# THREE DIMENSIONAL FINITE ELEMENT MODELING OF FLEXIBLE PAVEMENTS

By:
Beena Sukumaran,
Associate Professor, Civil and Environmental Engineering, Rowan University
Nishanth Chamala, Michael Willis, Josh Davis, Scott Jurewicz, and Vishal Kyatham,
Graduate Assistants, Rowan University, Glassboro, NJ 08028, USA
Phone: (856) 256-5324; Fax: (856)256-5242
sukumaran@rowan.edu

PRESENTED FOR THE
2004 FAA WORLDWIDE AIRPORT TECHNOLOGY TRANSFER CONFERENCE
Atlantic City, New Jersey, USA

April 2004

## ABSTRACT

This paper documents the use of finite element analyses techniques to determine the failure mechanism in a pavement system under moving aircraft loads. The flexible pavement system that is modeled is on a medium strength subgrade. The stress-strain response of the medium soft clay is simulated using an elasto-plastic model. The three-dimensionality of the failure surface under actual wheel loads with wander requires that computationally intensive three-dimensional models be used. The finite element techniques employed are verified against available failure data from the National Airport Pavement Test Facility (NAPTF) of the Federal Aviation Administration based in Atlantic City. The paper will discuss the advantages and limitations of the non-linear elastic models that are currently used in pavement analysis. In addition, the paper will also discuss efficient finite element techniques that can be utilized for three-dimensional analysis that will reduce computational time without sacrificing accuracy.

## INTRODUCTION

Three-dimensional finite element analysis tools are increasingly viewed as the best approach to answering certain fundamental questions about pavement performance (Chen, et al [1], Cho. et al [2], Kuo et al [3]), but the tedious processing and time required to accurately model pavement systems have hampered the use of these analyses. While two-dimensional axi-symmetric models can be utilized for a single wheel load analysis, such a constraint would lead to an inaccurate three-dimensional analysis, particularly for pavements subjected to multiple wheel loads and wander.

The objective of this paper is to discuss finite element modeling strategies that can be used for analyzing pavement systems. The goal of this paper is to discover a less computationally intensive model that will still maintain the accuracy of an infinitely integrated model. To accomplish this task, a reduced integration element is used that would take advantage of the ABAQUS modeling software. The material model employed for the clay and gravelly material is an elasto-plastic Drucker-Prager model. In addition, meshing strategies for pavements and issues of mesh refinement and element aspect ratios required for accuracy is discussed. To accommodate the memory constraints, mesh gradation and infinite elements are used which reduce computation time with a reduced overall model size. In addition, the use of symmetry in the three-dimensional model is explored by demonstrating the ability to predict pavement responses for symmetrically loaded conditions.

Before further research on the behavior of the pavement structure under single and multiple loads can be completed, the material model utilized has to be validated. This was accomplished by utilizing elasto-plastic material models in ABAQUS and comparing predicted values of CBR with measured values (Sukumaran et al [4]). The final goal of this study is to develop a working model of a flexible pavement structure capable of modeling pavement failures caused by heavy loads from taxiing aircrafts.

## BACKGROUND

As stresses and strains are used more and more to predict pavement distresses, and thus the relative condition of the various layers in the pavement structure, the need for consideration of non-linear material behavior becomes increasingly important. Linear elastic approximations of unbound material behavior are no longer acceptable in pavement analysis. The stress state dependency of granular materials, and strain based subgrade soil models must be considered for an accurate estimation of true pavement response (Nazarian and Boddspati [5]).

Past flexible pavement models used multi-layer elastic analysis, which assumes static loading, whereas in reality pavements are subjected to both static and moving loads. However, asphalt mixtures are viscoelastic material and clays exhibit plasticity (Zaghloul and White [6]). The model used in the study conducted by Zaghloul and White [6] incorporated an elasto-plastic model for the base, sub-base and subgrade and a visco-elastic model for the asphalt layer. Zaghloul and White [6] researched the ability of three-dimensional dynamic finite element programs (ABAQUS [7]) to predict the response of moving loads on pavement structures. In their study, the granular material was modeled using the Drucker-Prager model. This assigns elastic properties to materials at low stress levels and plastic properties when the stress level reaches the yield stress. The clay subgrade was modeled using the Cam-Clay model. The validation of their model was accomplished by testing the model’s ability to predict deformations under static and dynamic load conditions. The final results show that their model was capable of simulating truckloads and realistic deformation predictions were obtained.

## FINITE ELEMENT MODEL

ABAQUS [7], a commercial finite element modeling program, has been widely applied for pavement analysis. Chen et al [1] did a comprehensive study of various pavement analysis programs and showed that the results from ABAQUS program were comparable to those from other programs. Zaghloul and White [6] simulated the pavement responses under FWD loading for flexible pavements using three-dimensional dynamic analysis in ABAQUS. The main capabilities of ABAQUS in solving pavement engineering problems include:

- linear and nonlinear elastic, viscoelastic, and elasto-plastic material modeling,
- two-dimensional and three-dimensional calculation,
- static, harmonic dynamic, and transient dynamic loading simulation,
- interface modeling with friction,
- cracking propagation modeling, and
- thermal gradient analysis.

