بسم الله الرحمن الرحيم

COMPARATIVE STUDY OF MECHANISTIC BEHAVIORAL ANALYSIS FOR FLEXIBLE PAVEMENT USING FINITE ELEMENT MODELLING

Md. Shariful Islam

M Irteza Alam

Shadab Morshed Khan

Department of Civil and Environmental Engineering (CEE)
Islamic University of Technology (IUT)
2022

PROJECT APPROVAL

This is to certify that the thesis entitled **"Comparative Study of Mechanistic Behavioral Analysis for Flexible Pavement Using Finite Element Modelling"** submitted by Md. Shariful Islam, M Irteza Alam, and Shadab Morshed Khan has been approved as partial fulfillment of the requirement for the Degree Bachelor of Science in Civil Engineering at Islamic University of Technology (IUT).

Supervisor

Dr. Nazmus Sakib

Assistant Professor

Department of Civil and Environmental Engineering (CEE)

Islamic University of Technology (IUT)

Board Bazar, Gazipur-1704, Bangladesh

DECLARATION

We hereby certify that the undergraduate research work reported in this thesis has been performed by us under adept supervision of Dr. Nazmus Sakib. Appropriate precautions have been taken to ensure that the work is original. This work has not been plagiarized and submitted elsewhere for any other purpose (except for publication).

________________________	________________________
Md. Shariful Islam	M Irteza Alam
Student ID: 170051006	Student ID: 170051042
Date:	Date:

____

Shadab Morshed Khan

Student ID: 170051073

Date:

DEDICATION

We dedicate this thesis to our parents, who have sacrificed their valuable time, livelihood, and effort over many years to ensure that we could be who we are today. They have motivated and encouraged us to fulfill our engineering ambitions without ever looking back. We shall be eternally grateful to them.

We would also like to express gratitude to our supervisor Dr. Nazmus Sakib for his continuous support and inspiration, without whom this work would not be possible.

ACKNOWLEDGEMENTS

"In the name of Allah, the Most Gracious, the Most Merciful."

All the praises to Allah (SWT) for giving us the opportunity and strength to complete our thesis work including this book successfully. We would like to express our earnest gratitude to our parents who have given us constant support and encouragement.

We would like to convey our appreciativeness to our supervisor, Dr. Nazmus Sakib, Assistant Professor, Department of Civil and Environmental Engineering, Islamic University of Technology, for his adept advice and continuous encouragement. His technical and editorial contribution was necessary for the completion of this academic project. Without his guidance this paper would not be completed.

Furthermore, we place on record, our sense of gratitude to One and All, who directly or indirectly, have contributed to this venture.

Abstract

Year after year, the number of heavy trucks on Bangladesh's highways increases. Heavy loads are one of the most typical causes of pavement distress. The distribution of contact stresses between tires and the pavement surface has a big impact on how pavement distresses start and spread. Hence, to accurately and exactly describe pavement responses, the distribution of contact stresses ought to be 1st investigated thoroughly. Pavement is a complicated structure made up of numerous layers of various materials that determine how it behaves under stress. Finite Element Modelling (FEM) is becoming a more prominent tool for researchers to solve complex structural mechanics problems in engineering. There is currently a shift toward more mechanistic design methodologies in pavement analysis to reduce the limits in calculating stress, strain, and displacement. We used ABAQUS software to perform flexible pavement finite element analysis in our study. This research aims to learn more about the stress-strain characteristics of asphalt pavement by examining a variety of traffic and loading parameters, as well as their relationships with pavement distress and failure initiation. For validation, we compared our ABAQUS results to IITPAVE.

KEYWORDS: Finite Element Analysis, ABAQUS, Flexible Pavement, Meshing, Static Analysis, Tire Imprint Area, Numerical Analysis

Table of Content

Table of Figures

Table of Tables

Chapter 1 Introduction

A highway is a large facility that facilitates mobility within a country, assists commuters in reaching their various destinations, and invariably affects a country's financial situation. In terms of the entire land area of Bangladesh, there are currently 6-7 percent of roadways, with asphalt covering more than 98 percent of them (*Bangladesh - The World Factbook*, n.d.). Despite land constraint, demand for road pavement is rising rapidly to meet basic transportation needs and boost economic activity. As a result, there is significant pressure for investment in the transportation sector. However, the unit length of pavement construction cost is gigantic according to empirical design guidelines since the complicated interaction of materials in different layers cannot be simply analysed and explored. Engineers have come up with a variety of methods to investigate pavement structures over the years. Each technique has its advantages and disadvantages. Structural failure can be caused by a variety of factors, including the materials utilized, the maintenance culture, the materials used for construction, the system, the incorrect front-end style approach, and the construction methodology, to name a few. These concerns have an impact on the amount of road service provided and cause them to not last as long as they should.

A cost-effective and efficient pavement system can be created utilizing mechanistic behavioural analysis and finite element analysis. A suitable tool for modelling and evaluating many sorts of structures is the finite element method. This technology gives a framework for resolving complicated pavement structure problems(Ahirwar & Mandal, 2017). The finite element method's key advantages are its applicability and flexibility for analysing various boundary conditions and material attributes.

In our research, we used ABAQUS software to perform flexible pavement finite element analysis. This research aims to learn more about the stress-strain characteristics of asphalt pavement by examining a variety of traffic and loading parameters, as well as their relationships with pavement distress and failure initiation. For validation, we compared our ABAQUS results to IITPAVE.

Chapter 2 Literature Review

(Azimi et al., 2016) conducted a research on extended finite element model and used linear elastic fracture criteria for justification of numerical outcomes from the simulation and also discussed the impact of optimum element size in fracture characteristic in various load cycles. And concluded with the best mesh sensitivity and the optimum value of element size is not always available due to high computational cost.

(V.Chaudhari & A. Chakrabarti, 2012) researched on 3D model of a concrete cube using smeared crack model and concrete damage plasticity approach. And one of their findings is that for 3D model of concrete cube, smeared crack model is more suitable as it gives desired results at coarser mesh size but for damage plasticity model finer mesh is necessary to obtain actual desirable value.

(Mo et al., 2008) conducted a research on 2D and 3D meso-scale finite element models for ravelling analysis of porous asphalt concrete and concluded that for finite element analysis 3D model tends to be loaded more heavily compared to the equivalent 2D representation and analysed stresses are much higher for porous asphalt concrete.

(Rahman MT et al., 2011) on a research used the information that tire imprint area is needed to be rectangle with two semicircles at both sides. Circular, rectangular or ellipsoid tire contact area is not appropriate because they generate fewer amounts of stresses and strains for the equal area.

(Maqbali & Ragab, 2021) conducted a study on various free and commercial software used in road designing to solve the problems of road designing and eradicate road catastrophes.

(Shashikant & Bindu, 2015) researched on the impact of mesh size on the correctness of numerical analysis findings are investigated in this research. These findings are used to develop suggestions for selecting the best mesh method for finite element modelling.

