International Journal of GEOMATE, July, 2017, Vol.13, Issue 35, pp.174 - 179
Geotec., Const. Mat. & Env., ISSN:2186-2990, Japan, DOI: http://dx.doi.org/10.21660/2017.151216

EVALUATION OF SOIL REINFORCED WITH GEOGRID IN SUBGRADE LAYER USING FINITE ELEMENT TECHNIQUES

*****Saad Farhan Ibrahim AlAbdullah[1] and Noor Sabah Taresh[2]

[1]Faculty of Engineering, Middle East University,Amman,Jordan,[2]Highway and Transportation Engineering Department

*Corresponding author, Received: 12 Dec. 2016, Revised: 2 Feb. 2017, Accepted: 29 Feb. 2017

ABSTRACT: The paper describes Finite Element (FE) models that are used to simulate the behavior of two types of geogrid laid between subbase/subgrade layers in flexible pavement structure. The properties of materials used to predict behavior of the sand soil is required to give accurate predictions of pavement performance. Physical and Mechanical properties of the pavement structure components of are established using standard laboratory tests including repeated load test with geogrid reinforcement. The results of analyses show that the finite element model results and implementation reported in this study can accurately predict permanent deformation in the pavement structure.

Keywords: Improvement, Geogrid, Reinforcement, Dynamic, Subgrade

1. INTRODUCTION

Finite element method (FEM) is a numerical analysis technique proposed to be used in the present study to determine the stress, strain and deflection of the pavement layers .FEM is one of the most common numerical methods used to obtain an approximate solution for complex problems in several fields of engineering. Abaqus, a commercial finite element modeling program, has been widely used for pavement analysis. Al-Qadi and Wang Hao simulated the flexible pavement responses under repeated load, by using three dimensional dynamic analyses in Abaqus, it is suggested to use ABAQUS as a suitable software to carry out the analysis of this study [2].the main objective of this study is to evaluate permanent deformation of pavement structures using advance analysis. This paper is studying the use of geogrid to improve the properties of weak subgrade.

2. F.E MODELLING USING ABAQUS

Abaqus also includes many material models such as linear elastic, elasto plastic, and viscoelastic models. FEM calculates the values of stresses and strains by node to node lumping of loads. Therefore, appropriate element selection and meshing are also very necessary [5]. ABAQUS is the most widely used general-purpose program to solve engineering problems based upon the finite element method applicable to nonlinear and linear solutions [3]. A three-dimensional FE model of three-layer pavement structure is applied to simulate the pavement response in terms of viscoelastic and viscoelastic [4].

Permanent deformation strains under repeated loading. The simulation results show that, the VE and VP vertical strain zones develop with the increasing number of cycles and then, at certain number of cycles, the distributed zones do not increase in size, whereas, the magnitude of strains continues to increase [4]. An infinite element model can be used to model the infinite boundary conditions in the horizontal and vertical directions in a pavement system. ABAQUS provides many element types that are useful for pavement analysis. The ABAQUS suite of engineering analysis software packages are used all over the world to simulate the physical response of solid bodies and structures to load, impact, contact, temperature, and other environmental conditions [1].

Abaqus program user is the representation of a particular situation or analysis for a certain problem and then how to find a solution for it. These models should be used for defining material properties, geometry and other physical properties and then submit the model for analysis [5]. Building a FE model requires the following steps:

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

3. COMPONENTS OF ABAQUS

The ABAQUS finite element system includes:
1- ABAQUS /Explicit, an explicit dynamics finite element program.
2- ABAQUS /Standard, a general-purpose finite element program.
3- ABAQUS /CAE, an interactive environment used to create finite element models, submit ABAQUS analyses, monitor and diagnose jobs, and evaluate results.
4- ABAQUS /Viewer, a subset of ABAQUS /CAE that includes only the post processing capabilities of the Visualization module.(Mohammed, 2015)

4. MATERIAL CHARACTERIZATION

Three layers of pavement structure asphalt, granular subbase and sand subgrade are used. Asphalt pavement, subbase materials are assumed to respond linearly and elastically, while subgrade materials elasto plastic, use drucker prager. Elastic properties (modulus of elasticity and Poisson's ratio)[5].