ABAQUS provides many element types that are useful for pavement analysis. An infinite element model can be used to model the infinite boundary conditions in the horizontal and vertical directions in a pavement system. ABAQUS also includes many material models such as linear elastic, viscoelastic, hypoelastic and elasto-plastic models.

## MATERIAL MODEL VERIFICATION

The adequacy of finite element modeling utilizing plasticity models are demonstrated in the following by virtue of their performance in accurately calculating the California Bearing Ratio for a subgrade soil. The subgrade soil utilized for this modeling purpose is the medium strength subgrade used in the construction of a section of the NAPTF facility. The properties of the soil used are shown in Table 1.

Table 1.
Properties of Medium Strength Subgrade Soil

| **Soil Property** | **Values** |
|---|---|
| Moisture content | 30.5% |
| Undrained shear strength | 13.3 psi |
| Dry density | 90.5 pcf |
| Elastic modulus | 12,000 psi |

The finite element mesh used for the analysis is shown below in Figure 1. The finite element analysis is conducted using ABAQUS. A von Mises shear strength idealization is used to model the clay. The von Mises model implies a purely cohesive (pressure independent) soil strength definition. A three dimensional response is simulated using quasi three-dimensional Fourier analysis elements (CAXA) available within ABAQUS. CAXA elements are biquadratic, Fourier quadrilateral elements. The number of elements and nodes in the mesh are 185 and 6260 respectively. CAXA elements were used because of their ability to accurately predict the response of axially symmetric loaded models. They are used to give a simulated three-dimensional response by revolving a two-dimensional surface around the centerline of symmetry. The use of CAXA elements increases the efficiency of the model, when compared to a true three-dimensional model, while still maintaining accurate results.

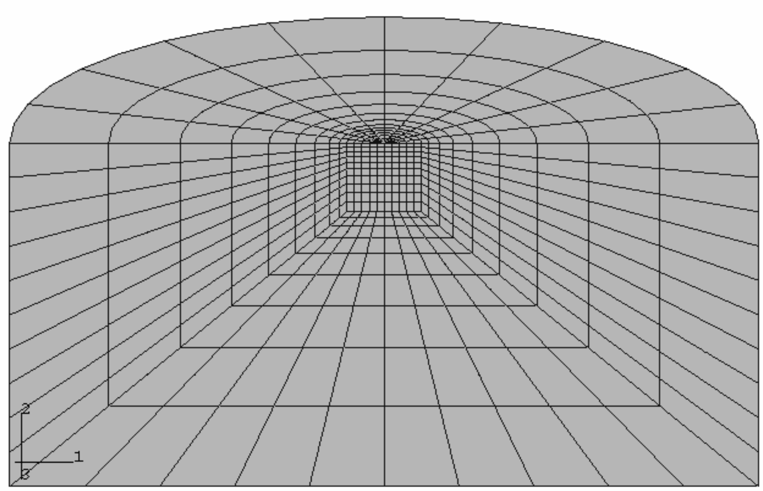
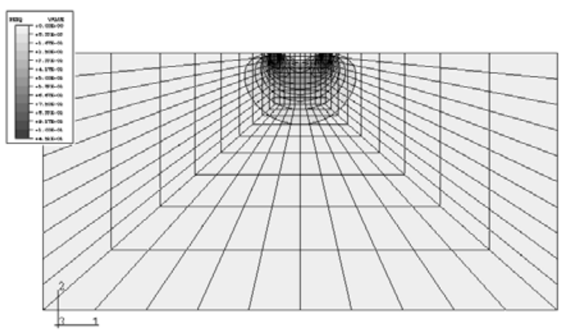


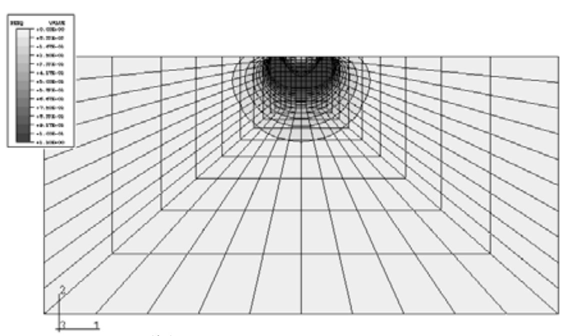

Figure 1: Finite Element Mesh Used in the Verification Study

Sukumaran et. al. [8] showed that a CAXA element is capable of simulating complicated model conditions, normally done utilizing three-dimensional elements, with the actual computational time when compared to three-dimensional models cut down by about 20 %. A second study was conducted by Cho et. al. [2] in which they compared the predictive ability of three different finite element models on pavement structures. They tested a plane strain, an axisymmetric and a three-dimensional model, with the results showing that an axisymmetric model with appropriate element sizes, less then 12.7 mm, can accurately and efficiently model symmetric loading on a pavement structure