Chapter 3 Objectives

The specific objective of this research is to

- Develop proper element type and effective meshing strategies for flexible pavement in finite element software.
- Analyse proper tire imprint area for inflation pressure.
- Analyse stress-strain characteristics of pavement.
- Validating the model by comparative analysis.

Chapter 4 Methodology

For this thesis project, at first a multi-layered flexible pavement structure was selected consisting of three layers on top of the natural subgrade. Multi layered pavement structure was chosen as the stress is distributed among the top layers and as stress passes through each layer vertical and horizontal strain reduces resulting minimum stress on the subgrade which gives very little deflection. As Abaqus FE modelling software is used for modelling the pavement the model geometry was created in the finite element modelling software. Then all the layers were given proper mechanical properties. After that the loading was defined and magnitude and amplitude of load was given as tire load mainly creates the stress on pavement. As finite element analysis depends on mathematical equations, dividing the total system into very small pieces and solving complex mathematical equations to derive results (Bathe, 2016). The process of dividing the system into smaller pieces is called meshing. After analysis the results are compared with results from IITPAVE to validate the results. IITPAVE is chosen as in India and Bangladesh this software is used for most of the design purpose and deemed to be accurate (IRC, 2018). If the results were not satisfying another mesh was created and comparison was done again until a satisfying result was not found. The workflow is shown below in the flowchart.

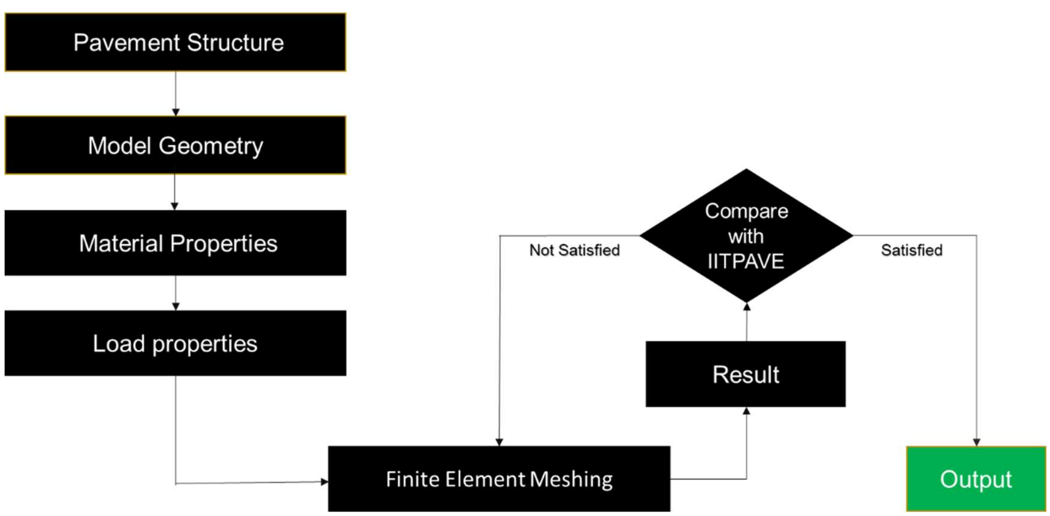

Figure 1 : Flowchart of Methodology

4.1 Pavement Structure

The pavement structure selected for the study is a multi-layered flexible pavement. The layers selected for the pavement structure is Asphalt Surface Layer, Base layer, Sub-Base Layer and Natural Sub-Grade.

On top there is asphalt concrete layer which is given and it is directly in contact with external forces due to tire loading and also weather impacts. Repetitive heavy tire load and friction with tire causes decay of the asphalt layer and it is there to protect the bottom layers from this decay as recoating the asphalt layer is very cost effective.

The asphalt surface layer is exposed directly to traffic loads and weather changes. With the passing of heavy vehicles, tangential stress occurs on the contact surface between the tires and the asphalt layer(Hou et al., 2018). Directly below the asphalt layer there is the base layer. This layer is mostly composed of various aggregates, stone and slag. As this layer is comprised mostly of coarse grains this layer is called granular base. The main objective of this layer is to reduce excessive stresses and impacts of changes in temperature of the environment on the subbase or else the subbase might collapse. Under the granular base layer is subbase layer. It transfers stress from base to subgrade and is an extension to base layer. It consists of same types of materials used for base layer but the quality of materials is lower. It also acts as a seal and stops water from reaching the subgrade for this reason the subbase must be compacted properly. This layer can be omitted where the subgrade is of good quality. At the bottom of all the pavement layers is natural subgrade. It is mainly the existing soil where the pavement is constructed. The design required for the subgrade is dependent on the quality of subgrade. Most research work shows that moisture content of the existing soil and traffic load on top of the pavement is the main controlling factor which determines the longevity of a flexible pavement structure. High moisture content in subgrade results in rapid deterioration of pavement which is found in clay soil and to address this situation replacing or adding sand to the soil can improve the lifespan of the pavement greatly (Hassan et al., 2004).

