

REINFORCEMENT EFFECT OF GEOGRID IN THE BALLAST AND SUB-BALLAST OF THE RAILWAY TRACK

Saad Farhan Ibrahim¹, Ali Jabbar Kadhim² and Harith Basim Khalaf²

¹Faculty of Engineering, Isra University, Amman, Jordan;

²Faculty of Engineering, Al-Mustansiriya University, Baghdad, Iraq

*Corresponding Author, Received: 4 Feb. 2018, Revised: 21 Feb. 2018, Accepted: 20 March 2018

ABSTRACT: The railway transportation in Iraq is essentially irreplaceable for its significant role in economic growth through revenue derived from the movement of goods and raw materials and passengers between Iraq main cities. The objective of this paper is to study the effect of geogrid reinforcement on railway tracklayers through the use of Abaqus program. The ABAQUS program was considered as multipurpose modeling program, the program cannot only solve the stress-strain problems but also other parameters like heat transfer, Contact, dynamics, and electromagnetics. The model dimensions are (800*800*600 mm), subgrade layer is 400 mm thick and sub-ballast layer and ballast layer are 100 mm thick. Some laboratory testing was carried out to determine the materials properties that were used as inputs for the finite element Abaqus program. The tested models are an unreinforced model, sub-ballast reinforced model, and ballast reinforced model and a double reinforced model consisting of geogrid reinforcement in the sub-ballast layer and ballast layer. The result showed that using reinforcement layer would reduce model settlement with a range between (38-18) percentage for total model settlement for the different models that have been tested. In addition, it is obvious that use of geogrid under the ballast layer is more effective to reduce model settlement if there is an economical restraint to use only one layer of reinforcement.

Keywords: Railway, Abaqus, Geogrid, Reinforcement, settlement.

1. INTRODUCTION

A Finite element method is a broad approach for modeling the track system. Since 1960, several attempts have been made to overcome the limitations in track modeling by using this technique. Meacham was made the first attempt to develop a track model suitable for computerized analysis (both digital and analog) [1]. The ABAQUS is an effective finite element software package. This has been utilized in many different engineering fields around the world. ABAQUS program can execute a static and dynamic evaluation and simulation of different complicated engineering as well as non-engineering issues. The Abaqus program can handle with bodies with different loads, interactions, properties, boundary conditions, temperatures, element types, and other environmental conditions.

Keene Developed a linear-elastic 3D finite element model using ABAQUS 9.6 where mechanical and material properties of the superstructure and substructure taken from the Army Corps of Engineers Railroad Design and Rehabilitation Technical Instructions [2].

Jianxin Liu presented a numerical study of Response of Railway Subgrade using Infinite Element Method, the work implements a continuous elastic beam support approach to model the railway track structure and using ABAQUS

program a three-dimensional finite element model of heavy haul railway subgrade developed. The dynamic reactions of subgrade layer to various axle loading are investigated in longitudinal, vertical, transverse direction [3].

2. METHODOLOGY

The ABAQUS program provides extensive possibilities for simulating the nonlinear and linear application. When a problem is composed of several components, each one of the components is defined with the suitable interactions between them and with specifying the appropriate geometry and materials models. The ABAQUS during the analysis of the problem, the program will choose automatically the tolerances of convergence, the suitable increment of load, and continuously modify them through the analysis to make sure the solution is obtainable accurately and efficiently [4]. Because the fundamental reason for the Abaqus consumer program is the rendering of a specific situation or even solve a particular problem after which to find a solution, therefore it is needed to deal with this program in its personal language, including creating a design then examining it. These types of modules ought to be used for determining geometry, materials properties along with other physical attributes and then publish the type for evaluation.

Building a finite element model requires the following steps:

- 1) Modeling tools
- 2) Geometry
- 3) Meshing
- 4) Analysis procedures
- 5) Advanced materials modeling
- 6) Contact
- 7) Visualization

Because the behavior of the layered railway system may not be approximated utilizing shell, truss, or beam elements, A 3d continuum solid elements in many cases are selected in order to simulate the issue in concern. The first model consists of three connected layers and a geogrid layer intrusion in the last three models. A metric (SI) consistent has been for the model details input for Abaqus program.