To verify the material models, three studies were conducted. The first study utilizes the ultimate shear strength of 13.94 psi as the yield strength. The second study was conducted using the von-Mises model with unconfined compression stress-strain data. Stress-strain response can be better captured if stress vs. strain data from unconfined compression tests, triaxial tests or direct simple shear test are input to obtain the plasticity model parameters. It can be seen from Figure 2 that the zone of plastic strain increases as penetration depth increases as would be expected. The third study conducted utilized the instantaneous elastic modulus, which was calculated from the unconfined compression stress-strain data. This study shows the response from an elastic model. The results show an elastic response with no clear failure plan, as would be expected.



(a)



(b)

Figure 2: Plastic Strain Distribution at a) 0.1” piston penetration
b) 0.2” piston penetration

Table 2 summarizes the results obtained. It can be seen that the von-Mises model utilizing the ultimate shear strength input predicts CBR values that are closer to the higher end of the measured CBR values, while the other two cases predict values closer to the lower end of the CBR values measured.

Table 2.
Results of the Finite Element Verification Studies on the Medium strength subgrade

| **Finite Element Model Utilized** | **CBR values computed** |
|---|---|
| Von-Mises with ultimate shear strength input (Analysis 1) | CBR at 0.1″= 8.6<br>CBR at 0.2″= 5.7 |
| Von-Mises with stress-strain data input (Analysis 2) | CBR at 0.1″= 5.6<br>CBR at 0.2″= 4.8 |
| Elastic model utilizing stress-dependent elastic modulus (Analysis 3) | CBR at 0.1″= 4.2<br>CBR at 0.2″= 4.1 |
| Field measurements (NAPTF test pits, November 1999) | CBR at 0.1″= 3.4-8.4<br>CBR at 0.2″= 2.8-7.2 |

In order to understand the stress-strain response of the soil, stress vs. displacement plots were studied for the three cases mentioned above and compared with the field test data. The stress-displacement plots are shown in Figure 3.

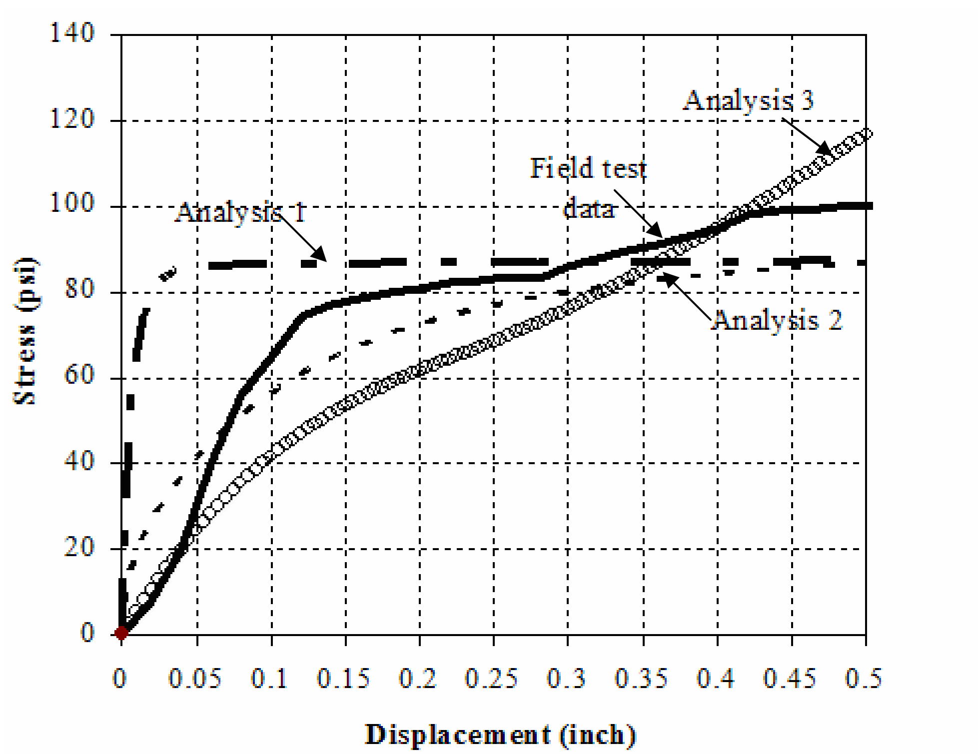



Figure 3: Stress vs. Displacement plot for the various verification studies compared with field test data

The load vs. displacement response computed shows a remarkable similarity to what was observed in the field. Consequently, the prediction of the CBR value also improves. The first study showed a very steep initial slope, which could be attributed to the use of the ultimate shear strength as the yield strength and shows a better prediction for higher values of displacement. The second study clearly shows the best prediction of CBR. The use of stress-strain data allowed for a better-defined yield surface, allowing ABAQUS to model the response of the mesh and fit it to the curve giving the most accurate prediction. The result is a response that fits the field data curve very well for displacements up to 0.35 inches. The third study showed a classical elastic response that results in no defined failure and an almost linear relationship between displacement and stress. The initial slope of the study fit the field data almost exactly and was also very good at predicting later displacements.

