associated with changes in configuration, as in large deflection of a slender beam), and material nonlinearity (associated with changes in material properties, as in plasticity). The latter is adopted in the present work by using, Willam and Warnik five-parameter model to simulate concrete material, Drucker-Prager two-parameter model to simulate soil material and Von Mises with bilinear isotropic hardening one parameter model to simulate the behavior of steel material of reinforcement.

2.1 Willam and Warnik Model

This model is used to simulate concrete material, which predicts the failure of brittle materials. Both cracking and crushing failure modes are accounted for [1] and [2], the three-dimensional failure surface in principal stress space can be seen in Figure 1.

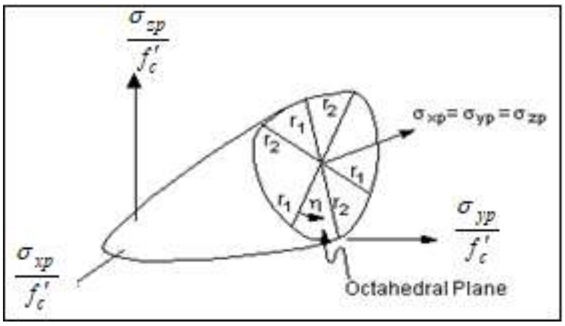

Figure 1: 3-D failure surface in principal stress space [1]

2.2 Drucker-Prager Model

The Drucker-Prager yield criterion with an associated flow rule is adopted in this study to simulate the nonlinear behavior of soil materials. The yield surface does not change with progressive yielding, hence there is no hardening rule and the material is elastic- perfectly plastic [1] and [2], Figure 2 shows the stress strain behavior of Druker-Prager yield criteria.

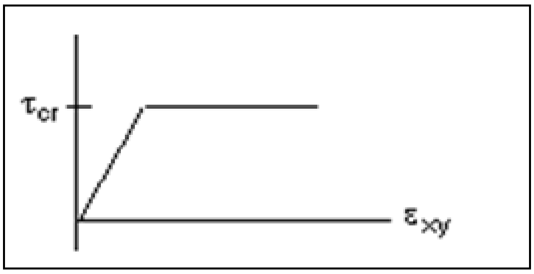

Figure 2: Stress strain behavior of Druker-Prager yield criteria [1]

2.3 Von-Mises with Bilinear Isotropic Hardening Model

The bilinear isotropic (work) hardening with the Von Mises yield criterion and an associated flow rule adopted in the present work to represent the nonlinear behavior of steel material of reinforcement.

3. Finite Element Formulation

The finite element method had been adopted to simulate the case study by using 8 nodes brick element for both concrete lining and the soil media, and an interface element used to represent the soil structure interaction. For abbreviation consideration the formulation of the elements that used to represent the case study will not encountered in this study, thus used elements with some general notes of them are listed as follows:

3.1 Three-dimensional reinforced concrete element (SOLID65)

The brick element denoted as SOLID65 in ANSYS is used for the three-dimensional modeling of concrete with or without reinforcing bars (rebars). The element is capable of cracking in tension and crushing in compression by using Willam and Warnik model. The element is defined by eight nodes having three degrees of freedom per node, translations in the nodal x, y, and z directions. Up to three different rebar specifications may be included in this element. The most important aspect of this element is the treatment of nonlinear material properties. The element is capable of cracking (in three orthogonal directions), crushing, and plastic deformation. The rebars are capable of tension and compression, but not shear. They are also capable of plastic deformation. The element is defined by eight nodes and the isotropic material properties. The element has one solid material and up to three rebars materials. Rule of full integration points (i.e. 2×2×2 integration points) is used in this study.

3.2 Three-dimensional (SOLID45) element

The brick element of eight nodes and three translational degrees of freedom per node, denoted as SOLID45 in ANSYS is used for the three-dimensional modeling of soil media with the Drucker-Prager material modeling.