The ballast and sub-ballast part were modeled with equal dimension just as the tested model, which is ((0.8*0.8*0.1)m) and the subgrade part dimensions, is ((0.8*0.8*0.4) m).using the part manger, A deformable solid have been chosen for creating the three layers of the railway and a deformable shell has been chosen for the geogrid. The 3D railway layers are shown in Fig.1.

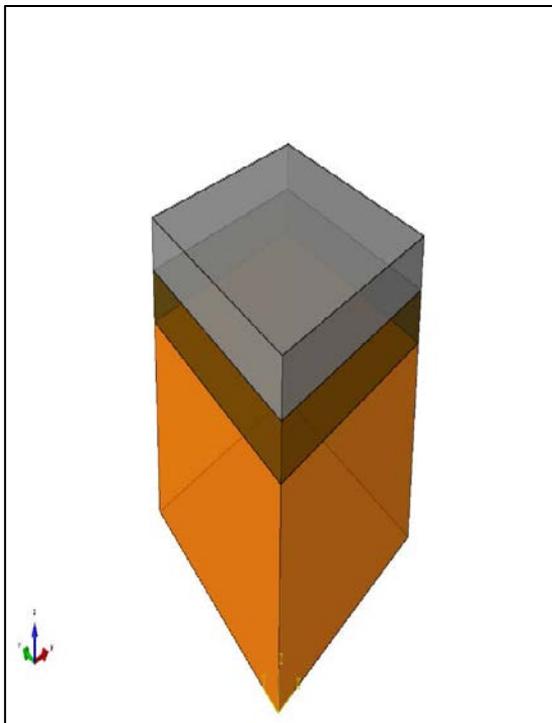


Fig. (1) Model layers parts.

The model structure has the same layers for raw materials of the railway track (ballast, sub-ballast and subgrade) except the addition of geogrid reinforcement layer between railway layers; the modeled geogrid is shown in fig.2

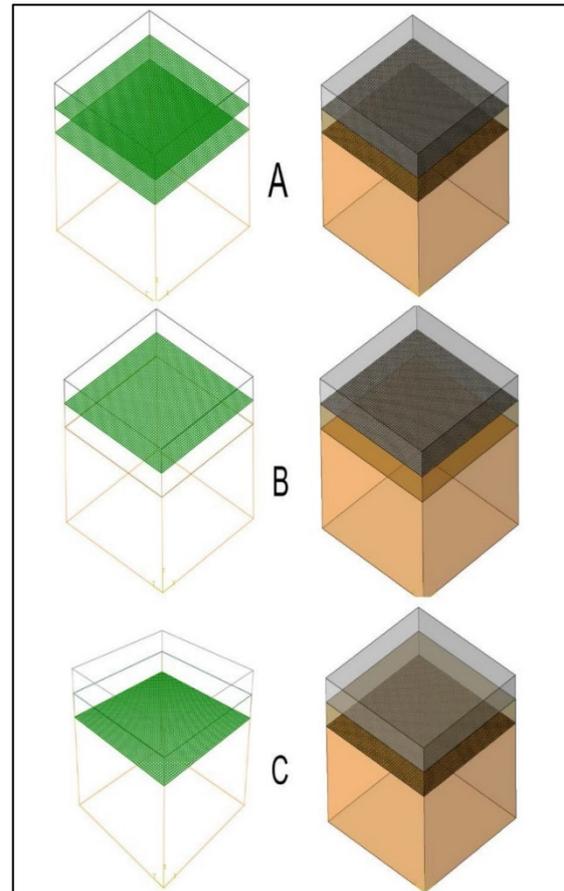


Fig. (2) The reinforced model (A: double reinforced - B: ballast reinforced - C: sub-ballast reinforced).