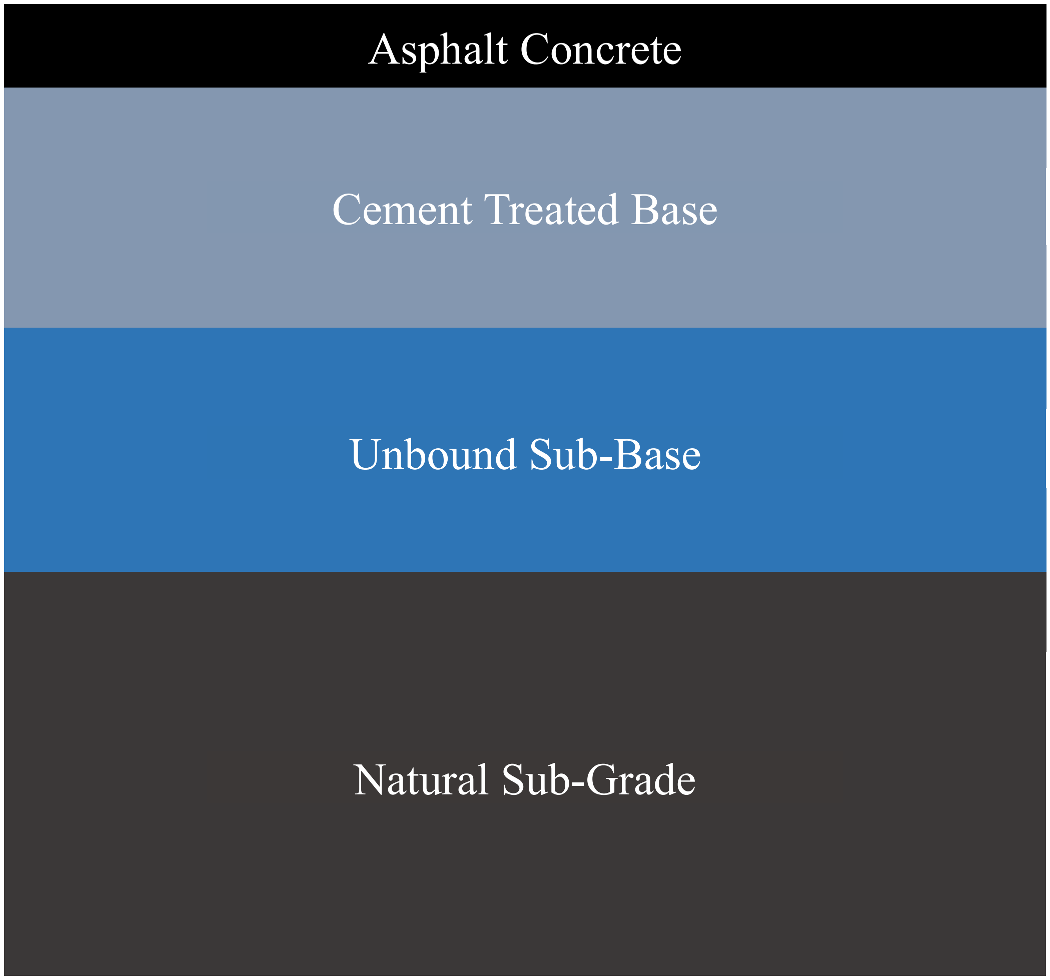

Figure 2 : Multi-Layered Flexible Pavement Structure

4.2 Model Geometry

The analysis was run using the finite element modelling software ABAQUS 2021 and at first step the pavement structure was recreated within the software. The model was created using 3D modelling space using deformable part type and solid shape base feature. The base dimension of horizontal and vertical was set to be 10 m by 5 m. The loading will be at the centre of the model and the analysis zone is expected to be very small but to simulate infinite long road and circumvent boundary error value this huge model was created. The base was extruded by 2 m to create the 3D pavement model. The extrusion was then sectioned to simulate different layers of pavement the sections were made so that asphalt layer is of 100 mm, base layer is of 300 mm and subbase is of 300 mm and the rest is subgrade. The subgrade is also kept large to simulate real world scenario (Alkaissi, 2020). The boundary conditions are set in a way that vertical and horizontal displacement of the subgrade is restricted.

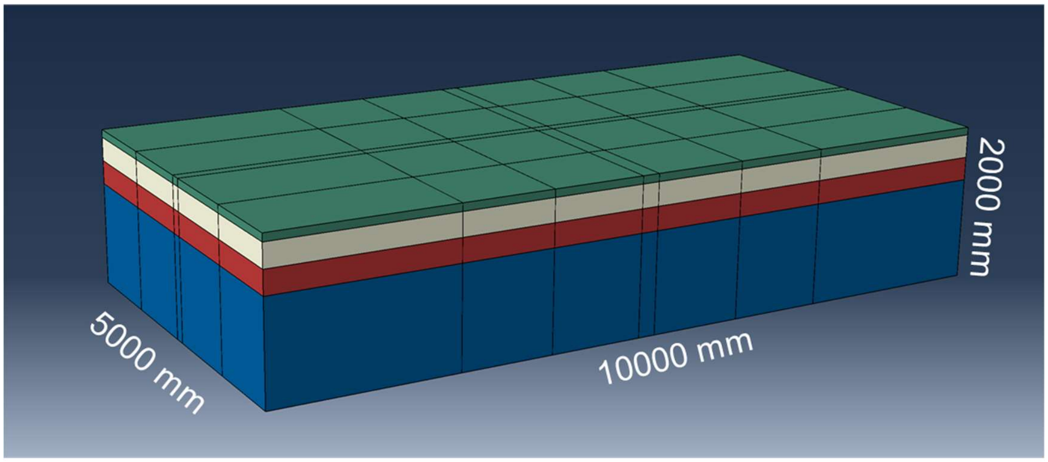

Figure 3 : Multilayered Pavement Modeled in ABAQUS

4.3 Load Properties

Simulating traffic load properly is very much complex. So, simplifying the process, we took the load as a tire pressure on a pavement area which is the contact area of the pavement with the tire. The pressure represents the vehicle load. The standard axel load was set to be 20 kN and contact pressure was set as 0.56 MPa (IRC, 2018). The loading amplitude is set so that the pressure starts from zero and linearly increases to the maximum pressure.

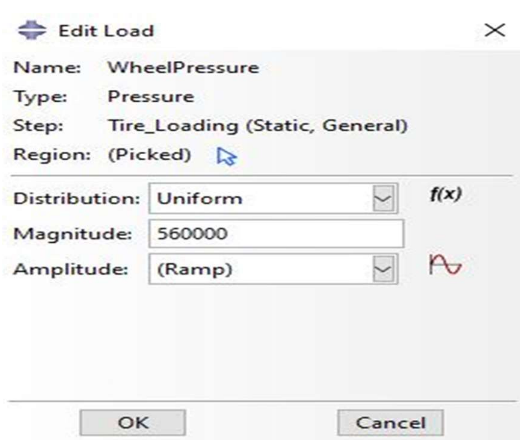

Figure 4 : Loading in ABAQUS

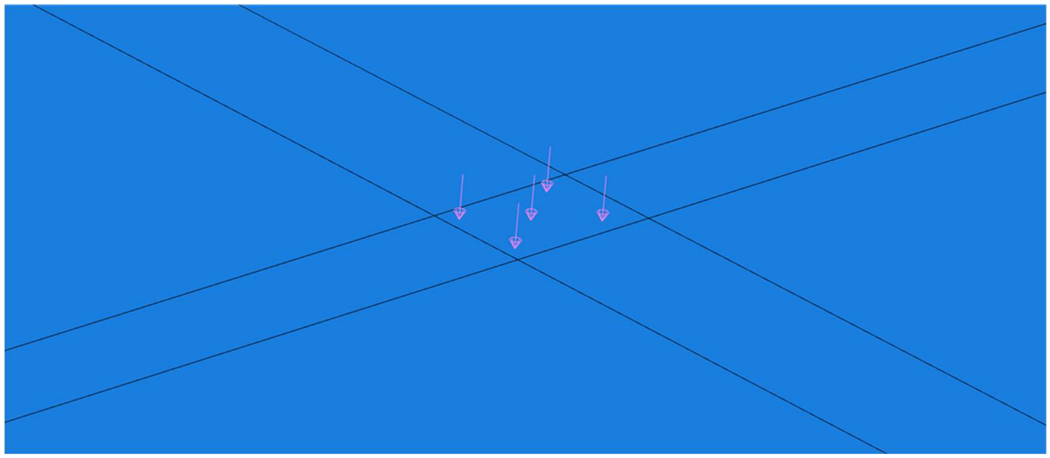

Figure 5 : Traffic Load Shown on The Model

4.4 Contact Area

Properly simulating the contact area is very complex. The true shape of the contact area is an ellipse. The ellipse shape can be approximated by using a rectangle and two semi-circles consisting a length of L and width of 0.6L.This shape can be further simplified as a rectangle occupying an area of 0.5227 L^2 and width of 0.6 L (Yang H. Huang, 2004).