From the results, it can be seen that three-dimensional finite element modeling can accurately capture the stress-strain response of the subgrade soil. Based on this conclusion, it was decided to model the subgrade, subbase and base material using the von-Mises material model.

## FEATURES OF FINITE ELEMENT MODEL 1

*Model Geometry*

The mesh comprises of the four layers of the pavement structure as shown in Figure 4, with each layer assumed to be perfectly bonded. The pavement section is comprised of asphalt concrete, crushed aggregate, uncrushed aggregate and cohesive soils. This section is the same as the pavement structure that was tested at the NAPTF's test facility. The thickness of each layer is as follows: 5.12″ of P-401 asphalt surface, 7.88″ of P-209 crushed aggregate for the base layer, 12.13″ of P-154 uncrushed aggregate for the subbase layer and 96″ of Dupont clay, which forms the subgrade.

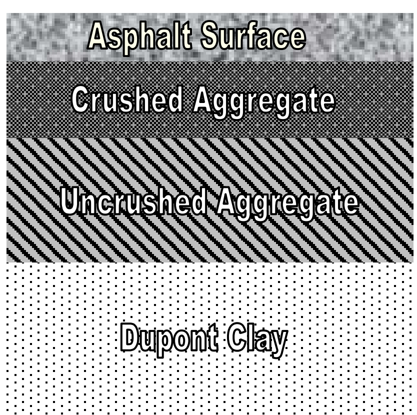



Figure 4: Pavement Cross-section

The finite element mesh developed has the following dimensions; 45 feet in x-direction (length), 10 feet in the y-direction (height), and 60 feet in the z-direction (width) and models the MFC section at the NAPTF facility. The degree of mesh refinement is the most important factor in estimating an accurate stress field in the pavement. The finest mesh is required near the loads to capture the steep stress and strain gradients. The mesh presented in Figure 5 has 21676 nodes, 4526 three-dimensional reduced integration elements and 336 three-dimensional reduced infinite elements. The use of infinite elements allows the displacement and stress fields to decay to zero at infinity (Bettess [9]), providing a good alternative to boundary truncation.

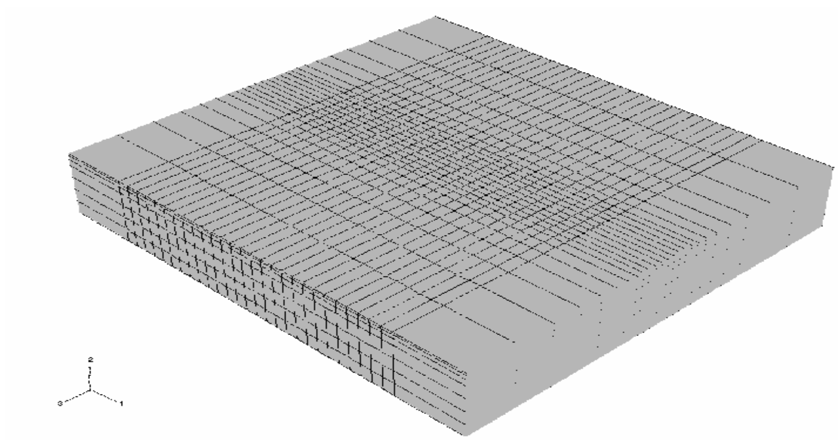


Figure 5: Finite Element Mesh

*Material Properties*

The pavement material is divided into three groups: asphalt mixtures, granular materials, and cohesive soils. Asphalt mixtures were modeled as elastic materials. Granular materials, which consist of base and subbase, are modeled using the Mohr-Coulomb material model. This is an elasto-plastic model in which granular materials are assumed to behave as elastic materials for low stress levels. When the stress level reaches a certain yield stress, the material will start to behave as a plastic material. The yield surface is specified using a friction angle. The medium strength subgrade, Dupont clay is modeled using a von-Mises model. The ultimate shear strength is specified. All the material models also require elastic material properties, which include the specification of the modulus of elasticity and Poisson's ratio.