3. MATERIAL PROPERTY

The key aspect of Finite Element Studies is the use appropriate material property of the content characteristics; due to the fact, their conducts could definitely influence often the responses with the modeling.

The model materials have been assumed to react elastically and linearly to the displacement load as the loading are being applied in the linear perturbation step. These properties regard the input parameters to the program. These parameters are the elastic modulus (E), the Poisson's ratio (ν) and the density. The used input that has in the analysis has been stated in the table (1).

Table 1 Material Properties

layer	Density (Kg/m ³)	Young's Modulus (MPa)	Poisson's Ratio (ν)
ballast	1783	130	0.45
Subballast	2146	110	0.35
Subgrade	1681	10	0.3

4. ANALYSIS STEPS

To start a simulation of the geotechnical problem, it should start with the geostatic state. The geostatic state is the equilibrium configuration for the steady state condition of the undisturbed rock body or soil during a geostatic load applied on them and typically including the vertical and the horizontal component. Therefore, the establishment of layers initial conditions is very important in order to start solving the problem from the equilibrium state. The actual geostatic process could be used to confirm that the user-defined initial stress is certainly within equilibrium using the specified boundary condition as well as geostatic load [4].

The actual geostatic process is typically utilized as the initial step of the geotechnical analysis; in these instances, loads of the gravity are used through this stage and the masses and primary stress must closely equilibrate also creating an approximate of zero deformations at step end. For establishing the equilibrium for this model, the enhanced method used which in the program automatically calculates the actual equilibrium related to the initial settings and the initial load, permitting just minor displacement within a user-specified tolerance. (Is the default threshold).the displacement in the model after the geostatic step is shown in fig.3.

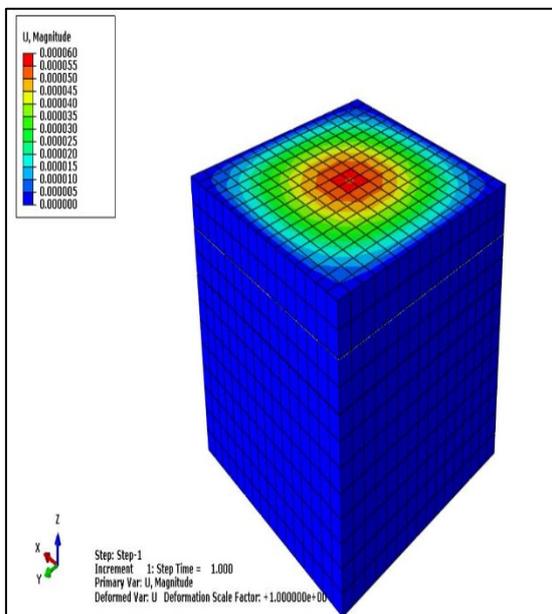


Fig. 3 Model displacement after geostatic step.

A two-step approach was used to analyze the model, the first one was a geostatic step to be able to reach the equilibrium state and the gravity loading was applied to it. The second step was a dynamic one with a defined timing similar to the experimental test and the external loading was applied to it.

5. LOAD AND BOUNDARY CONDITION

Two types of loading used for the analysis of the model first load was a gravity load in which the gravitational acceleration of $9.81(m/s^2)$ is applied to the model. the second load was applied using controlled displacement method, in which the load is represented by a displacement at a constant rate similar to the speed of the laboratory tested model on an area is similar to the area of the footing of the tested model.

The Boundary conditions greatly affecting the prediction for finite element model responses. Therefore, the actual boundary circumstances have a substantial influence on predicting the actual response from the model. The base surface of the model, as well as sides associated with layers, is actually assumed to become fixed. Fig.4 shows the boundary conditions used in the analysis.