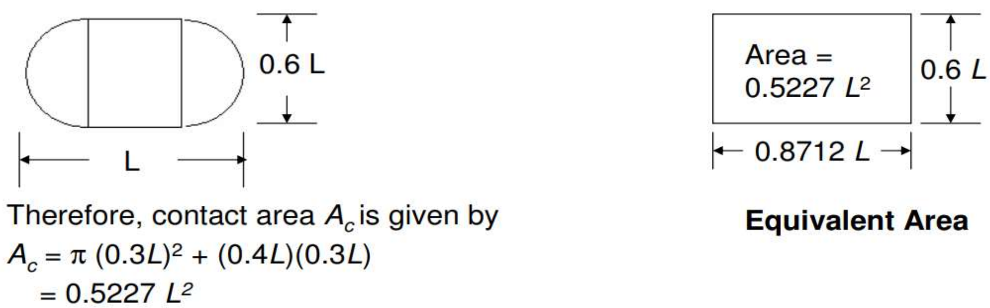

Figure 6 : Approximating the Contact Area

So, $A_c = 0.5227\ L^2$

$$L = \sqrt{\frac{A_c}{0.5227}}$$

And the contact area can be obtained from load and inflation pressure

$$A_c = \frac{p}{p_i}$$

Using the previously mentioned axel load and inflation pressure the value of L is calculated to be 0.261 m.

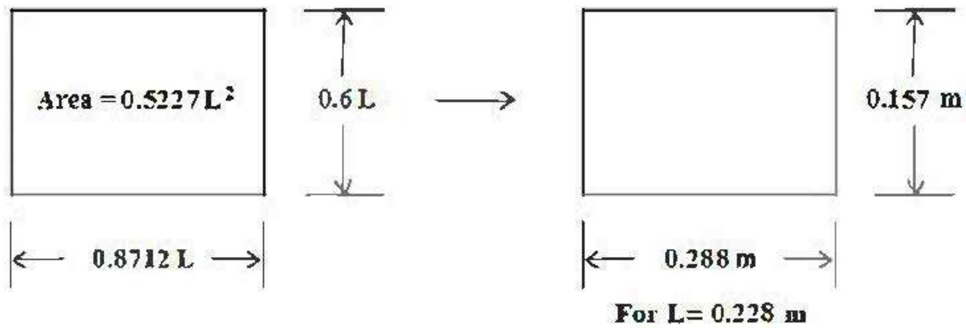

Figure 7 : Determining the Contact Area

So, the contact area can be defined using a length of 0.288 m and a width of 0.157 m.

4.5 Material Properties

The pavement structure is of three layers - asphalt surface, granular base and subbase which is on top of a natural subgrade. The pavement constituents were expected to respond linearly and elastically to the traffic load as it is simulated through a static load with linear amplitude growth. Elastic properties such as modulus of elasticity and Poisson's ratio of the materials were obtained from previous investigation(Gupta & Kumar, 2015).

Table 1 : Properties of Flexible Pavement Layer

Layer	**Modulus of Elasticity (MPa)**	**Poisson's Ratio**	**Thickness (mm)**
Asphalt	229.8	0.35	100
Base	114.9	0.3	300
Sub-Base	46	0.3	300
Sub-Grade	5.74	0.4	∞

4.6 Finite Element Meshing

Finally, to analyse the impact of load on the pavement structure meshing is done to divide the part into fine elements. The meshing is done in such a way that the analysis area has the finest element mesh and the further away from the loading point the less elements are there as the stress due to the load is not supposed to distribute this far. The meshing element is set to be 8-node linear brick reduce integration elements which is in short called C3D8R to reduce rate of convergence.

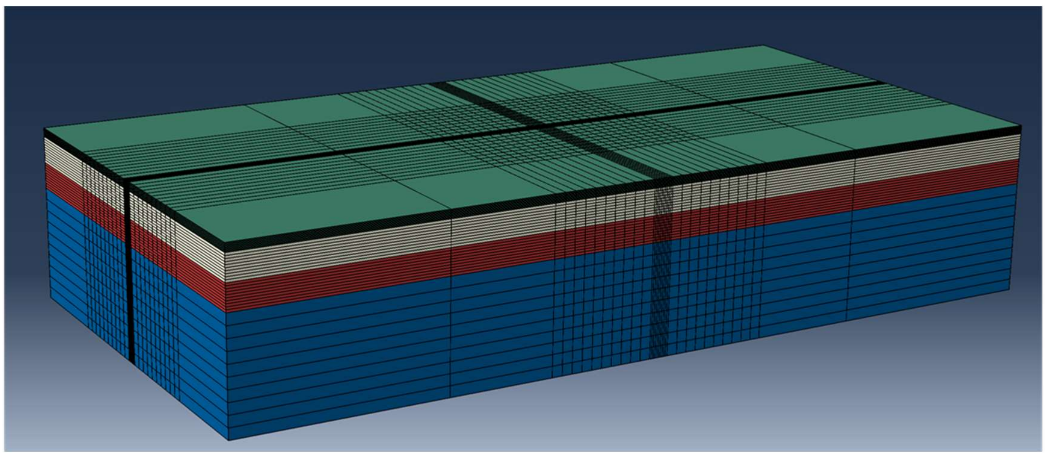

Figure 8 : Meshing of the Pavement Model

Chapter 5 Analysis

A mesh sensitivity study is the most scientific method for establishing the optimum mesh resolution required for accurate results. For that, we Simply run our model using different mesh sizes and plot the stress values of that result. We simulated a multilayer pavement using five different meshes to illustrate mesh sensitivity analysis. The various elements that we used in our analysis are listed below