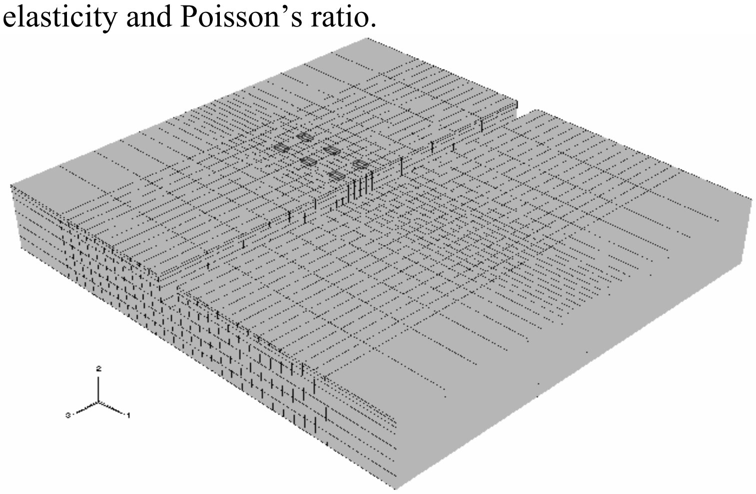


**Figure 6: Finite Element Mesh with Trench**

*Model Analysis*

Since the boundary conditions have a significant influence in predicting the response of the model, the model is constrained at the bottom and on the sides, which do not have infinite elements. In order to verify the model, a static punch test is simulated using the mesh shown in Figure 6 and the results compared with a similar test done at the NAPTF facility. In the test, a 6–wheel configuration is used to apply a static load. The wheel spacing is 54 inches dual (transversely between the wheels) and 57 inches tandem (longitudinally between the wheels) and the closest wheel to the trench is at a distance of 20 inches. The load vs. deflection curve obtained from ABAQUS is compared with experimental data obtained from NAPTF. Pictures of the test performed by the NAPTF and the model created can be seen in Figures 7 and 8, respectively.

Figure 7: NAPTF Static Punch Test

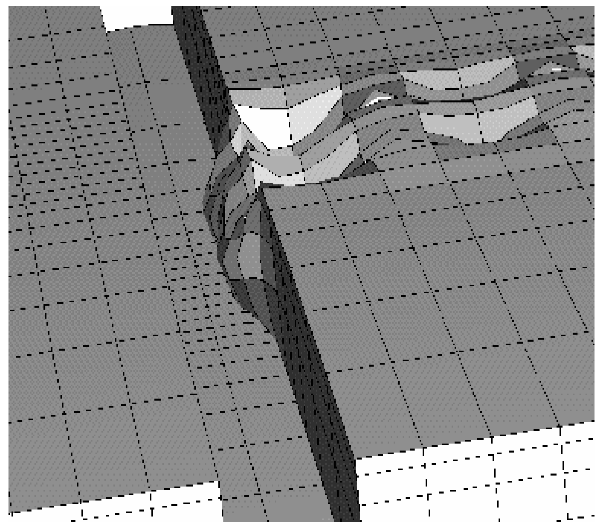


Figure 8: Modeled Static Punch Test

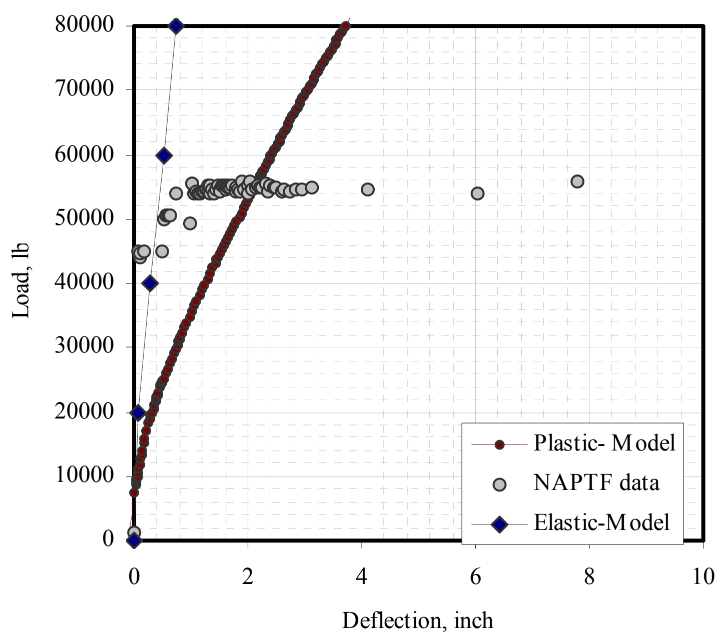



Figure 9: Model Prediction vs. NAPTF Data

The results from the static punch test can be seen in Figure 9. The data from the NAPTF test, which is indicated as module 1-1, module 1-2, and module 1-3 is compared with the values obtained from the finite element model. The two load-deformation plots shown in the Figure are the load vs. deflection values obtained from the finite element analysis of the wheel closest to and furthest from the trench. It can be seen from Figure 9 that the load-deformation response is initially softer. Due to the use of an elastic model for the surface layer, no visible failure load is predicted. However, the deformation pattern in the underlying layers appears to be captured as evidenced from Figures 7 and 8. The figures show that the predictive model is capturing the same type of failure as witnessed during the test at the NAPTF facility. A similar bulging shear zone is seen, close to the point of application of the load, while still keeping the overall trench intact. Since this study will be used mainly to determine the response of the underlying layers rather than the surface layer, the model is suitable for further studies with a moving wheel load.