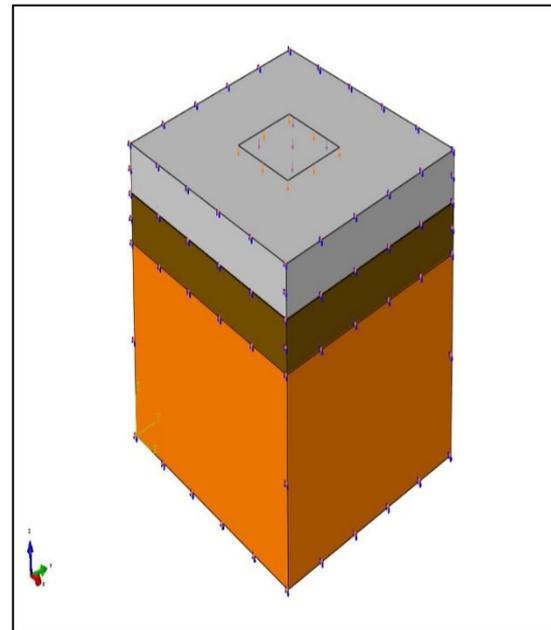


Fig. 4 The Boundary conditions distribution in the model

6. INTERACTION MODELING TECHNIQUES

The contact interaction between two layers of the model using ABAQUS (6.14-4) require to define surfaces of interaction, therefore surfaces have been created at the top and the bottom of layers that will be in contact with other layers. The ABAQUS program is providing many contact formulations. Each one of the formulations is been founded on a choice of a contact discretization, Some sort of tracking technique, and plan of “master” and “slave” roles towards contact types of surface. The duty of “master” and “slave” roles to your contact is used to model typically the

interaction involving layers.

In addition, the interaction is a step dependent substance, and that will mean, you should take care in defining them, especially when choosing the step where the interaction will be active or not. The interference between the model's layers was modeled using Surface-to-surface contact, which is used to define Interference between two surfaces.

For the interaction between model layers with each other a tie constrain module has been used and for the geogrid interaction with model layers a complete interlocking between model layers and geogrid is actually assumed and for that reason, the geogrid-soil interactions had been modeled utilizing 2 completely bonded slave/ master contact areas. This ensures that no slippage will happen in the geogrid-soil interference. The dependent constraints of the surface tend to be forced the elimination of the degree of freedom from slave surfaces to preserve exactly the equivalent rotational as well as translational movement same to the ones from the master surfaces. It had been done through the assignment of the actual geogrid layer with a slave surface and soil layers with master surfaces. Fig.5 shows the location of contact states in the model.

