Proceedings of International Structural Engineering and Construction
Holistic Overview of Structural Design and Construction
Edited by Vacanas, Y., Danezis, C., Singh, A., and Yazdani, S.

ISSN: 2644-108X

FINITE ELEMENT MODELING OF MECHANICALLY STABILIZED EARTH WALLS BUILT WITH WELDED WIRE WALL PANELS

SAMAN HEDJAZI and DEJUAN SOLAN

Dept of Civil Engineering and Construction, Georgia Southern University, Statesboro, USA

In this paper, a finite element modeling of Mechanically Stabilized Earth (MSE) walls using welded wire wall panels was performed. The implementation of finite element modeling and analysis proved to be quite efficient in simulating the three-dimensional behavior of wall panels that are a part of MSE walls. The comprehensive finite element model included defined concrete and steel material properties in order to present both the realistic behaviors of each component in the model as well as better facilitating and increasing the accuracy of the simulation of numerous finite element analysis (FEA) cases. FEA was employed to simulate welded wire wall panels under the applied loads and to consider varying parameters of the model. The standard finite element tool (Abaqus) was used to conduct the analysis. Demonstrated behaviors and the model's performance were observed throughout the implementation of soil pressure and pullout loads on an anchorage system. The captured results were used to prove that the possibility of implementation of 3D panels as MSE wall facings, and to determine the mode of failure of panels, and to establish a sufficient anchorage system.

Keywords: Abaqus, MSE walls, FEA of walls, Soil pressure.

1 INTRODUCTION

A formal definition of a retaining wall would be a wall that serves the function of retaining soil by resisting the lateral forces generated and any surcharge loads associated with that fill (Walters *et al.* 2016). "Mechanically Stabilized Earth (MSE) walls are earth retaining structures that are constructed by placing alternating layers of reinforcement and compacted soil behind a facing element to form a composite material which acts integrally to restrain lateral forces" (Alzamora and Anderson 2009). For the use of MSE walls one must consider numerous factors such as geologic and topographic conditions, environmental conditions, size and nature of the structure, aesthetics, durability considerations, performance criteria, availability of materials, experience with a particular system or application, and cost (Berg *et al.* 2009). The facings of MSE walls can be considerably costly which is incurred by its weight, which increases not only material costs but also those for time, labor and equipment.

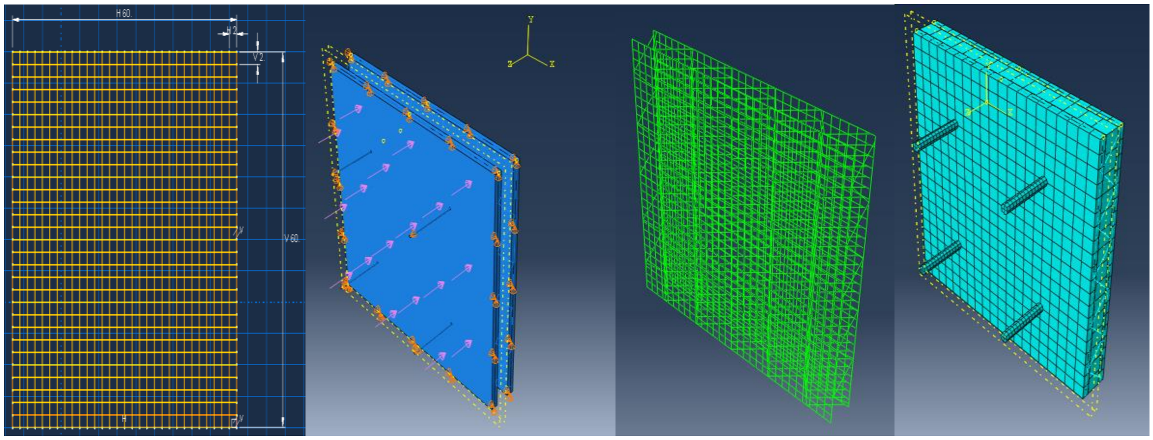
The welded wire panels also referred to as 3D panels are prefabricated panels that consist of a super-insulated core of rigid expanded polystyrene (EPS) sandwiched between two sheets of steel welded wire fabric mesh. Essentially galvanized steel truss wires pierce the polystyrene core at various offset angles in order to improve performance and are then welded to the sheets of steel welded wire fabric mesh (WWFM) on the outer layers of the panel. In its usage 3D panels are placed into its intended position where layers of concrete or any other relevant material can be applied. Additionally, the implementation of rebar is applicable in these instances. Therefore, due to the information stated through research in this paper, the implementation of 3D wire panels as a viable alternative for MSE wall facing was investigated. Benefits for using 3D welded wire panels include the ability for rapid construction as well as reducing transportation costs, installation costs, and stability problems. The panels can be transported easily and quickly due to their shape and size. For layering of the concrete or shotcrete a layer needs to be applied to a thickness that encases the welded-wire fabric and an additional layer to achieve the final thickness is required (Building Materials and Technology Promotion Council Ministry of Housing and Urban Poverty Alleviation 2015). Note that the curing and other practices related to concrete work should be in accordance to general concrete practices. Strengthening of panels is also possible, through the sacrifice of weight reduction and the partial removal of the polystyrene portion of the panel in order to fill in with some concrete. Additionally, the implementation of rebar is applicable in these instances. In a previous research and experimentation, it has been found that 3D panels tested under lateral loads showed tremendous results for both post-cracking strength and ductile behavior (Poluraju and Apparao 2015). Based on previous works this paper aims to examine the capacity of these panels as MSE walls.