*Moving-Wheel Model*

After validation of the model under the static punch test, the model was run using one dynamic wheel load, taken as a pressure load moving across the top of the mesh. The most common way of applying wheel loads in a finite element analysis is to apply pressure loads to a circular or rectangular equivalent contact area with uniform tire pressure (Huang [10]). A pressure load equal to 55 kips was applied to the element, which was created to be the same size as the wheel imprint of a large airplane, about 1 foot by 1.7 foot. The contact area was approximated as a rectangle. The rectangular element used to apply the contact pressure was then subjected to a velocity boundary condition of 6 mph to simulate the moving wheel load.

This model was found to require a long computation time and was further refined by removing the infinite elements. The results of further verification studies found that the inclusion of infinite elements is not necessary in achieving accurate results and the mesh could be further simplified by reducing the geometrical size and therefore the number of elements. To determine if the reduction in mesh size resulted in any loss in accuracy, further tests were conducted and are described in the next section.

## FEATURES OF FINITE ELEMENT MODEL 2

The material properties, material model, and pavement layers are kept the same as Model 1, and the only changes made are to the geometry of the model and in exclusion of infinite elements.

*Model Geometry*

The reduced finite element mesh has the following dimensions; 49 feet in x-direction (length), 10 feet in the y-direction (height), and 20 feet in the z-direction (width). The mesh presented in Figure 10 has 16093 nodes and 3360 three-dimensional reduced integration elements.

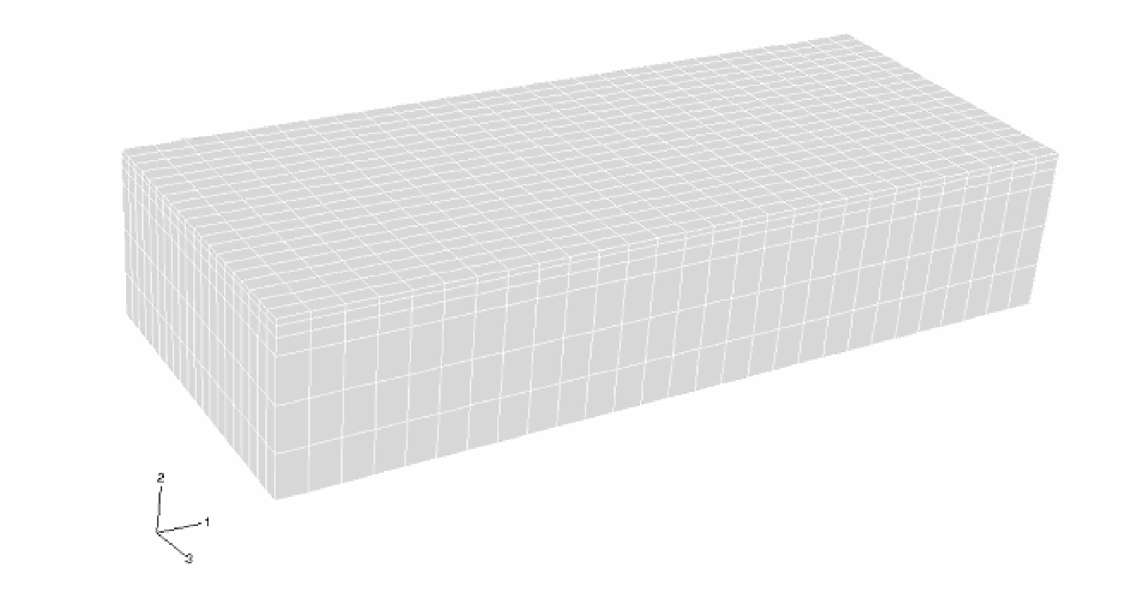



Figure 10: Finite Element Model

The model was further reduced in size by the use of symmetry. By using asymmetric boundary conditions with roller supports on the line of symmetry, the model can deform in the y-direction, while still constrained in the x- and z-direction. This allowed for a drastic increase in overall efficiency.