5. ABAQUS MODEL GEOMETRY

Table1.shows three layers of pavement; subgrade, subbase, and asphalt surface. The pavement layers are modeled, as volumes (3D model), Figure 1.

Models in series

Description of model	Symbol
Unreinforced with pavement thickness 50 mm at 55% relative density	S-50-UNRE
Reinforced with Iranian geogrid pavement thickness 50 mm at 55% relative density	S-50-RE-G1
Reinforced with Nelton CE121 geogrid pavement thickness 50 mm at 55% relative density	S-50-RE-G2

Table 1. Pavements layers

Figure 1 shows the geometry of model, the asphalt layer thickness which is 50mm, subbase layer 100mm and subgrade200mm.The model is with dimensions of (600x500x350mm). ABAQUS comprise of modules which are used for the model inputs, application and imagining.

Material used in Abaqus to simulate subgrade drucker Prager is used to simulate elasto plastic. The Triaxial test used to determine Modulus of Elasticity and the direct shear test is used to determine the angle of friction of sand subgrade,

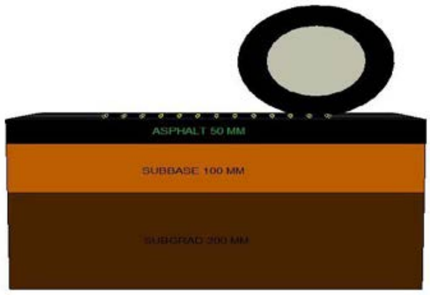


Fig. 1 General Geometry of 3-D Model of the Pavement Layers by Abaqus Program.

In this numerical approach, the effect of geogrid types of reinforcement on permanent deformation of the flexible pavement will be studied, under repeated traffic load. Table 1. Shows the series symbols.

6. MATERIAL PROPERTIES

3D model is built by using the finite elements program Abaqus (6.12-3), to understand, with more precision, the distribution of the deformation in the pavement layers system, when different types of geogrid are used. The properties of the materials used in this analysis are concise in Table 1. These properties regard the input parameters to the program. These parameters are: the elastic modulus (E), the Poisson's ratio (ν) and the density.

Table 2. Material Properties of Input Data

Layers	Density (Kg/m³)	Modulus of Elasticity (MPa)	Poisson's ratio(ν) *	Angle of friction (θ)
Asphalt layer	2335	1200	0.35	-
Subbase layer	2250	111	0.35	-
Subgrade medium Layer	1778	2.2	0.3	35

* Poisson ratio Huang[4]

7. SIMULATION OF MOVING LOAD

The wheel load applied in the Abaqus is 96 kg (0.96 kN) and is distributed uniformly over the total contact area of asphalt layer. The resulting uniform contact pressure is 550 MPa which is equal to the pressure of tire are used. Two parameters, transverse and longitudinal distribution of vertical pressure on loaded area are used to simulate moving load [2]. Loading is applied to simulate wheel horizontal movement in a pre-determined speed. In this method, loading location should be moved in a regular form in order to have a complete wheel rolling, Figure 2[7].

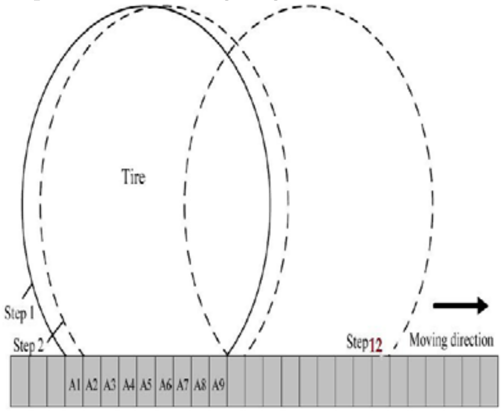


Fig. 2 Schematic Illustration of Tire Moving along Pavement Surface**.**

To simulate the moving load on the surface of pavement structure as in test, the pavement is divided into 12 steps which simulate the track of tire on asphalt layer, each step has number of cycles by using time-amplitude tabular, as shown in Figures 3. , 4. and 5.