3.3 Interface element

This element, which denoted as Contac 52 in ANSYS can represent any two surfaces which may maintain or break physical contact and may slide relative to each other. The element is capable of supporting only compression in the direction normal to the surfaces and shear (Coulomb friction) in the tangential direction. The element has three degrees of freedom at each node: translations in the nodal x, y, and z directions. The element is defined by two nodes, two stiffnesses (k_n and k_s), an initial gap can be added. The orientation of the interface is defined by the node locations. The interface is assumed to be perpendicular to the I-J line or to the specified gap direction. The element coordinate system has its origin at node I and the x-axis is directed toward node J. The interface is parallel to the element y-z plane. This element may have one of three conditions: closed and stuck, closed and sliding, or open [1].

3.4 Birth and death of element technique

This technique is used in the present work to represent the excavation process (construction procedure). To achieve the element death effect, the ANSYS program does not actually remove (killed) elements. Instead, it deactivates them by multiplying their stiffness by a severe reduction factor. This factor is set to 1.0E-6 in the present work. Element loads associated with deactivated elements are zeroed out of the load vector. Similarly, mass, damping, and other such effects are set to zero for deactivated elements. The mass and energy of deactivated elements are not included in the summations over the model. An element's strain is also set to zero as soon as that element is killed. In a similar manner, when elements are born, they are not actually added to the model they are simply reactivated [3].

4. Applications

4.1 Bangkok Sewer Tunnel

The Bangkok Sewer Tunnel is a part of the water transmission project undertaken by Bangkok Metropolitan Water Work Authority. A simplified soil profile with a finite element mesh is shown in Figure 3. 20 MPa as the modulus of elasticity (E) and 1700 kg/m^3 as a bulk unit weight are used for all soil profile and 2.66 m diameter of tunnel [4]. A plane strain condition is assumed in this problem. The solution is done by using two load steps, the first is applying the gravity load for all soil media to simulate the prestressed environment and the second is killing (deactivating) the tunnel elements to simulate the excavation process. The results and the comparison with field observations are shown in Figures 4 and 5, which can be considered to give acceptable prediction.

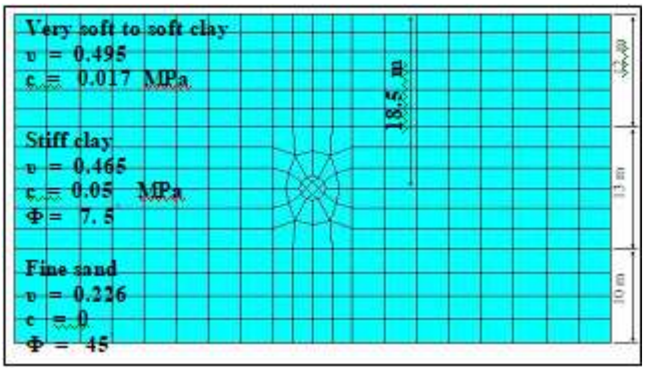

Figure 3: Finite element mesh of sewer tunnel with simplified soil profile

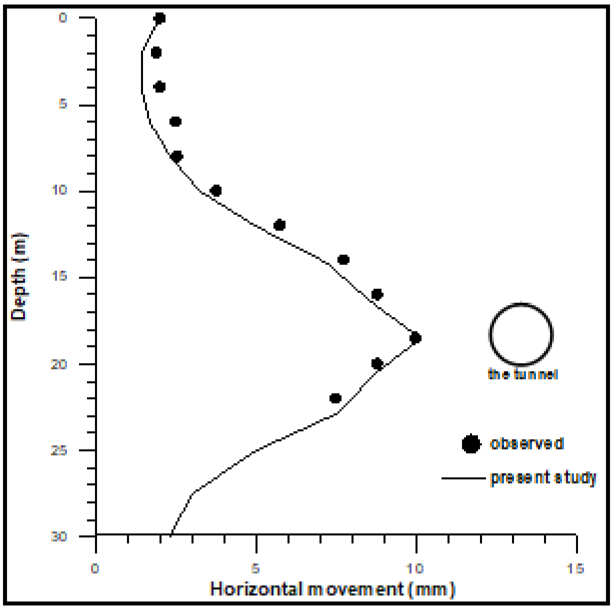

Figure 4: Observed and resulted horizontal displacement, 4 m away from centerline

