2023-08-01

Rigid Pavement Analysis System (rpas): An Enhanced Graphical User Interface

Samina Samrose
University of Texas at El Paso

RIGID PAVEMENT ANALYSIS SYSTEM (RPAS): AN ENHANCED GRAPHICAL USER INTERFACE

SAMINA SAMROSE

Master's in Civil Engineering

APPROVED:

Cesar Carrasco, Ph.D., Chair

Imad Abdallah, Ph.D.

Alejandra G Castellanos, Ph.D.

Stephen L. Crites, Jr., Ph.D.
Dean of the Graduate School

Dedication

To my family members, who gave me inspiration the whole time to achieve my goals.

To my friends, especially Anannya Doris and Linda Nodjimbadem, for their unconditional love, constant support and heartfelt prayer at every stage in my journey.

RIGID PAVEMENT ANALYSIS SYSTEM (RPAS): AN ENHANCED GRAPHICAL USER INTERFACE

by

SAMINA SAMROSE

THESIS

Presented to the Faculty of the Graduate School of

The University of Texas at El Paso

in Partial Fulfillment

of the Requirements

for the Degree of

MASTER OF ENGINEERING

Department of Civil Engineering

THE UNIVERSITY OF TEXAS AT EL PASO

August 2023

Acknowledgements

First of all, I would like to express my heartfelt gratitude to my advisor, Dr. Cesar Carrasco, for his continuous guidance and support throughout the course of this research. His patience, mentorships and insightful feedback have been invaluable in shaping the outcome of this study. Without his dedicated supervision, this accomplishment would not have been possible.

I am deeply grateful to my thesis committee members, Dr. Imad Abdallah and Dr. Alejandra G Castellanos for generously offering their time to become committee members.

Additionally, I would also like to thank the students and staff at the Center for Transportation Infrastructure System (CTIS) at UTEP. My especial recognition goes to Dr. Cesar Tirado, who assisted me tremendously in this research. His expertise and insightful discussions have significantly contributed to the success of this study.

Abstract

In recent decades there has been a significant interest among researchers in developing pavement analysis tools that will work with high efficiency and provide accurate results within a short timeframe. The University of Texas at El Paso (UTEP) has developed a finite element-based pavement analysis software named "Rigid Pavement Analysis System" (RPAS) capable of analyzing Jointed PCC pavements of any number of PCC and soil layers. The processing core of this software was developed in MATLAB with a pre- and post-processing user interface developed in C++. The MATLAB code is compiled into an executable and together with the C++ interface they are installed by the user as a stand-alone application that runs in Windows. While this combo of tools has worked well, it has limited the implementation of new features because it requires development skills in both MATLAB and C++. For this reason, it was decided that to further improve the usability and development of RPAS a new user interface developed in MATLAB was needed.

The work leading to the development of this new interface is described in this thesis. The new interface was developed to have a similar appearance to the one developed in C++ to ease the transition for users already familiar with the previous interface.

A series of case studies were used to verify the functionality of the interface. There was no need to implement a full verification and validation of the results since that has already been preciously done. The purpose of the case studies was to verify that the user inputs were appropriately passed to the processing core.

This research may have a significant contribution to the field of pavement analysis by providing a new tool that in the future could be accessed through a website without the need for a stand-alone application.

Table of Contents

List of Figures

Chapter 1: Introduction

For many decades engineers and researchers have had a significant interest in pavement analysis as a tool to determine pavement performance, condition, material properties, the lifetime of a pavement and many other characteristics. Recent advances in technology has led to more sophisticated pavement analysis techniques, accurate predictions, and better pavement performance. Because of this increased interest, many researchers have explored the creation of more modern and easier to use analysis systems leading to more accurate pavement performance predictions.

For many decades, Final Element Analysis (FEA) has become one of the most used mathematical models for payment analysis. After an immense amount of research on different software tools, researchers have come to a conclusion that finite element-based software tools provide the most reliable results compared to other software tools (Srikanth M R and Sri Jayachamarajendra College of Engineering, 2015). To determine the crack propagation, stress of the pavement, pavement response and different properties of flexible pavement and rigid pavement, FEA has been commonly used (Hu et al., 2008),(Wu et al., 2014). Such applications show the effectiveness of FEA in providing valuable insights for optimizing pavement design and enhancing performance.

Recently, engineers and researchers have been very interested in using MATLAB for different kinds of computer-based modeling. To predict the performance of the flexible pavement, researchers have used MATLAB with fuzzy logic theory (Olowosulu et al., 2022). Moreover, it is widely used in many other contexts such as the effect of any kind of irregularities present due to dynamic interaction between vehicle and roadway, high load vehicle, heavy traffic and so on are being assessed by MATLAB software.