Table 2 : Element Numbers of Different Meshes

Analysis Number	Total Number of Elements
1	9021
2	21120
3	43520
4	57720
5	73920

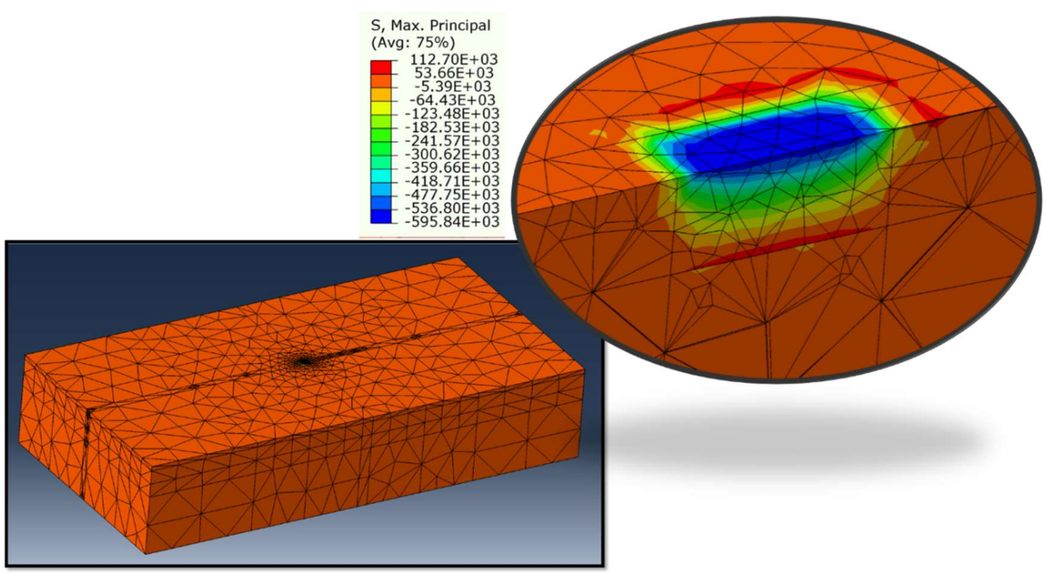

Figure 9 : Maximum Principle Stress for Analysis 1

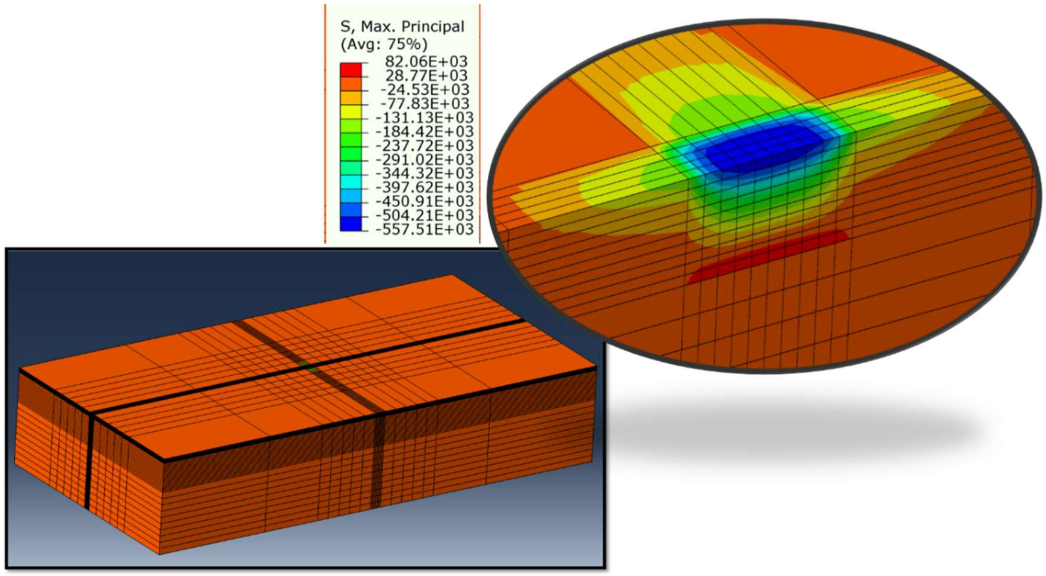

Figure 10 : Maximum Principle Stress for Analysis 2

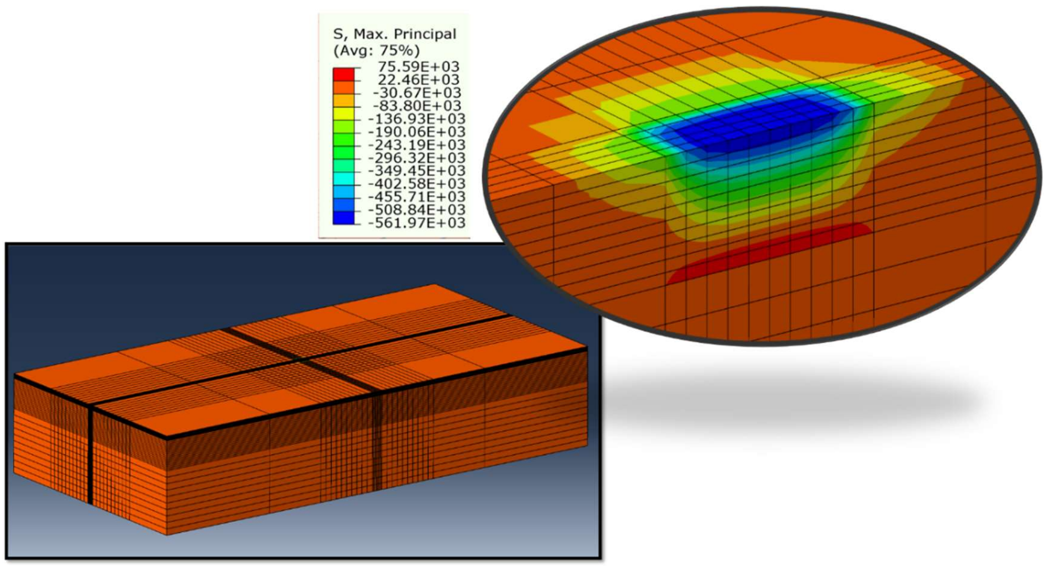

Figure 11 : Maximum Principle Stress for Analysis 3

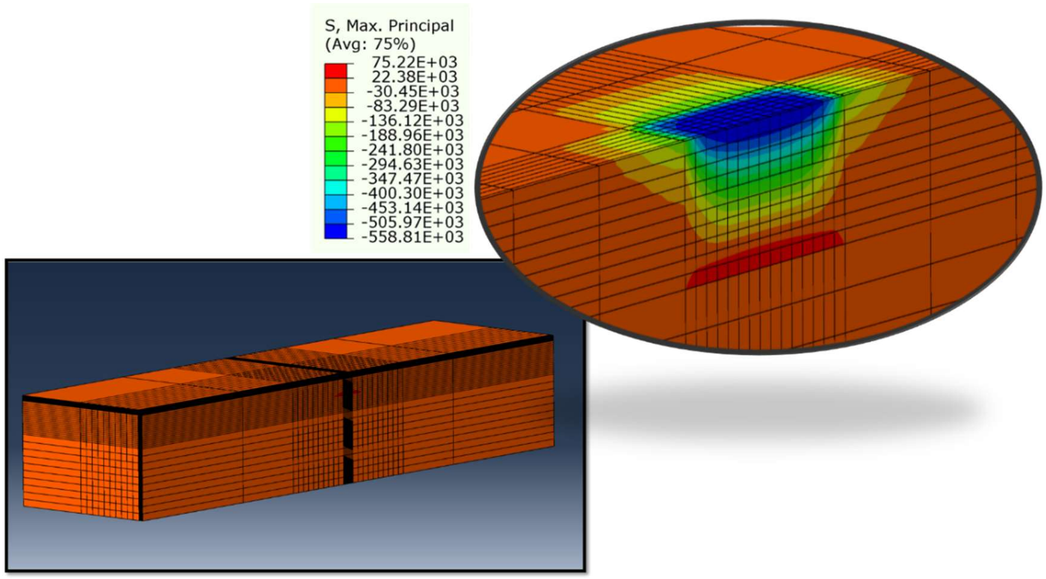

Figure 12 : Maximum Principle Stress for Analysis 4

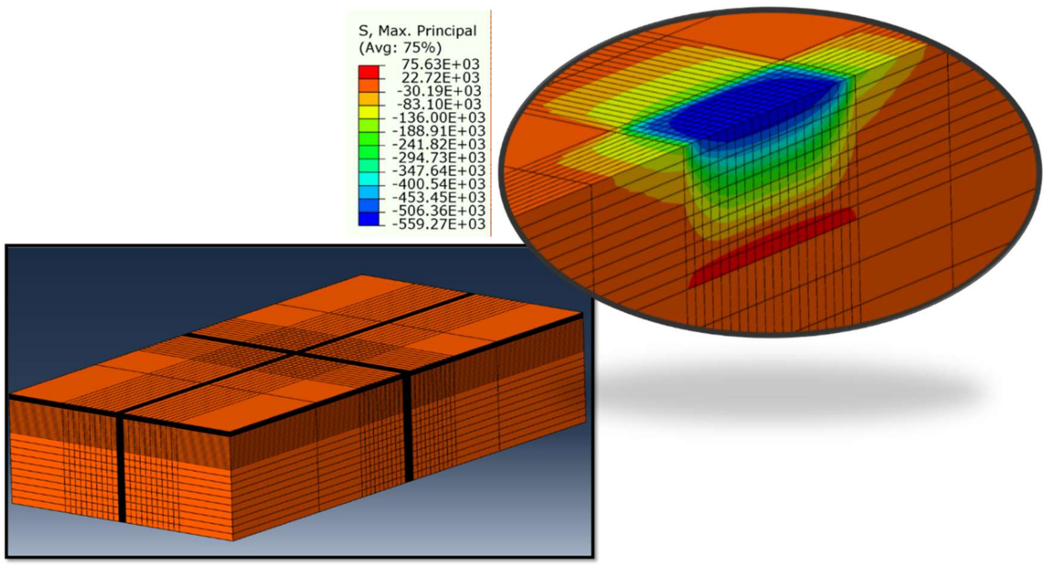

Figure 13 : Maximum Principle Stress for Analysis 5

We have also performed our analysis in IITPAVE to validate our results.


```
No. of layers              4
E values (MPa)          229.80  114.90   46.00    5.74
Mu values                 0.350.300.300.40
thicknesses (mm)        100.00  300.00  300.00
single wheel load (N) 20000.00
tyre pressure (MPa)       0.56
Single Wheel
  Z         R      SigmaZ      SigmaT      SigmaR      TaoRZ       DispZ      epZ         epT         epR
100.00     0.00-0.3236E+00 0.7665E-01 0.7633E-01 0.0000E+00 0.1630E+01-0.1641E-02 0.7101E-03 0.7082E-03
100.00L    0.00-0.3236E+00-0.2219E-01-0.2235E-01 0.0000E+00 0.1630E+01-0.2700E-02 0.7101E-03 0.7082E-03
400.00     0.00-0.2849E-01 0.4487E-01 0.4521E-01 0.0000E+00 0.1297E+01-0.4832E-03 0.3468E-03 0.3507E-03
400.00L    0.00-0.2848E-01 0.1064E-01 0.1078E-01 0.0000E+00 0.1297E+01-0.7589E-03 0.3468E-03 0.3506E-03