2 FINITE ELEMENT MODELING

In this paper the finite element model was developed to investigate the behavior of MSE wall panels. Abaqus is used for finite element analysis (FEA) of the welded wire wall panels for linear and nonlinear analysis. Abaqus/CAE was employed for creating the model of the welded wire panels, monitoring the analysis job, and view the results of the analyses, and Abaqus/Explicit was used for the analyses of the panels when highly non-linear materials are showcased in a model and it allows for the explicit integration scheme to solve a system of equations in very small time increments through numerous steps and allows models to undergo large deformations (Dassault Systemes 2013).

2.1 Geometric Modeling and Boundary Conditions

Three-dimensional solid elements were used to model the panels. The concrete layers and steel anchorage system were modelled using three-dimensional solid elements. The welded wire steel mesh and diagonal truss members were considered as beam elements. Two concrete layers were modeled, one layer in each side of the wall and were designed as 1.52 m x 1.52 m layers with different spacing between the layers. The steel anchors are composed of steel bars and steel plates which were placed inside the concrete layers. The steel plates are 152 mm x 152 mm steel plates and have 6.35 mm thickness. The steel wire mesh was originally considered to have a 3.175 mm diameter welded wires with a spacing of 50.8 x 50.8 mm (Figure 1). Another consideration for the model were the diagonal truss members with 3.175 mm diameter made of steel wires attached to each mesh in both concrete layers. Boundary conditions simulating each panel as a part of the wall facing were included in the modeling process. The in-plane degrees of freedom (x and y direction) were constrained leaving the z direction perpendicular to the wall and free to move. Note that at the end of the bar portion of each anchor only the z direction was restricted. Soil pressure, which represents the soil acting on the concrete, was applied perpendicular to the inner concrete layer facing which is parallel to the direction of the steel bars in the anchorage system. This is done because this face of the concrete would be in direct contact with the soil that the wall would be retaining.



(a) Welded wire mesh. (b) Panel with anchorage. (c) Steel wires within panel. (d) Elements of panel.

Figure 1. Facing wall details.

2.2 Material Modeling

In an effort to capture the proper behavior of welded wire panels with the FE model, the material components incorporated into the FE model had to accurately describe the properties of the constituted materials and the interactions that take place between them. The behavior of the concrete, steel, and the element types used in this study are depicted in Figure 2. The materials properties are summarized in Table 1 and 2.