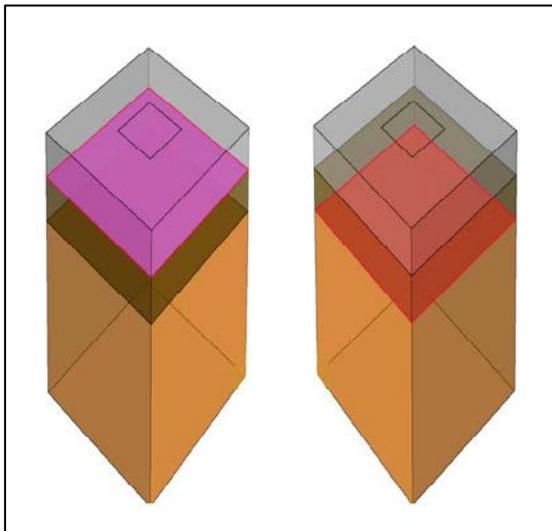


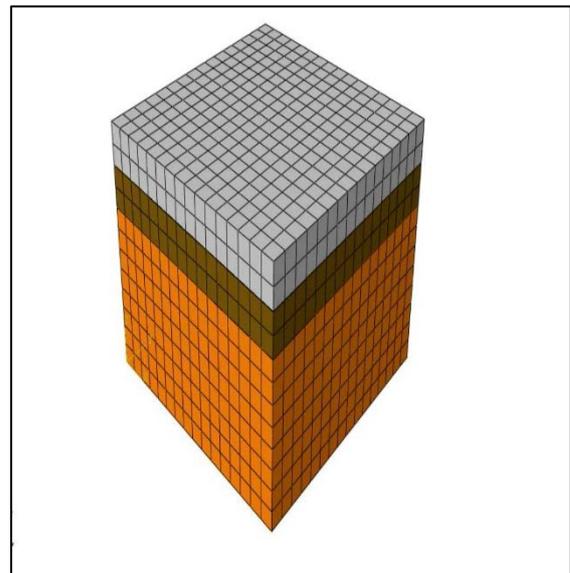
Fig. 5 the location of a contact in the Model.

7. MESH SIZE AND ELEMENT TYPES

The (3D) finite element evaluation was performed by taking benefit of Abaqus capabilities. The model was constructed of 3014 elements along with 4368 nodes. The model meshed with two types of element.

The layers of the particular model except the geogrid are generally modeled utilizing the 8-node continuum three-dimensional (C3D8R) brick element with decreased order statistical integration obtainable in ABAQUS (6. 14.4). This particular element has the capability of representing large

geometric deformation, as well as material non-linearity. The Solid element (C3D8R) offers 3 degrees of freedom for every node, movement in the crucial x, y, and z directions. The element (C3D8R) is really a first order (linear) element and it does not have a rotational degree of freedom. All the layers were simulated using the same form to preserve typically the continuity associated with nodes among consecutive layers as shown in fig.6. The Membrane element (M3D4R) have the ability to simulate thin shells in the space that do not have a stiffness for bending but have higher strength in the plane of the element. So the geogrid reinforcement layer was modeled using (M3D4R) are four node quadrilateral, hourglass control with reduced integration elements. Surfaces of this layer were utilized to simulate interactions between the model



layers and geogrid

Fig. 6 3D finite element mesh density.

8. ABAQUS MODELING RESULTS

The end result of the ABAQUS program outcomes can be obtained via entering certain parameters for every model, precise parameters consist of poisson ratio, modulus of elasticity, time and dimension. Then Finite Element Program is applied to measure the displacement that will occur. The obtained results have been compared with the experimental test results.

8.1 Unreinforced model

The vertical displacement in the ABAQUS model analysis used as the main factor to compare between different models. The displacement is considered as a response to applying boundary condition displacement. The magnitude of the displacement U3 beneath the center of the load is can be viewed in Fig.7. The displacement across the

top of the model is shown in fig.8.

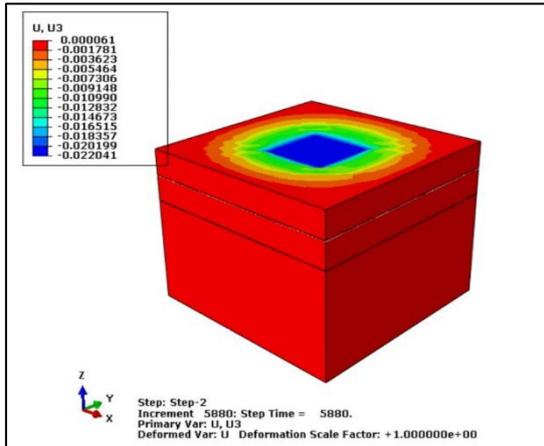


Fig.7 Displacement for the unreinforced model.

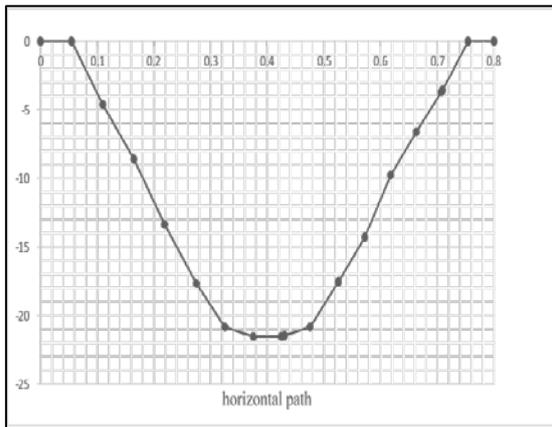


Fig. 8 Displacement along a horizontal path for the unreinforced model.