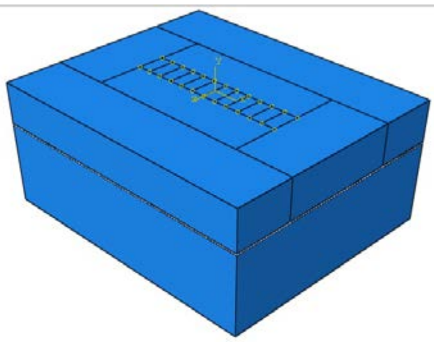


Fig. 3 Steps Which Used in Abaqus.

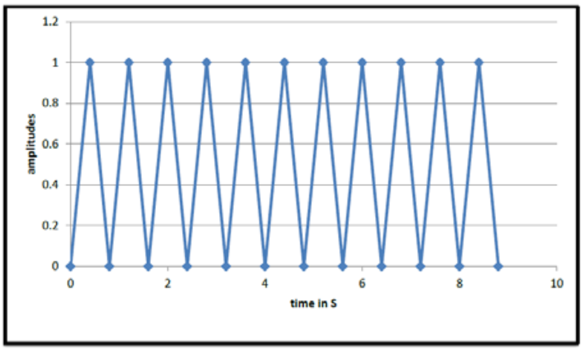


Fig. 4 Simulate Repeated Load by Use Time /Amplitude Tubular [6].

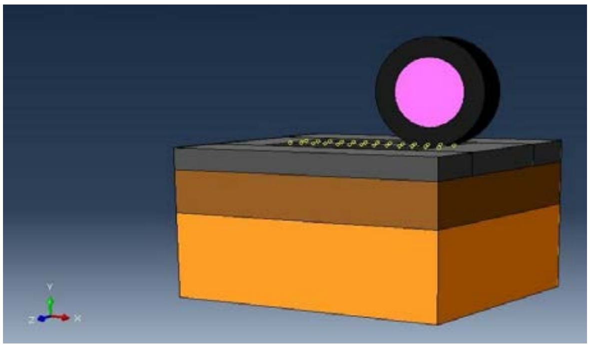


Figure 5. The Rubber Tire on Path

8. Boundary Condition

The boundary conditions have a significant influence in predicting the response of the model to the stress , the sides of layers, and the bottom surface of subgrade layer is assumed to be fixed, which that means that nodes at the sides of layer and bottom of the subgrade cannot move vertically or horizontally, where (U1=U2=U3=UR1=UR2=UR3= 0) this represents the steel box condition. Figure 6. Shows the boundary conditions used in the models analysis. The most important aspect of FE Analyses is the simulation of the material characteristics, since their behaviors could really influence the responses of the modeling. In this study, several materials are involved with various properties, but basically they are divided into two categories: linear elastic and elasto-plastic.

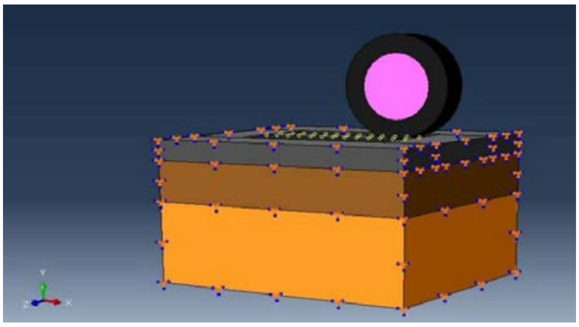


Fig. 6 Boundary Conditions for Sides and Bottom for Models

9. Element Types and Mesh Size

All of the model parts are modeled using the 8-node continuum three dimensional brick element (C3D8R) with reduced order numerical integration available in Abaqus (6.12-3). This element has the capability of representing large deformation, geometric and material nonlinear solid element (C3D8R) which has three degrees of freedom at each node, Figure 7. C3D8R Element, All layers are imitated with the same shape to preserve the continuity of nodes between consecutive layers, (Mohammed, 2015). Figure 8. Shows the total model meshing.