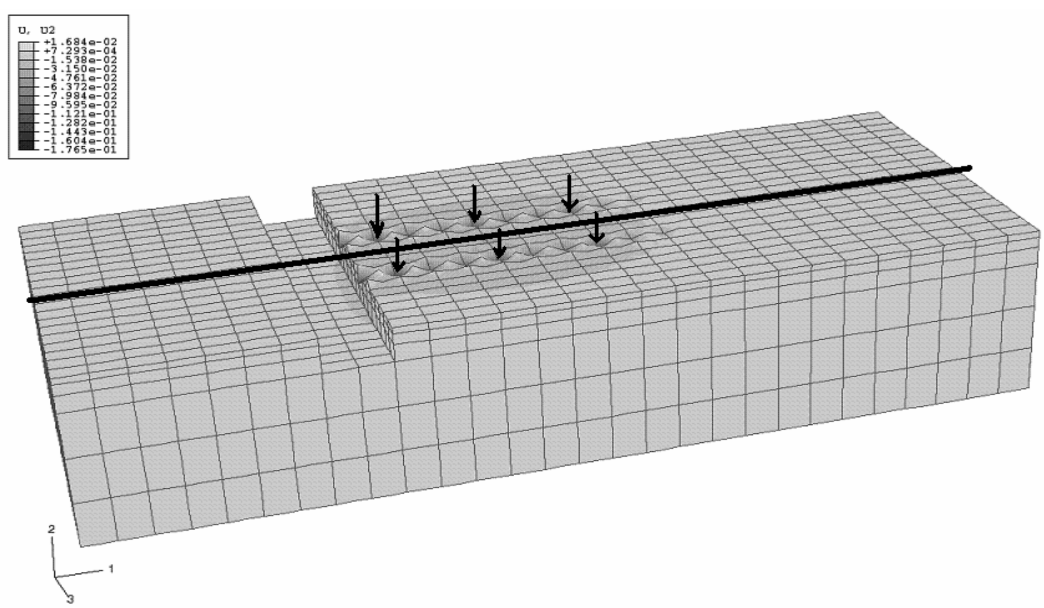



Figure 11: Line of Symmetry

The resulting model had the following dimensions; 35 feet in x-direction (length), 10 feet in the y-direction (height), and 10 feet in the z-direction (width). The reduced model is displayed in Figure 12 and has 6014 nodes and 1200 three-dimensional reduced integration elements.

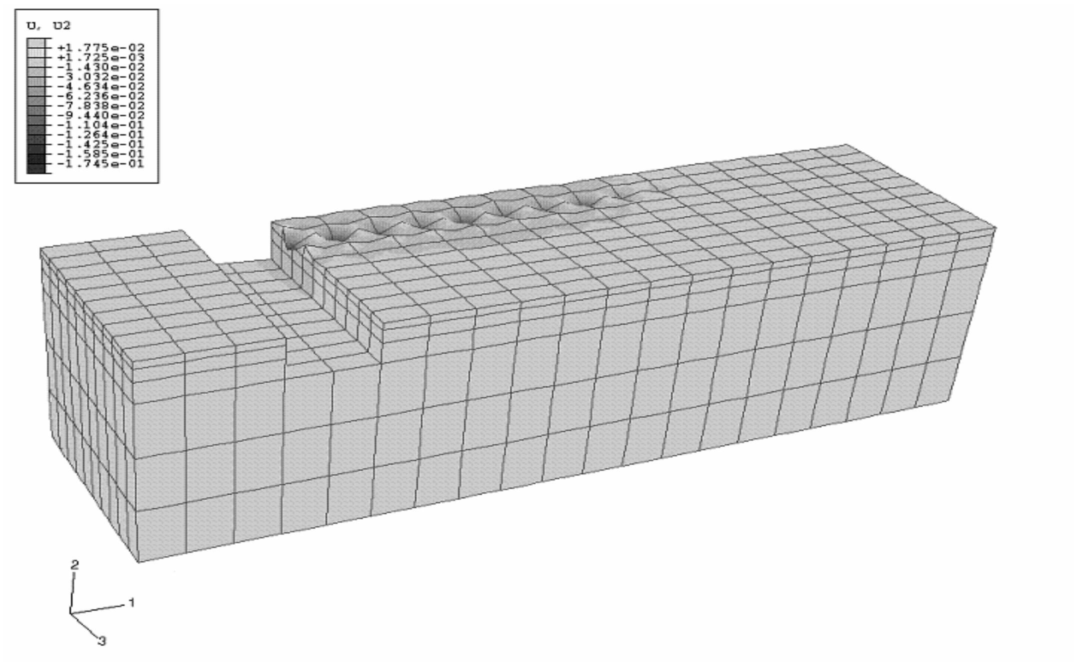



Figure 12: Symmetric Model

The reduced model shown in Figure 12 is then used to simulate the static punch test with elasto plastic models for the base, subbase and subgrade.

Both the reduced mesh and the full mesh were used to predict the response from the static load test on the trench model with elastic material properties. Running the model with linear elastic properties allowed for a quick comparison of the response. The two models compare very well, providing further evidence that the symmetric model was capable of predicting responses just as well as the full model.

Figure 13 shows the response of the model under the static load as compared to the NAPTF's static punch test. Figure 13 shows that the FAA test data at 55 kips recorded about a 1-inch deflection. The symmetric model predicted a similar response showing about the same trend providing further evidence that the reduced mesh was suitable for analysis.