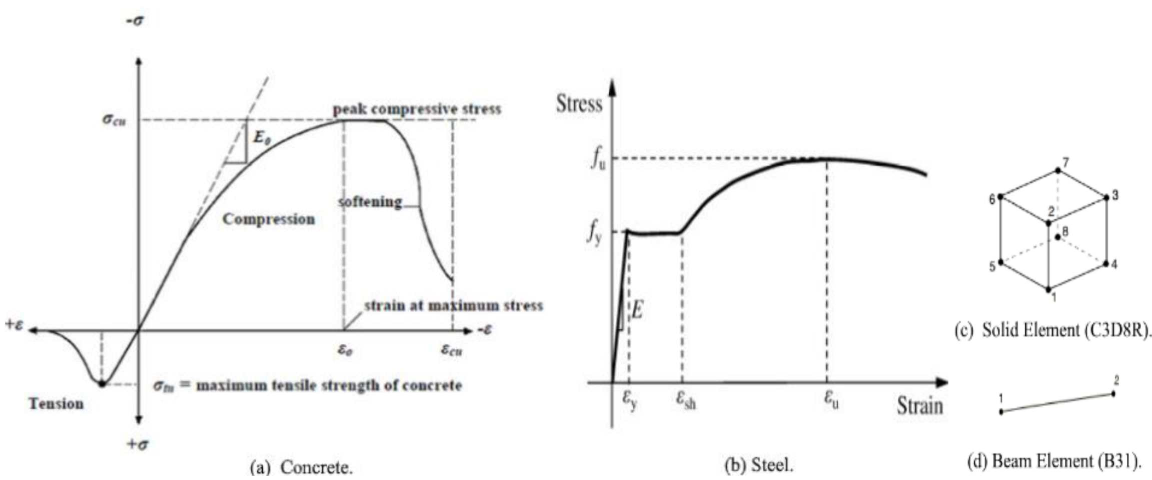


Figure 2. Stress-strain curves for materials and elements types.

Table 1. Material property for concrete.

	Density ρ (kg/m^3)	Modulus of Elasticity E (MPa)	Poissons' Ratio, v
Concrete	2380	22680	0.15

Table 2. Material property for steel.

	Density ρ (kg/m³)	Modulus of Elasticity E (MPa)	Poissons' Ratio, v	f_y (MPa)	f_u (MPa)	ε_u
Steel	7860	200000	0.32	345	470	0.0112

Additionally, the inclusion of a modeled bond between the steel reinforcement and concrete layers are of great importance going forward because it was assumed that there is perfect bond between the steel reinforcement and concrete. In the FE model the perfect bond assumption was employed by embedding the steel elements including the mesh anchors within concrete layers using the embedded element option available in ABAQUS. This option imposes a perfect bond between reinforcement and the surrounding concrete by rigidly connecting the nodes of the reinforcing elements to the nodes of the concrete layers creating an ideal situation that simulates the interaction of both materials within the welded wire panel.

2.3 Element Types

In order to model the concrete layers and anchorage system the reduced integration elements (C3D8R) were employed. Employing reduced integration elements is an effective option as this element is a 3D hexahedral shaped eight node linear brick elements with reduced integration. The 8 node element is effective when considering time constraints while still retaining precision. The elements have three degrees of freedom at each node meaning possible translations and rotations in the local x, y, and z direction at each node. The mesh size for solid elements is optimized for time of analysis and the precision of results. The steel wire mesh and steel diagonals were modelled using the two node linear beam elements (B31) (Figure 2).

2.4 Finite Element Analysis

A step by step analysis was performed to investigate the behavior and performance of the panels in linear and nonlinear domains. The amount of load/deflections is increased in each step gradually to have a better understanding of the behavior of panels.

2.5 Verification of the Model

In an effort to validate this work's accuracy, an existing experimental work was selected (Li 2007). The experimental study on a pullout test was selected and the results were provided in the report. The results from the FEA were compared to the experiment result because the current FE analysis utilizes similar geometry and material properties as the experimental work. Good agreement between the results was observed. The results produced by FEA showcase a graph that appears to be "stiffer" than the experimental results as it should be (Figure 3).