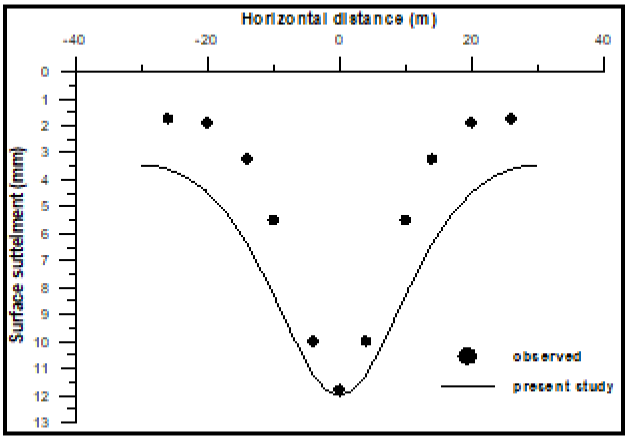

Figure 5: Observed and resulted surface settlement

4.2 Baghdad metro line

Because of the lack of information about the tunnel projects even though in the researches concerned with specific tunnel projects, the proposed Baghdad metro line can be considered as the case study in spite of some information were assumed, which they are not available.

The proposed Baghdad metro line consists of two routes of 32 km long and 36 stations designed as fortified shelters, as a first stage. The tunnels are circular in cross section with a 5.9 m outer diameter and a 0.45 m of a reinforced concrete lining thickness. The unavailable information are the reinforcement details and the method of construction details. For this reason it can be said that the case study is about to be the proposed Baghdad metro line. The reinforcement distribution is assumed to be 16 mm in diameter at 200 mm c/c for both directions (longitudinal and hoop) in two layers with 75 mm outside and inside covers. The method of construction is assumed to be as shield-driven method which proposed that 100% of the primary stresses are assumed to act on the lining [5] and [6]. This assumption is based on some information derived from The Populace Company of Executing the Transportation and Communication Projects.

A typical geological section for a specified position with tunnel axis depth of 18 m [7] is shown in Figure 6. This Figure also illustrates soil properties and cross section in the finite element mesh. Concrete and steel material [8] are listed in Table 1. Three different meshes of the tunnel along the length are shown in Figure 7, and a 3-dimensional view for the mesh of the whole case that is corresponding to mesh type c in Figure 7 is shown in Figure 8. The applied boundary conditions are displacement restrictions in x direction for all nodes at the sides of the model that are shown in Figure 8, and in y direction for the nodes at the bottom, and in z direction for nodes at the two faces.

Table 1: Concrete and steel properties [8]

Concrete	*Steel*
E_c = 20000 MPa	E_s = 200000 MPa
υ = 0.15	υ = 0.3
f'_c = 30 MPa	f_y = 400 MPa
f_t = 3 MPa	E_t = 4000 MPa
γ = 2400 kg/m^3	γ = 7800 kg/m^3
β_t = 0.5	
β_c = 0.6	