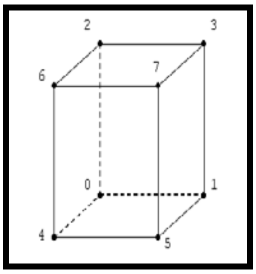


Fig. 7 C3D8R Element

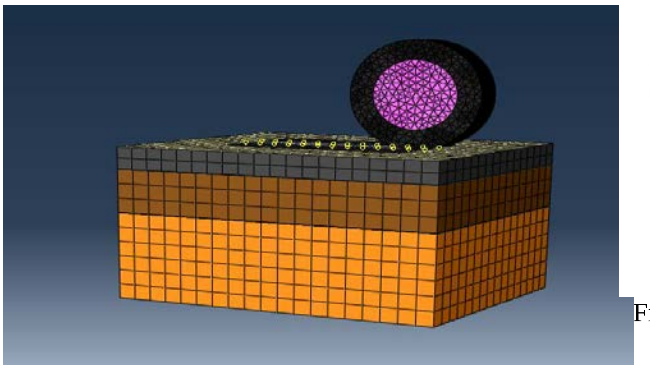


Fig. 8 Mesh of All Layers.

10. Interaction Modeling Techniques

Generation of contact interaction between the layers of the model using Abaqus (6.12-3) needs to define surfaces of interaction for each layer. Abaqus/Standard provides several contact formulations. The assignment of “master” and “slave” roles to the contact surfaces is used to model the interaction between layers. The surface - to-surface contact with small sliding is considered for all the contact interactions in the model which assumed a relatively small sliding. In the Interaction model, one can constrain the degrees of freedom between regions of a model. Interaction allows one to combine two regions, (Teama, 2014). Figure 9. Shows formulation of contact states in the model. Figures 10. and 11. show interaction between layers and formulation of contact element used in the model of pavement respectively.

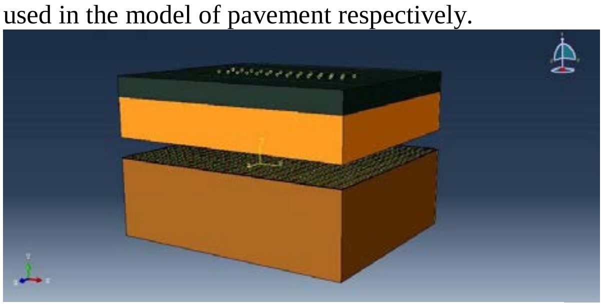


Fig. 9 formulation of contact states in the model.

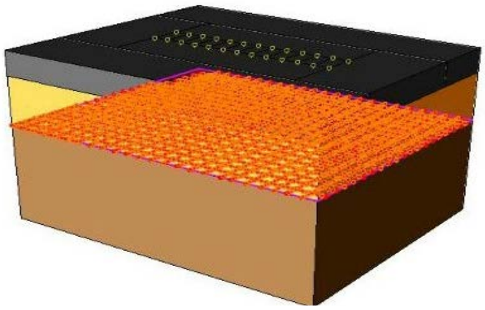


Fig. 10. Interaction between layers.

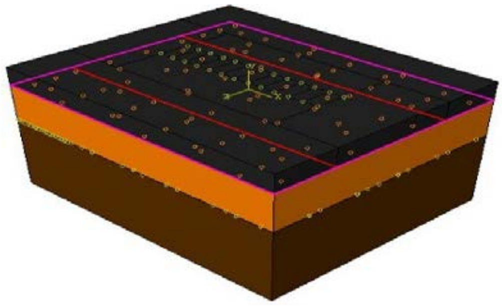
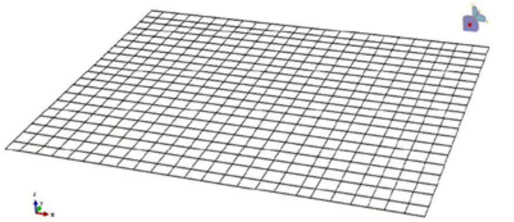


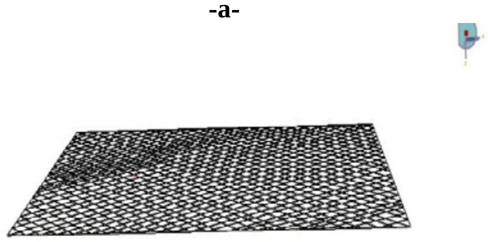
Fig. 11. Formulation of Contact Element Used in the Model of Pavement.