3 RESULTS AND DISCUSSION

The numerical model results for 3D welded wire panels with different configurations are presented and compared. The main goal for the present paper is to investigate the behavior of the models using pressure versus deformation curves. All the models observed showcased the performance of the panels in linear and nonlinear domains. FE analyses were conducted employing the previously described 1.52 m x 1.52 m panel. In addition to the 1.52 m x 1.52 m panel a 1.22 m x 1.22 m panel was modeled. 3D panels with different thicknesses of EPS between the concrete layers were modeled as well. In addition to concrete the yielding behavior of welded wire mesh and diagonal bars are investigated. The FE analysis results for pressure vs. deformation of the aforementioned models were compared in different graphs.

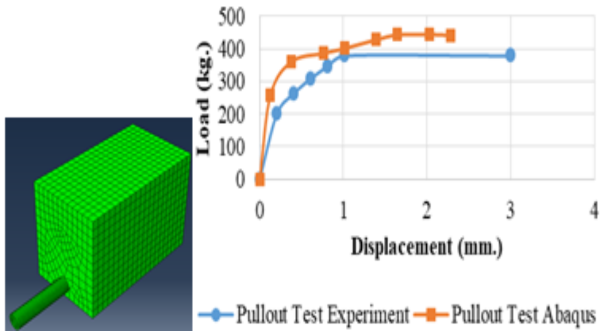


Figure 3. Numerical vs experimental model.

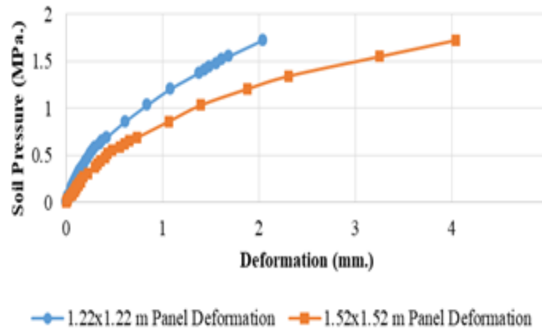
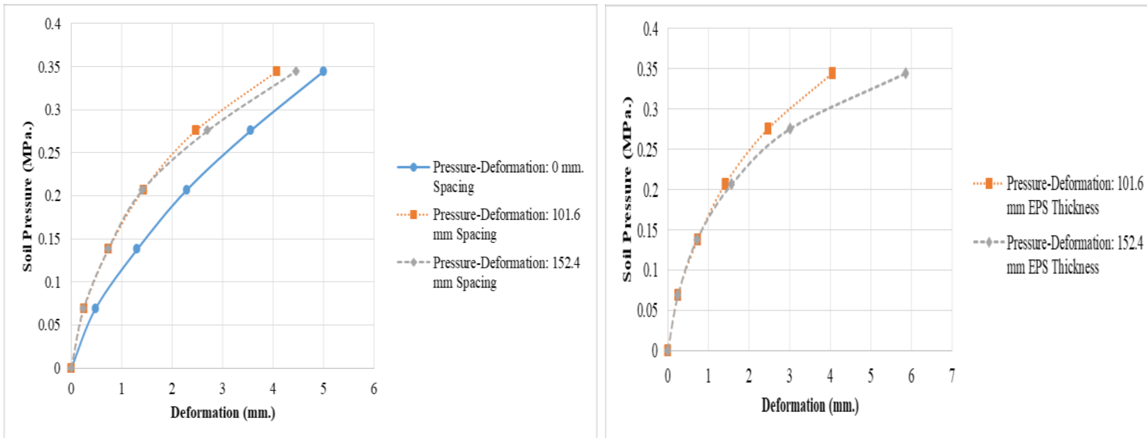


Figure 4. Soil pressure vs. Corner deformation.

Figure 4 displays the soil pressure vs. deformation at the corner of the panel with respect to the anchorage point for the models of the 1.22 m x 1.22 m panel and the 1.52 m x 1.52 m. The vertical axis represents the soil pressure on the panel (MPa) and the horizontal axis represents the deformation of the corner of the concrete layer with respect to the anchorage that the pressure is acting on (mm). It can be observed that the load versus deflection behavior has three segments: un-cracked, cracked concrete, pre yield and post yield of steel. The un-cracked phase shows a significant rise in the load with small increases in deformation. The cracked phase of the results show that as the concrete begins to crack and the deformation increases significantly with the increase of load. After the yielding of the wires the deformation increases even more rapidly with increases in soil pressure. It is observed that the 1.22 m x 1.22 m panel shows less deformation for similar soil pressure compared to the 1.52 m x 1.52 m panel.