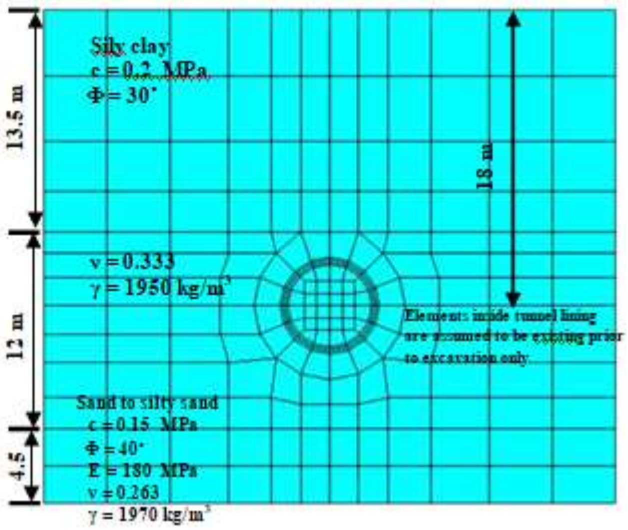

Figure 6: Geological profile with soil materials properties and F.E. mesh

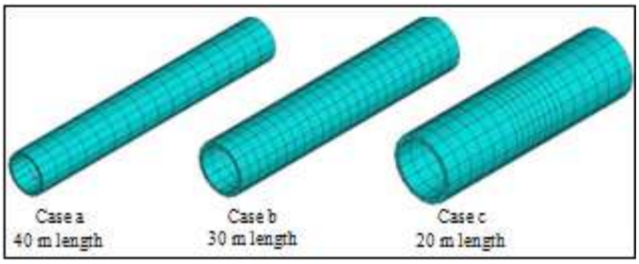

Figure 7: Different F.E. meshes of the tunnel

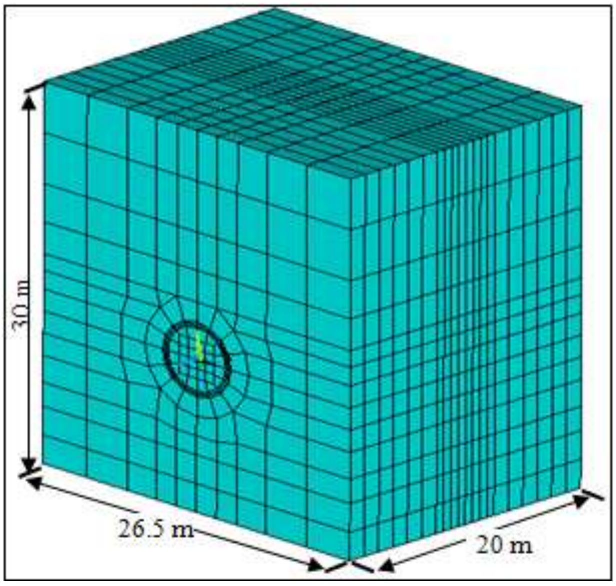

Figure 8: 3-D view of the F.E mesh for the whole model

The analysis consists of important load steps, and these are:

1) Performance of the prestressed environment. This is made by applying the gravity load of the soil media (and of the lining in the following step). This gravity load affects essentially the stresses in the soil media and as a result, the stresses of the structure (tunnel lining).
2) Performance of the method of construction, which gives significant effects on the amounts and the distribution of the stresses and the displacements of the lining that differ from method to another. This load step is employed by:
   a) Killing (deactivating) the elements of the core of the tunnel.
   b) Activating tunnel-lining elements as concrete lining elements by activating the concrete material (Willam and Warnik model with plasticity, cracking, and crushing abilities).
   c) Activating the steel material (bilinear isotropic hardening model) to achieve the steel reinforcement of the lining.

Cracking pattern for the upper half of the tunnel under the gravity load is shown in Figure 9, the cracks appear in this figure are in the inner surface only. Some notes can be observed when the static analysis had been implemented, these are:

a) Negligible differences in the results were appearing from analyzing different F.E. meshes of the tunnel for the two static load steps. These meshes are corresponding to cases a, b and c of Figure 7. Table 2 outlined some of these results. The similarity in results came from (the tunnel case can be considered to be a very good condition for the plane strain modeling in static analysis).
b) To preserve the tunnel cross section circle after applying the gravity load and before performing the second load step, some distortion for the cross section of the tunnel elements before applying the gravity load may be an adequate solution. To reveal this matter, Figure 10a shows the tunnel cross section with nodes movements inside the arrows in magnitudes appear above these arrows. Movement magnitudes are contrived from trial and error. Figure 10b shows the vertical displacements after applying gravity load. It is evident that they are negligible relative displacements.
c) A small amount of rigid body movements of tunnel elements after applying the gravity load can be acceptable.