700.00     0.00-0.4908E-02 0.2171E-01 0.2149E-01 0.0000E+00 0.1157E+01-0.3885E-03 0.3639E-03 0.3576E-03
700.00L    0.00-0.4908E-02 0.1877E-03 0.1702E-03 0.0000E+00 0.1157E+01-0.8800E-03 0.3629E-03 0.3586E-03
```

Figure 14 : Analysis Results from IITPAVE

Chapter 6 Results

6.1 Analysis Results for Different Types of Mesh

Total 5 analysis were conducted for our multi-layered pavement structure using different meshing. The results for each meshing are shown in the table below.

Table 3 : Stress value at bottom of each layer for different meshing

Total Elements / Layers	9021	21120	43520	57720	73920
Asphalt Layer Stress (MPa)	0.099	0.081	0.076	0.075	0.076
Base Layer Stress (MPa)	0.044	0.034	0.039	0.039	0.039
Sub-Base Layer Stress (MPa)	0.021	0.017	0.019	0.019	0.019

6.2 Optimum Mesh Deduction

ABAQUS is a finite element modelling software which analyses using linear and quadratic equations for each element. Fine meshing is necessary to get accurate results but on the other hand it takes a lot of computational power to solve equations for a large number of elements. This makes it a necessary evil. So, to minimise this problem an optimum mesh size must be used to get results that are almost accurate as well as has the lowest number of elements. Plotting the stress vs element number graph, it can be observed that with increasing number of elements the stress values become higher and accurate but after 43520 elements the stress difference is almost negligible. So, it can be said that the third meshing is the most optimum mesh.