In order to obtain a better understanding of the 3D welded wire panel additional models utilizing the same conditions were created and analyzed. Compared to the other models previously discussed the models in this section will have different spaces between the two concrete layers. The spaces between the concrete layers are 101.6 mm, 152.4 mm, and there is a model with no spacing. These panels have no EPS in between them. Similar to the models of 3D welded wire panels with different spacing between the concrete layers there are models that were created using different EPS thicknesses in between the concrete layers. In these models the thicknesses of the EPS block to be employed are 101.6 mm and 152.4 mm. In Figure 5(a), the comparison of the results for the FE analysis for panels with different spaces between the concrete layers is displayed. The panels with the 0, 101.6, and 152.4 mm space between the concrete layers are analyzed. It is observed that the steel diagonals in these cases play an important role in the panel's behavior therefore, when the diagonals yield or buckle, the deformation in the overall panel is affected. Figure 5(b) showcases the results for the panels with varying EPS thicknesses. It was observed that the stiffness of the panels increase by increasing the distance between the concrete layers from 0 to 101.6 mm but with even more increase in the distance between layers, buckling and yielding of some of the diagonal bars case reduction in stiffness of the panel. The behavior of the panels are very similar, however the panel with the 101.6 mm thick EPS does seem to be slightly stiffer similar to Figure 5(a).



(a) Varying spacing between concrete layers. (b) Varying eps thicknesses.

Figure 5. Soil pressure vs. deformation comparison of panels with various spacing and eps thicknesses.

4 CONCLUSION

In order to evaluate the behavior and performance a finite element modeling of Mechanically Stabilized Earth (MSE) walls using welded wire wall panels was performed in Abaqus. It was proven that the finite element method was an appropriate tool for the analysis of the 3D welded wire panel. Also, the anchorage system employed was sufficient. The anchors in each model were not that caused of failure in any case and there was no yielding detected in it during analysis. It was observed that the use of a thicker EPS does not always improve performance of the 3D wire welded panel. In fact, in this work the 101.6 mm thick EPS shows less deformation than the 6 inch thick EPS. Through the analysis it was determined that the mode of failure for the 3D welded wire panel was the yielding and/or buckling of the steel diagonals. The possibility of using 3D welded wire panels as the facings of Mechanically Stabilized Earth (MSE) Walls was examined and proved for the panels and materials within the scope of this paper.

References

Alzamora, D. and Anderson, S., *Review of Mechanically Stabilized Earth Wall*, Transportation Research Board Annual Meeting, Washington D.C., 2009.

Berg, R., Christopher, B., and Samtani, N., *Design of Mechanically Stabilized Earth Walls and Reinforced Soil Slopes – Volume 1*, Federal Highway Administration, Woodbury, Minnesota, 2009.

Building Materials and Technology Promotion Council Ministry of Housing and Urban Poverty Alleviation, *Quik Build Panels*, New Delhi, India, 2015.

Dassault Systemes, *Abaqus 6.13 Online Documentation,* 2013.

Li, X., *Finite Element Modeling of Skewed Reinforced Concrete Bridges and the Bond-Slip Relationship between Concrete and Reinforcement,* Auburn University, Auburn, Alabama, 2007.

Poluraju, P., and Apparao, G., *Sandwich 3D Panel System: Construction Practices and Experimental Investigations*, The 7th Asia Pacific Young Researchers and Graduates Symposium, Kuala Lumpur, India, 231-240, August 20-21, 2015.

Walters, B. X., Collins, M. P., Funk, N. E., Vessely, M. J., Widman, B. L., Koonce, J. W., Thompson, P. D., *Colorado Retaining and Noise Walls Inspection and Asset Management Manual, Version 1.0 (April 2016),* Colorado Department of Transportation, Denver, 2016.