11. Model Geometry for Geogrid Model.

The addition of geogrid between subbase and subgrade layer is simulation in program to obtain the effect of reinforcement in models, as shown in figure (5.12) two types of geogrid.



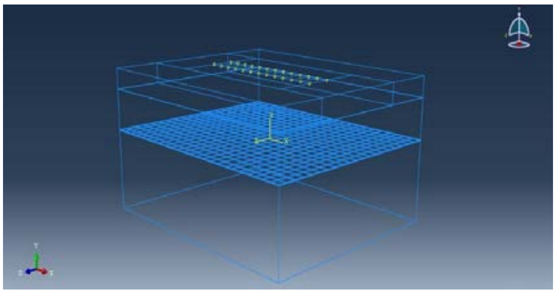
-a-



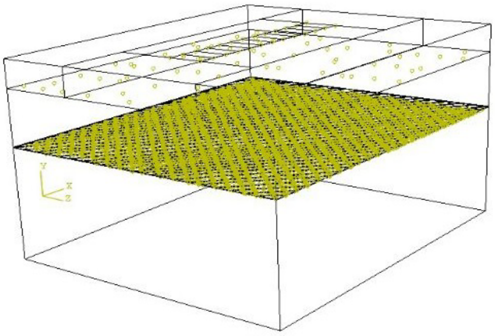
-b-

Fig. 12 Geogrids Geometry
a.Iranian geogrid
b. Nelton CE121 geogrid.

Figure 13. Shows translucent model of geogrid once with Iranian geogrid and the other with Nelton CE121 geogrid.



a.



b.

Fig. 13 Translucent Model of Geogrid
a. Iranian **b**.NeltonCE121

12. Finite Element ABAQUS Result

From laboratory tests No. of passes are used in Abaqus program and the output of rutting is shown in Figures 14. , 15. and 16. S2-50-UNRE, S2-50-RE-G1 and S2-50-RE-G2 respectively. The results are very close to experimental work. The number of passes are obtained from laboratory tests input to Abaqus.

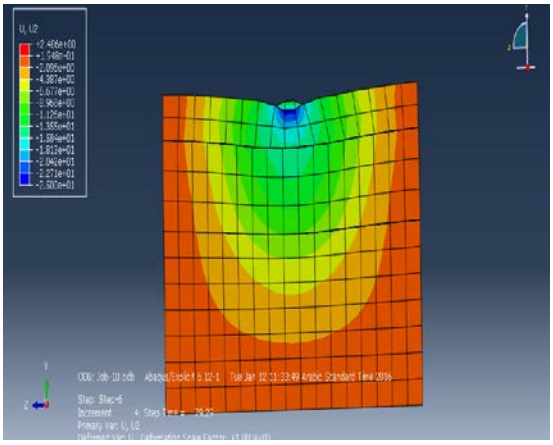
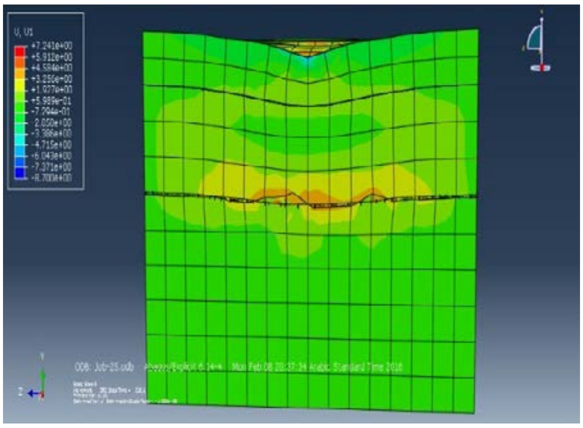
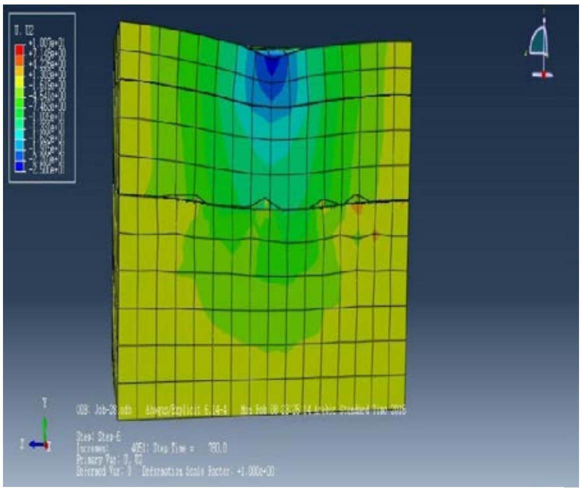