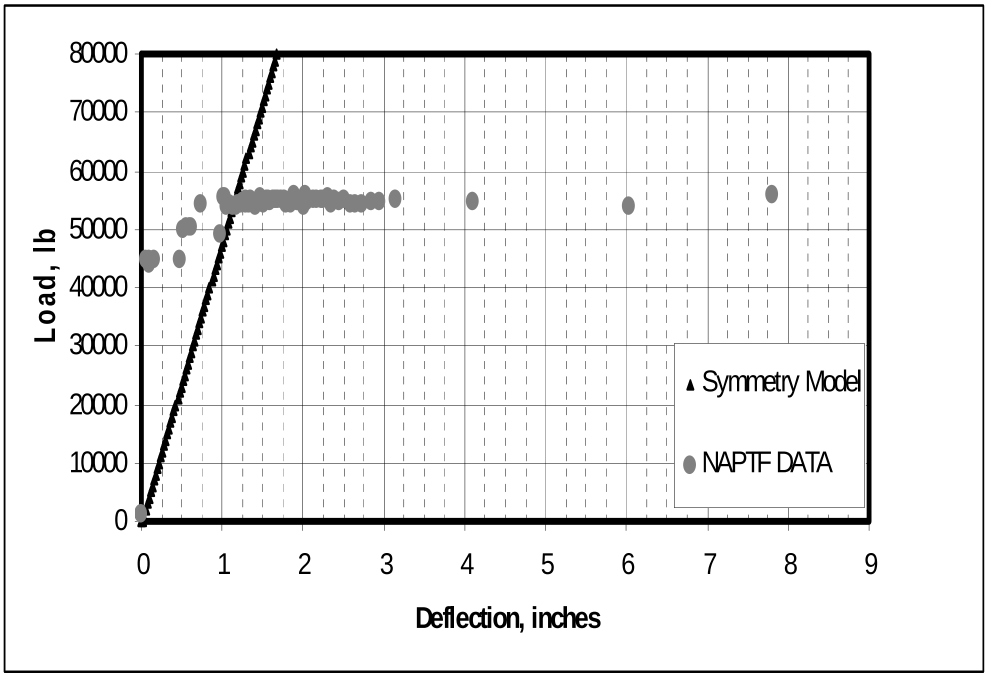



Figure 13: FAA Test Data vs. Symmetric Model

## CONCLUSION

This paper presents the modeling of three-dimensional analysis of pavements. The issues of mesh construction, mesh refinement, element aspect ratios and material non-linearities are discussed. Each of these factors affect the overall time efficiency. For the three-dimensional problems, a careful balance is required to meet the demands of solution time and memory without sacrificing accuracy.

Careful planning of the finite element model is needed to ensure an economical design with accurate results. The easiest design is not always the best, as can be seen by the results shown. The first model was designed to take care of all the needs for this study, the static punch test, moving wheel and moving wheel with wander. From the research that has been conducted it has been shown that making a single mesh to complete drastically different tasks will lead to a model that is highly impractical.

**REFERENCES**


1. Chen, D. H., Zaman, M., Laguros, J.Soltani, A., Assessment of computer programs for analysis of flexible pavement structure, Transportation Research Record 1482, TRB, National Research Counsel, Washington, D., C., 1995, 123-133.
2. Cho, Y-H., McCullough, B.F., and Weissman, J., Considerations on finite element method application in pavement structural analysis, Transportation Research Record 1539, TRB, National Research Counsel, Washington, D., C., 1996, 96-101.
3. Kuo, C. M., K. T., and Darter, M. I. Three dimensional finite element model for analysis of concrete pavement support, Transportation Research Record 1505, TRB, National Research Counsel, Washington, D., C., 1995, 119-127.
4. Sukumaran, B., Kyatham, V., Shah, A., Sheth, D., "Suitability of Using California Bearing Ratio Test to Predict Resilient Modulus," FAA Technology Transfer Conference, 2002
5. Nazarian, S. and K. M. Boddspati, "Pavement-Falling Weight Deflection Interaction Using Dynamic Finite Element Analysis," In Transportation Research Record 1449, TRB, National Research Council, Washington, D., C., 1995, 123-133.
6. Zaghloul, S.M., and T.D., White, "Use of a Three-Dimensional, Dynamic Finite Element Program for Analysis of Flexible Pavement," In Transportation Research Record 1388, TRB, National Research Council, Washington, D., C., 1993, 60-69.
7. HKS, ABAQUS Theory and Users Manual – Version 6.3, Hibbit, Karlsson & Sorenson, Inc., Pawttucket, Rhode Island., 2002
8. Sukumaran, B., McCarron, W.O., Jeanjean, P., and Abouseeda, H. (1999), "Efficient Finite Element Techniques for Limit Analysis of Suction Caissons under Lateral Loads," *Computers and Geotechnics*, Vol. 24, No. 2, pp 89-108
9. Bettes, P., Infinite elements, Penshaw Press, Sunderland, U.K., 1992.
10. Huang, Y.H., Pavement Analysis and Design, Prentice Hall, Englewood Cliffs, New Jersey, 1993.