8.2 The sub-ballast reinforced model

The magnitude of the displacement U3 beneath the center of the load for the sub-ballast reinforced model is shown in Fig.9. The displacement of the model top is can be viewed in fig.10.

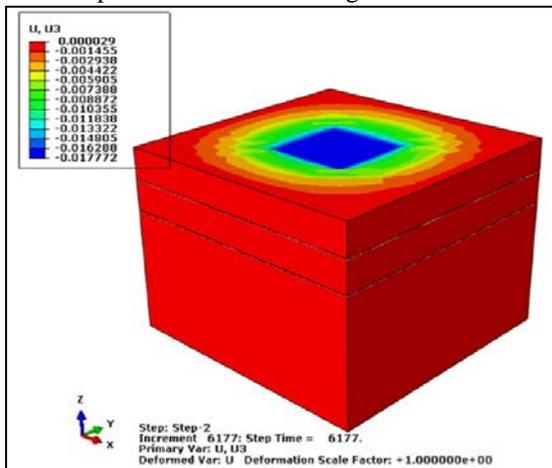


Fig.9. Displacement of sub-ballast reinforced model

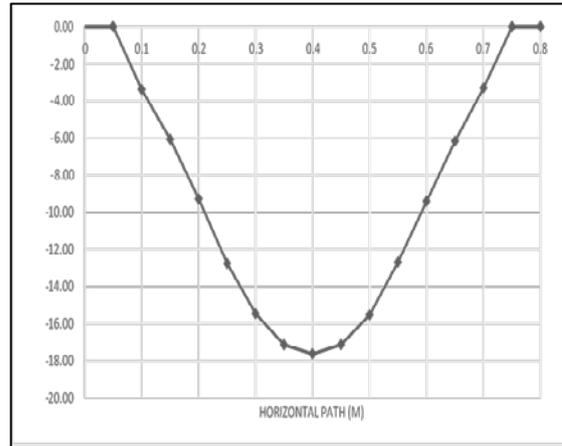


Fig.10 Displacement along a horizontal path for sub-ballast reinforced model

8.3 The ballast reinforced model

The magnitude of the displacement U3 beneath the center of the ballast- reinforced model is shown in Fig.11.the displacement across the model tops is can be viewed in fig.12.

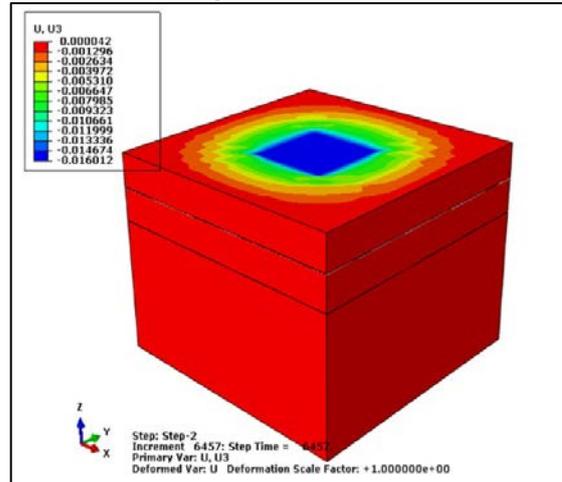


Fig. 11 Displacement of the ballast reinforced model

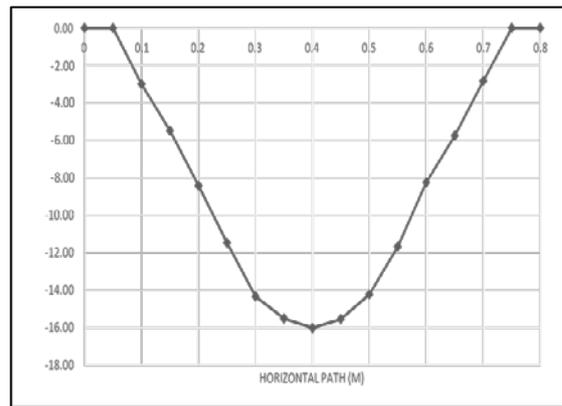


Fig.12 Displacement along a horizontal path for the ballast reinforced model.