Fig. 14 Vertical Displacement for the Model for S-50-UN.

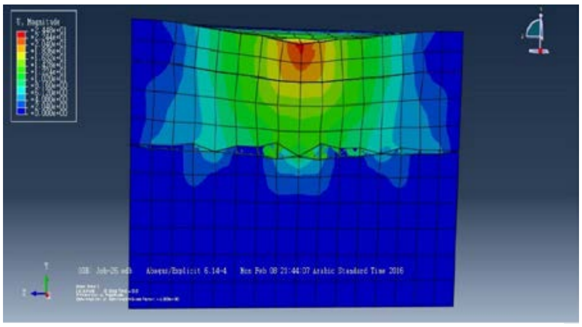


a.

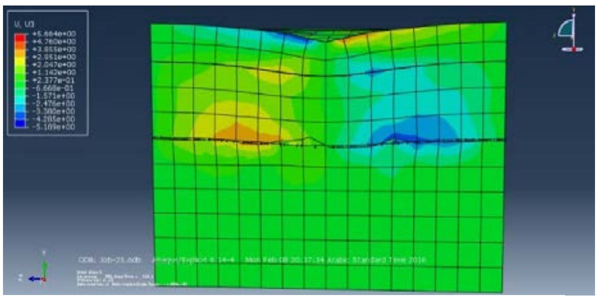


b.

Fig. 15. Displacement for the Model Materials for S-50-RE-G1 a.x-axis b. y-axis



a-



b-

Fig. 16 Displacement for the Model for S-50-RE-G2 a.magnitude b. z-axis

13. CONCLUSION

Through the simulation and analysis of the results of the repeated loads and the value of permanent deformation (rutting) that occurred in the models, the finite element program ABAQUS 3d results concluded that using Nelton geogrid is better alternative than other geogrids in improving bearing capacity of subgrade layer.

14. REFERENCES

1. ABAQUS Theory User's manual, (2009) version 6.9, Inc., Pawtucket, USA.
2. Al-Qadi and Wang H.(2009):" Pavement Damage Due to Different Tire and Loading Configurations on Secondary Roads" USDOT Region Regional University Transportation Center Final Report.
3. Hibbit, Karlsson and Sorensen, Inc. (2005). "*ABAQUS/Standard User's Manual*", Version 6.5. Hibbit, Karlsson and Sorensen, Inc., Pawtucket, R.I.
4. [4] Huang C. W., Abd Al-rub R. K., Masad E. A. and Little D. (2011): "Three-Dimensional Simulations Of Asphalt Pavement Permanent Deformation Using A Nonlinear Viscoelastic And Viscoelastic Model" journal of materials in civil engineering January 2011.
5. Masood, G.G. (2013): " Experimental and Numerical Investigation of stabilized unbound Granular Pavement Materials" AL-Mustansiriyah University College of Engineering Highway and Transportation Engineering Department MSc Thesis.
6. Teama, Z. T., (2014):"Suitability of Dune Sands Subgrade beneath Flexible Pavement Structure under Repeated Loads", M.Sc., thesis college of engineering, Al.Mustansiriyah university.
7. Mohammed D. E., (2015) "Developing a Criteria to Improve Permanent Deformation of Pavement Layers on Weak Subgrade Soil", M.Sc., Thesis College of engineering, Al.Mustansiriyah university.