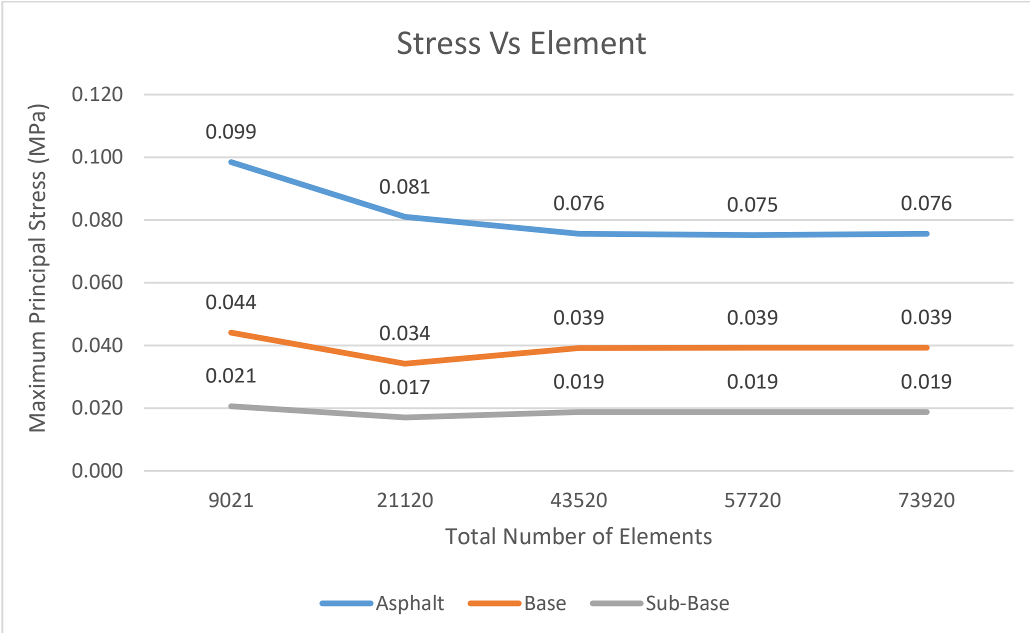

Figure 15 : Stress vs Total Elements Graph of Each Layer

6.3 Vertical Stress-Strain Values for Different Layers

Using the optimum meshing the vertical stress-strain values are computed for the multi-layered pavement structure. Examining the figure for stress values, it can be said that the stress is decreasing linearly as it is within its elastic range.

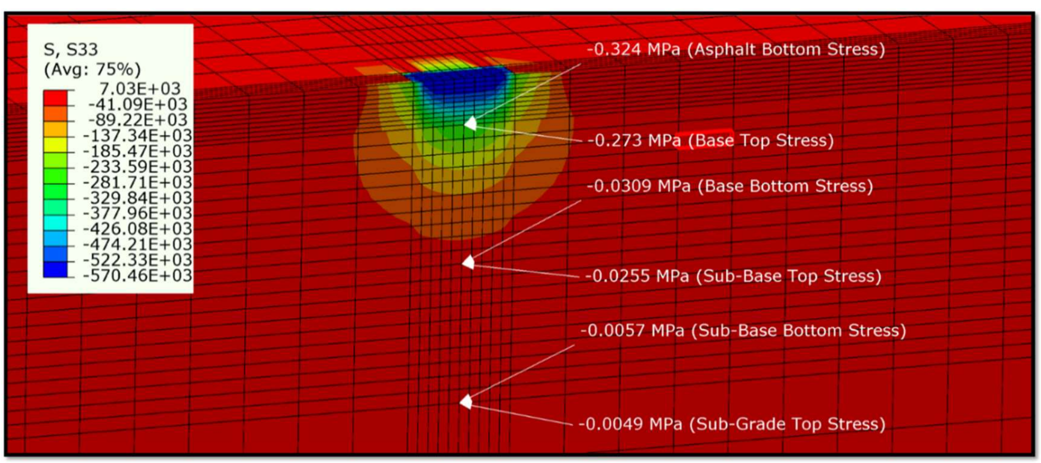

Figure 16 : Vertical Stress for Different Layers

But examining the figure for strain, the values decreases within a layer then increases for the next layer and then again decreases repeating this process. This is also natural as the stress decreases but each layer has different modulus of elasticity resulting variation in strain.

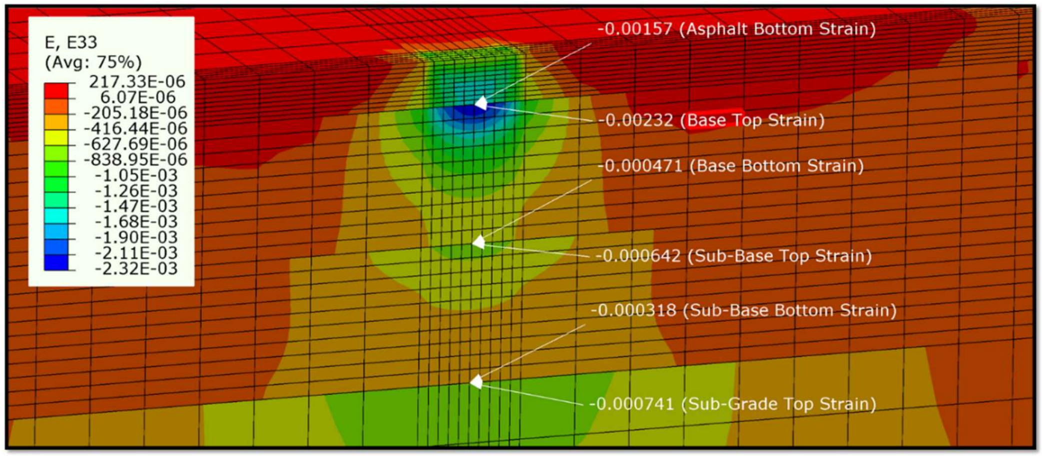

Figure 17 : Vertical Strain for Different Layers

All in all, it can be said that our pavement model analysis shows results which can be backed up by real life phenomenon.

6.4 Stress vs Strain Graph of Different Layers

Plotting the stress-strain values collected from ABAQUS and IITPAVE for different layers it can be seen that the graphs almost matches one another which also validates our results of optimum mesh.