Table 2: Some static analysis results of different cases of meshing

Displacement (mm)	*Step*	*Case a*	*Case b*	*Case c*
Vertical displacement at ground surface	*1*	-52.27	-52.27	-52.27
	2	-50.49	-50.58	-49.78
Vertical displacement at crown of the tunnel	*1*	-35.11	-35.12	-35.12
	2	-30.98	-30.86	-30.68
Vertical displacement at invert of tunnel	*1*	-23.87	-23.87	-23.87
	2	-23.86	-23.91	-23.82
Horizontal displacement at right side of tunnel	1	0	0	0
	2	3.01	2.97	2.92

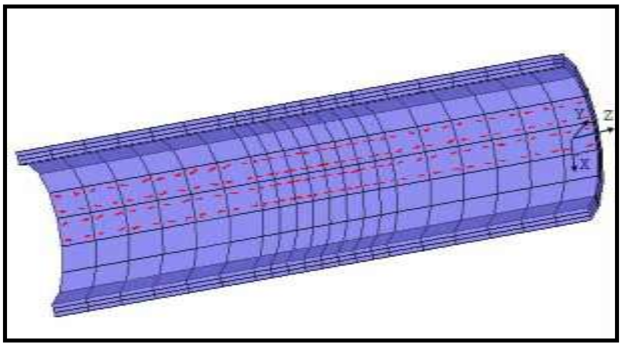

Figure 9: Cracking pattern under gravity load only

4.3 Effect of using the interface element

The above analysis was performed without using an interface element to represent the soil structure interaction. There are two main reasons to justify the using of the interface element in the model of the present work, the first was to assure for any extent the previous analysis could be admissible (because of the using of the interface element in this problem can give more realistic simulation), and the second was to show the ability to use the interface element from the researcher by using ANSYS and the method to implement this request.

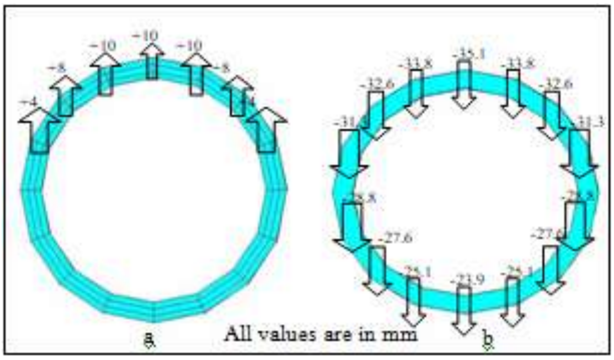

Figure 10: Tunnel cross section before and after applying the gravity load a) before applying gravity load b) after applying gravity load

Normal stiffness (k_n) and sticking stiffness (k_s) are calculated for the second load step as below:

$k_n \approx f\,E\,h$

where:

f = Factor that controls contact compatibility. This factor will usually be between 0.01 and 100 using f = 1 is often a good starting value.

E = Young's Modulus (if the contact between two different materials, the smaller value of E is used).

h = Characteristic contact "length." In 3-D configurations, h should be equal to a typical contact target length (that is, the square root of the target area) or a typical element size.

As a guideline, the sticking stiffness k_s should be 1, 2, or 3 orders of magnitude less than the normal stiffness, thus $k_s = k_n/100$ may considered a good estimation.

k_n was assumed to be of a very high value for the first load step to verify the connectivity and continuity of the soil media before the erection of the tunnel lining.

In the normal direction, when the normal force (F_n) is negative, the interface remains in contact and responds as a linear spring. As the normal force becomes positive, contact is broken and no force is transmitted [1].

In the tangential direction, for $F_n < 0$ and the absolute value of the tangential force (F_s) less than $\mu|F_n|$, the interface sticks and responds as a linear spring. For $F_n < 0$ and $F_s = \mu|F_n|$, sliding occurs. If contact is broken, $F_s = 0$. Noting that μ is the coefficient of friction, which is estimated as follow [9]:

$$\mu = \tan\delta \text{ and } \delta \approx \frac{3}{4}\varphi$$

where:

δ : the angle of wall friction.

ϕ : the angle of internal friction of the surrounding soil

The previous cases had been re-analyzed by using the interface elements, which are located between the concrete and the soil elements. The model that re-analyzed is of case c of Figure 7. The comparison of the static analysis result for the second load step can be seen in Figure 11, which illustrates the radial (D_r) and tangential (D_θ) displacement for nodes located at the outer periphery of the concrete lining and in the middle length of the model. Noting that +ev sign for the radial displacement refer to outward movement and vice versa, while the right hand rule can specify the tangential direction. This figure shows that there are no significant differences in displacements (less than 10 % for both directions) for the static analysis when the interface element is used or not. More conservative simulation for the model can be made by using interface element to simulate the soil structure interaction because it increases the response (displacement and stresses) in tunnel lining [3].