In the last decade researchers at the University of Texas at El Paso (UTEP) developed "Rigid Pavement Analysis System" (RPAS) capable of analyzing Jointed PCC pavements of any number of PCC and soil layers (Aguirre et al., 2019). RPAS was developed MATLAB with a user interface developed in C++. Over the last couple of years, the core finite element code developed in MATLAB has seen significant improvements. However, the user interface has not been updated to make use of these new capabilities because of the difficulties in working with both MATLAB and C++. For this reason, in this research a new user interface has been developed to have the entire code in MATLAB. With this new user interface it will be easy to modify or implement new features in the interface. Also, for this interface, users will not need to install any software; all that will be needed is MATLAB.

Chapter 2: Literature Review

Rigid pavements are constructed of Portland Composite Cement slabs resting on different foundation sublayers. The sublayers play a vital role in spreading the surface load over a wide area, consisting of three main layers- base, subbase, and subgrade. As rigid pavements have high modulus of elasticity, they deflect extraordinarily little under loading. There are several types of rigid pavements such as Jointed Plain Concrete Pavement (JPCP), Jointed Reinforced Concrete Pavement (JRCP), Continuous Reinforced Pavement (CRCP). JPCP does not have any steel reinforcement, but it may contain dowel and tie bars across the transverse and longitudinal joins, respectively. In JRCP steel reinforcement in addition to dowel and tie bars is used. CRCP is another type of rigid pavement and is only heavily reinforced without any dowel or tie bars.

2.1 ILLI-SLAB

ILLI-SLAB was first used to analyze jointed concrete pavement with Winkler foundation in 1979 (Tabatabaie et al., 1979). The main limitation of this finite element model (FEM) was that the shear deformation could not be calculated which is an essential parameter for designing thick slabs. Also, to analyze the curling behavior of pavement layers, it was considered that two layers could be simplified to an equivalent single-layer system. So, ILLI-SLAB could not perform individual analysis of the two layers.

The primary finite element tool possessed the capability to simulate Winkler foundation. However, over several improvement phases, the Boussinesq elastic solid, the stress dependent resilient subgrade and the Vlasov 2-parameter foundation subgrades idealizations became feasible to analyze as well (Ioannides et al., 1985). Over time, certain limitations were discovered within this existing analysis model. For instance, when the temperature gradient was being considered, the model was limited to analyze only a single layer overlooking the potential effects of the multi-layered structures. After acknowledging the essential limitations, further development was needed of the analysis model. An improvement of ILLI-SLAB had been done which modeled pavement by back calculation to get primary properties of the supporting pavement layers (Foxworthy and Darter, 1989).

To overcome some of the major limitations for enhancing accuracy and applicability, an extension of ILLI-SLAB was created which was ILSL2. ILSL2 was not limited only to Winkler model but could model other foundation models as well. With this tool, two layers were analyzed separately by considering multilayer as a series of springs and plates. Further, an Artificial Neural Network (ANN) model was developed to calculate the airfield pavement stresses and compared the result with ILLI-SLAB stresses result (Ceylan et al., 1998). It was concluded that a similar result could be derived in a fleeting time and did not require complex pavement properties with ANN.

2.2 FEACONS

To analyze existing concrete pavements, FEACONS III was introduced in 1986 as the primary tool. This method presented a significant advancement in the field, as it allowed the researchers to accurately simulate the behavior of JPCP.

In the FEACONS III model, the jointed concrete pavement was modeled as a rectangular plate, with each node possessing three degrees of freedom. This representation facilitated the analysis of important parameters such as deflection and stresses within the pavement structure. To accurately capture the behavior of the stiffness of the dowel bar, it was modeled with respect to slip distance. The stiffness varies linearly with the difference in deflection at the joint when the difference in deflection is less than the slip difference.

The subgrade was modeled as a Winkler foundation. This approach allowed researchers to account for the interaction between the pavement soil and underlying soil. Furthermore, subgrade voids were modeled as initial gaps between the slabs and the Winkler springs at specified nodes. It was concluded that this program could be used to estimate parameters and deflection and stresses within JPCP structures (Tia et al., n.d.). However, despite its efficiency and promising capabilities, no further development was pursued of this tool.

2.3 EVERFE

ILLI-SLAB and other similar software utilize 2D models of the slab and foundation which introduce some limitations in the level of detail and accuracy in the prediction of pavement response. To overcome this problem, a 3D FE model, EverFE1.02, was developed (Davids et al., 1998). This tool analyzed multilayered pavement considering the slab and first layer of foundation as 3-D brick element. Also, this tool can analyze complex geometric models in a brief time. The improved version of EverFE1.02 was developed in 2003 and was called EverFE2.2 (Davids et al., 2003). This version was modeled in C++. With this version, multiple slab layers and dowel and tie bars could be modeled easily. Also, in the earlier version, thermal gradients were not considered but in EverFE2.2.