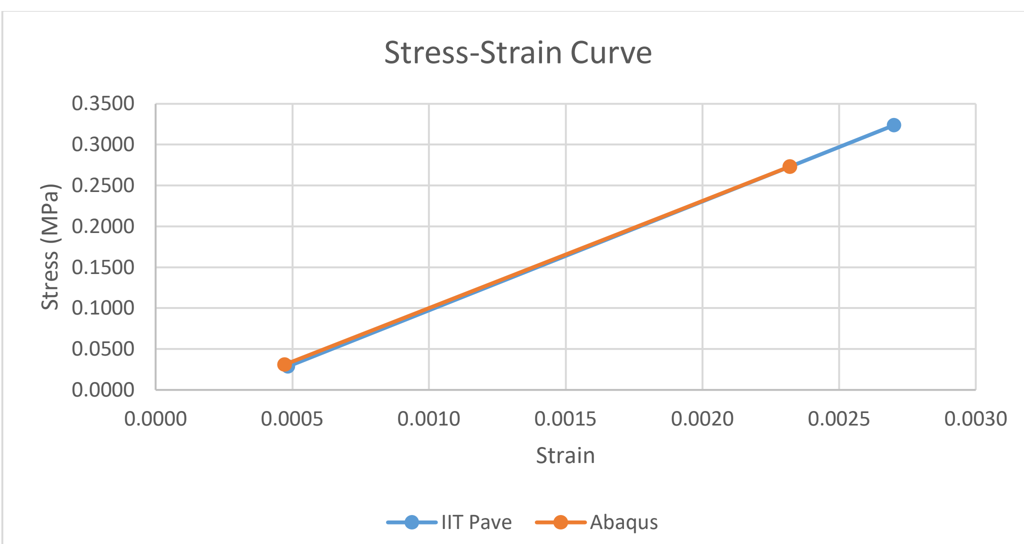

Figure 18 : Stress vs Strain Graph for Base Layer

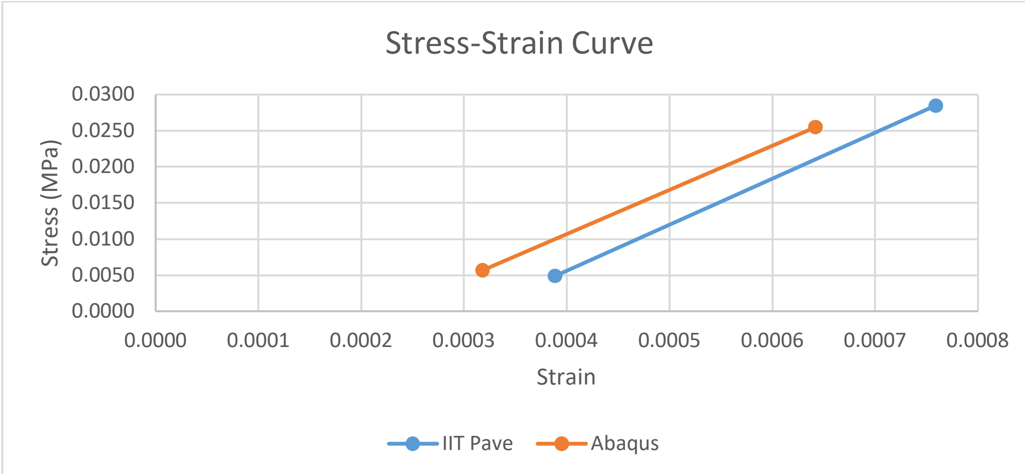

Figure 19 : Stress vs Strain Graph for Sub-Base Layer

6.5 Comparison Between ABAQUS and IITPAVE

The difference between stress-strain values collected form ABAQUS and IITPAVE are shown in percentage in the table below. Here the maximum difference is of 18.15% and the minimum difference is of 0.12%. As all the stress strain values are very small errors due to rounding cannot be excluded.

Table 4 : Comparison Between ABAQUS and IITPAVE

IITPAVE		**ABAQUS**		**Difference**	
Stress (MPa)	**Strain**	**Stress (MPa)**	**Strain**	**Stress (%)**	**Strain (%)**
0.3236	0.0016	0.3240	0.0016	0.12	4.33
0.3236	0.0027	0.2730	0.0023	15.64	14.07
0.0285	0.0005	0.0309	0.0005	7.80	2.52
0.0285	0.0008	0.0255	0.0006	10.46	15.40
0.0049	0.0004	0.0057	0.0003	13.89	18.15
0.0049	0.0009	0.0049	0.0007	0.16	15.80

Chapter 7 Conclusion

In conclusion, from our study we can say that,

- The difference between IITPAVE and ABAQUS values for stress were minimal. The highest contrast in stress was in the base top layer while for strain it was sub-base bottom layer.
- The variation in number of elements doesn't affect the analysis results very much after the optimal mesh.
- As we don't have any practical data, we cannot deduce whether ABAQUS or IIPAVE is more precise for real life behaviour.
- IITPAVE is better for design purposes as it is easier to learn and implement for engineers. On the other hand, ABAQUS provides much more detailed data of various mechanical properties which is more beneficial for research work.

Chapter 8 References

Ahirwar, S. K., & Mandal, J. N. (2017). Finite Element Analysis of Flexible Pavement with Geogrids. *Procedia Engineering*, *189*, 411–416. https://doi.org/10.1016/j.proeng.2017.05.065

Alkaissi, Z. A. (2020). Effect of high temperature and traffic loading on rutting performance of flexible pavement. In *Journal of King Saud University - Engineering Sciences* (Vol. 32, Issue 1, pp. 1–4). King Saud University. https://doi.org/10.1016/j.jksues.2018.04.005

Azimi, M., Mirjavadi, S. S., & Asli, S. A. (2016). Investigation of Mesh Sensitivity Influence to Determine Crack Characteristic by Finite Element Methods. *Journal of Failure Analysis and Prevention*, *16*(3), 506–512. https://doi.org/10.1007/s11668-016-0117-y

Bangladesh - The World Factbook. (n.d.). Retrieved May 17, 2022, from https://www.cia.gov/the-world-factbook/countries/bangladesh/#transportation

Bathe, K.-J. (2016). *Introduction to Finite Element Analysis (FEA) or Finite Element Method (FEM) Finite Element Analysis (FEA) or Finite Element Method (FEM)*. 1065. http://web.mit.edu/kjb/www/Books/FEP_2nd_Edition_4th_Printing.pdf

Gupta, A., & Kumar, A. (2015). Comparative Structural Analysis of Flexible Pavements Using Finite Element Method. *International Journal on Pavement Engineering & Asphalt Technology*, *15*(1), 11–19. https://doi.org/10.2478/ijpeat-2013-0005

Hassan, K., Elghali, L., & Sowerby, C. (2004). DEVELOPMENT OF NEW MATERIALS FOR SECONDARY AND RECYCLED AGGREGATES IN HIGHWAY INFRASTRUCTURE. *Undefined*.

Hou, S., Shi, X., Deng, Y., & Gu, F. (2018). Evaluation of rutting and friction resistance of hot mix asphalt concrete using an innovative vertically loaded wheel tester. *Construction and Building Materials*, *176*, 710–719. https://doi.org/10.1016/j.conbuildmat.2018.05.064

IRC. (2018). *IRC 37:2018_Latest*. *53*(9).

Maqbali, L. Al, & Ragab, O. (2021). A Review of Software in Flexible Pavement Design. *Sustainability in Environment*, *6*(2), p27. https://doi.org/10.22158/se.v6n2p27

Mo, L. T., Huurman, M., Wu, S. P., & Molenaar, A. A. A. (2008). 2D and 3D meso-scale finite element models for ravelling analysis of porous asphalt concrete. *Finite Elements in Analysis and Design*, *44*(4), 186–196. https://doi.org/10.1016/j.finel.2007.11.012

Rahman MT, Mahmud K, & Ahsan S. (2011). Stress-Strain characteristics of flexible pavement using Finite Element Analysis. *INTERNATIONAL JOURNAL OF CIVIL AND STRUCTURAL ENGINEERING*, *2*(1). https://doi.org/10.6088/ijcser.00202010104

Shashikant, T. M., & Bindu, R. S. (2015). Effect of Mesh Size on Finite Element Analysis of Plate Structure. *International Journal of Engineering Science and Innovative Technology (IJESIT)*, *4*(3).

V.Chaudhari, S., & A. Chakrabarti, M. (2012). Modeling of Concrete for Nonlinear Analysis using Finite Element Code ABAQUS. *International Journal of Computer Applications*, *44*(7), 14–18. https://doi.org/10.5120/6274-8437

Yang H. Huang. (2004). *Pavement Analysis and Design* (2nd Edition). Pearsons.