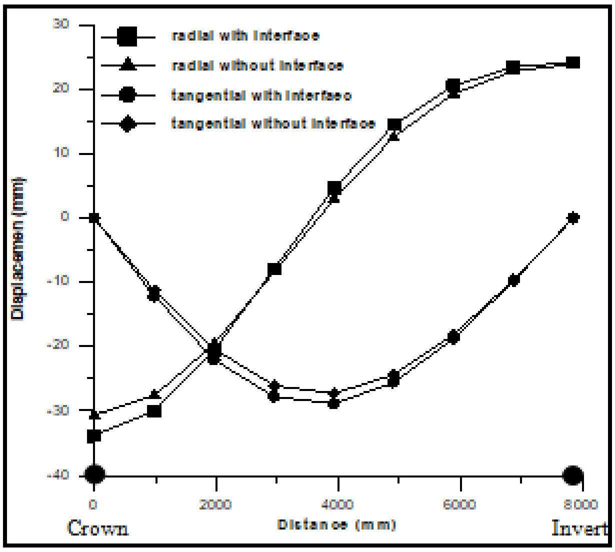

Figure 11: Radial and tangential displacement for the outer surface of the tunnel lining with and without interface element

5. Conclusions

The study aimed to investigate the 3-D F.A. modeling of underground reinforced concrete tunnels with the action of the method of construction. Negligible differences in the results were noticed when analyzing different F.E. meshes of the tunnel. By using interface elements, to simulate the soil structure interaction, more conservative simulation for the model can be made due to the increases in the response (displacement and stresses) in tunnel lining.

References

[1] ANSYS Manual.

[2] W.F. Chen Plasticity in Reinforced Concrete, Mc Graw-Hill, 1982.

[3] M. Najm, Dynamic Analysis of R.C. Tunnels Subjected to Internal Explosions, Thesis submitted to Collage of Engineering University of Baghdad, January 2004.

[4] N. Loganathan and H.G. Poulos ,"Analytical Prediction for Tunneling-Induced Ground Movement in Clays," J. of Geotechnique and Geoenviroment Engineering, 124 (9), pp. 846-856, September 1998.

[5] H. Duddeck, "The ITA's Approach to Structural Design of Tunnels," Tunneling and Underground Space Technology, 3 (3), pp. 235-249, 1988.

[6] H. Duddeck and J. Erdmonn, "On Structural Design Models for Tunnels in Soft Soil," Underground Space, 9, pp. 246-259, 1985.

[7] British Metro Consultants Group, "Baghdad Metro Project Stage 1," Report 6, Portion 2, Part B, February 1983.

[8] D. Consult, "Thickness Deign for Station and Tunnel," Final report, Baghdad Metro Project-Stage 1, February 1983.

[9] M.J. Tomlinson, Foundation Design and Construction, Pitman Publishing London, third edition, 1975.

Author Profile

Mohannad H. Al-Sherrawi received his B.Sc. in civil engineering in 1989, M.Sc. in structural engineering in 1996 and Ph.D in structural engineering in 2001 from University of Baghdad, Baghdad, Iraq. He has 28 years of experience in the design and construction field as structural engineer. Currently, he is an instructor in University of Baghdad.

Adnan F. Ali received his B.Sc. in civil engineering in 1976 and M.Sc. in structural engineering from University of Baghdad, Baghdad, Iraq and Ph.D in structural engineering in 1993 from Indian institute of technology, Bombay. Now he is a professor in structural engineering at University of Baghdad / College of Engineering. Member of the ASCE. Senior consultant and structures designer in Nameerva / Tuskish Company for construction.

Mohammed Najm received his B.Sc. in civil engineering in 2000 and M.Sc. in structural engineering in 2004 from University of Baghdad, Baghdad, Iraq. He worked in several companies in Iraq, UAE and Canada. Now he is P.Eng, PMP, CSSBB.