2.4 JSLAB

J-SLAB was developed in 1986 (Tayabji and Colley, 1986) in FORTRAN to analyze jointed PCC pavements. It had the capability to evaluate the effects of joints with non-uniformly spaced dowels across pavement joints. Although it could model dowel bars, it was only able to analyze a single slab under curling. The primary problem of this tool was it used to take a long time to execute. It could predict thermal stress, but the program ran very inefficiently (Smith et al., 1990).

The upgraded program, J-SLAB92, modeled dowel bar as beam elements which predicted the shear deformation and elastic support. The program analyzed the curling behavior in terms of linear temperature variation in the slab. Earlier versions did not add self-weight to the loading system, which was developed later in JSLAB92. In the curling analysis, a two-step procedure was performed to use self-weight. Along with the stress calculation, this version could locate the location and maximum value of stress (Ozbeki et al., n.d.).

Several developments and modifications were also conducted to improve the analysis process of J-SLAB. J-SLAB2004 could analyze jointed pavements with one or two unbonded or fully bonded layered pavement slabs resting under Winkler, spring, Boussinesq, two parameters, three parameters and ZSS foundations. One of its limitations was it needed more memory and computational power. Although this tool performed better under different, it had some limitations. (Limouee, 2009).

2.5 NYSLAB

With the improved JSLAB tool, which was NYSLAB, a lot of complications on analyzing pavement were solved. NYSLAB was coded in MATLAB. With this MATLAB code, different slab foundations such as Winkler, Vlasov and solid foundation model could be analyzed whereas J-SLAB was unable to analyze Vlasov foundation. It also modeled the contact between different layers more precisely. To predict the edge deflections and stresses, NYSLAB was being used as it was more user friendly. The model extended the foundation beyond the edge and edge deflection and stresses were measured. With the precise stresses and deflection information, the pavement design was easier and accurate. It improved the thermal profile which was considered linear previously. But with the improved scenario, non-linear thermal profile could be detected, and accurate structural characteristics could be detected (Carrasco et al., 2011).

It was concluded that NYSLAB could model with irregular geometry along with no limitations on number of foundation layers. Uniform and non-uniform meshes were performed, and it was concluded that the element aspect ratio did not affect convergence of deflection. For running a mesh analysis, aspect ratio had an impact on stresses. To verify the results, a comparison of JSLAB2004 and ISLAB2000 tools was made. Equivalent results were found without ISLAB2000 results as it did not consider self-weight of pavement. Furthermore, parametric studies were conducted to learn the effect of distinct characteristics of pavement. By extending the foundation layer beyond the edge of the slabs, a better result of deflection and stresses were developed (Limouee, 2009).

In the bending stresses condition, the connection between the slab and the foundation layer is significant. To portrait importance of thermal effect, a 2D plate element was analyzed to predict the frictional contact between unbonded layers. For this, an interface element was used as proposed (Barbero et al., 1995). The interface elements worked great in compression but in tension. Necessary steps were taken to avoid the unnecessary mesh refinement. It was examined that the entire slab area had a frictional traction impact (Zokaei-Ashtiani et al., 2014).

Several consecutive improvements to the NYSLAB software have been made to ameliorate the current one. Now with the improved NYSLAB software, pavement responses due to thermal and traffic loads could also be predicted (Zokaei-Ashtiani et al., 2013). Nonlinear simulation could be done to detect the thermal effect on the pavement. Although the result of temperature change was same for linear and non-linear cases, the nonlinear thermal terms could predict the change scrupulously. The result represented that nonlinear temperature gradient produces remarkable stresses which could cause under design of slabs.

In the course of time, complete thermo-mechanical responses were researched with NYSLAB. For a typical JPCP with different nonlinear temperature gradient, additional stresses were obtained. So, considering temperature gradient as linear term might cause underdesign of pavement. To comprehend the effect of friction, different coefficients of friction were used. Due to frictional resistance, uniform tensile stresses and negative bending moments occurred. Increment in coefficient of friction caused little bit of vertical deflection and reduced moment (Ashtiani, 2014).

2.6 RPAS

Several successive improvements of NYSLAB were performed and from that a new improved analysis tool was established named Rigid Pavement Analysis System (RPAS). This system used a finite element method to calculate arbitrary vehicle loading and temperature profile of pavement without any difficulty.