8.4 The double reinforced model

The magnitude of the displacement U3 beneath the center of the double reinforced model is shown in Fig.13. The displacement of the model tops is can be viewed fig.14.

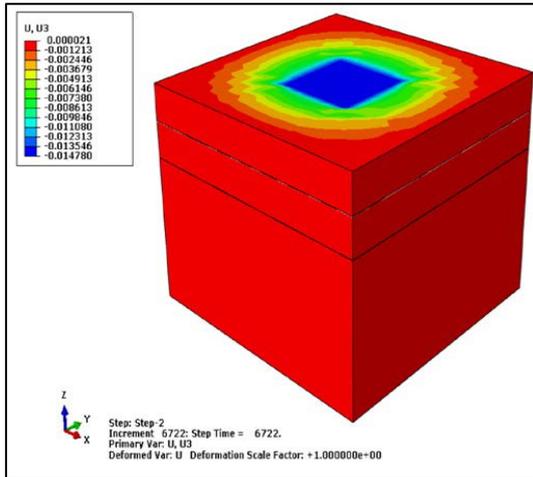


Fig.13 Displacement for the double reinforced model.

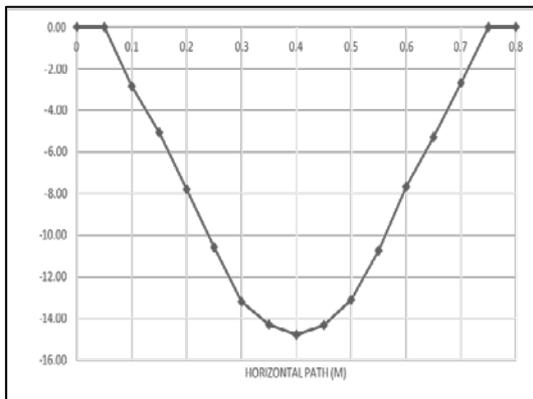


Fig. 14 Displacement along a horizontal path for the double reinforced model

9. CONCLUSION

There is an obvious effect of the geogrid position and number of geogrid layers in reducing the amount of settlement. Geogrid offer an enhanced combination of interlock within stabilizing railway track infrastructure via confinement of both ballast and sub-ballast particles. The amount of settlement increased with increasing the simulated load. The double reinforced model have the best result in settlement reduction (38%) As compared to equivalent single reinforcement model. The ballast-reinforced model is more firm and gives much more strength (28%) as compared to equivalent sub-ballast reinforced model (17%). Geogrid reinforcement is more effective for ballast layer because the ballast has the major role in Transmitting and distributing the load to formation uniformly.

10. REFERENCES

- [1] Meacham, H.G., Prances, D.E., Albeck, and Ieasuba, J.A., (1970), Study for Rail Vehicle Track Structure, Office of High-speed Ground Transportation, Federal Railroad Administration, Report No. FRA-RT-71-45, Washington, pp 223-273.
- [2] Keene, A, Tuncer, E, Fratta, D & Tinjum, J 2013. Modeling the effect of polyurethane stabilization on rail track response. Proceedings, Geo-Congress 2013, 3–7 March 2013, San Diego, CA.
- [3] Jianxin Liu, Pengmao Liu, Xiaozhong Tong, Chuanghua Cao: “Dynamic Response of Railway Subgrade by Infinite Element Method” Electronic Journal of Geotechnical Engineering, 2016 (21.11), pp 4151-4162.
- [4] ABAQUS Theory User's manual, (2011), Inc., Pawtucket, USA.