In order to have more efficient design structure, researchers of the University of Texas at El Paso did a comparative study on the Winkler foundation model and the 3-D foundation model. With a 3-D FE modelling tool in MATLAB, pavements having different geometric configurations, foundation models, temperature profiles and traffic loads could be assessed easily. For getting the precise pavement design, subgrade reaction value, k, was used in the MATLAB tool. The k value represents the support of the layers below PCC slab. A comparison between Winkler model and

3-D FE model was performed (Aguirre et al., 2019). Although similar results were found with both models, Winkler foundation presented more stresses which caused over design of pavements. Moreover, this model could only assess the PCC layer.

Researchers had worked relentlessly to improve the analysis software of concrete pavement. To get the accurate design value, several calibration processes were performed. It was shown that 3D solid foundation performed better than traditional methods (TaghaviGhalesari et al., 2020). With the use of rigid pavement analysis system, more accurate pavement design could be assessed. In this method, an adjusting factor was used which helped to check the precision of the model. This method was considered convenient because it made an average of multiple similar data. So, the overall processing time was reduced with a noticeable amount of time.

Earlier, mechanistic-empirical design method was used to find the design parameters of pavement layers. This process made a simplification of different layers into a single bed of spring element (*Mechanistic-Empirical Pavement Design Guide*, 2020). By the simplification of multiple layers, this process could not provide accurate pavement responses. Thereby, a development in Rigid Pavement Analysis System (RPAS) was done by establishing a 3-D finite element tool, which was an improvement of NYSLAB. This method was developed to remove the complexity in performing numerical analysis for different continuous reinforced pavement sections which was very time consuming. A study was performed to compare performance of 3-D solid foundation model and ABAQUS by capturing responses of concrete pavements under single tire load. Also, to measure the accurateness of the model, a comparison with RPAS results with National Airport Pavement Test Facilities data and field-testing data from Minnesota Road Research Project (MnROAD) had been carried out (Taghavi Ghalesari et al., 2020). The result showed that the simulation was sufficient for 3-D solid foundation model in comparison with ABAQUS.

A complete study on the necessity of calibration of software and calibration of RPAS was delineated by one of the researchers at the University of Texas at El Paso. After reviewing the capabilities and pros and cons of RPAS software, a calibration framework was established and compared with field data to assess the result. A discrepancy in result was found between the uncalibrated RPAS results and field results. So, to achieve a better pavement design, it was necessary to use calibrated tools. For this, a vast range of calibration factors ranging from 0.75 to 1.6 were used to find the optimum factor to get accurate results. Results showed that reliability of three-layer pavement increased from 50% to 96% for a 15% accuracy requirement (Taghavighalesari, 2020).

Recently, research on the reaction of subgrade layers, which is subgrade reaction value or *k* value, was conducted to improve RPAS performance. The higher the *k* value is, stresses on pavement due to wheel load is lower. It was a 2% variance (Aguirre, 2020).

Throughout a prolonged period, researchers have worked relentlessly to develop and improve software to analyze rigid pavement appropriately. With persistent efforts, new software has been developed gradually. Although several enhancements have been confirmed since 1979, researchers are still working to improve the existing software. The recent software which was developed in 2020 takes less memory and space in the computer and performs better than other ones. Researchers are trying to introduce innovative technologies in this field to make pavement analysis easier and more reachable.

It was assumed that the *k* value did not have any effect on the thickness of concrete layer. But with the increment in thickness of concrete layer, the *k* value decreased. From this, it was supposed that thickness of different layers had an impact. To assess this, a cement treated base and subgrade layer was modeled using both 3-D solid foundation model and Winkler model (Aguirre et al., n.d.). It was found that the Winkler model could not predict the supporting layer responses, but 3-D solid foundation model could. From RPAS analysis, it was concluded that the thickness of pavement layers had sufficient effect. With adequate thickness and stiffness of base layer, optimum pavement design could be achieved although having no change on roller compacted concrete layer.

RPAS was used to assess pavement health by determining new crack formation. With the maximum stresses within a specific section, crack formation probability was determined. Previous FE tools could not analyze the contact problem of different slab layers. But RPAS could examine this with a simple function in MATLAB. However, the performance of the new tool was impressive, it was necessary to verify the result as well. For this reason, the results were compared with ABAQUS model. And it was found that the results obtained from RPAS never exceeded.

Chapter 3: Graphical User Interface (GUI)

This chapter describes in detail the new Rigid Pavement Analysis System (RPAS) graphical user interface.

3.1 PROGRAMMING TOOL

Because the main processing core for RPAS was developed in MATLAB, it was decided that to make not only a user-friendly but also an easy to update pavement rigid pavement modeling tool the new GUI had to also be developed in MATLAB. MATLAB is a versatile programming tool used in many science and engineering fields. It was because of these capabilities that the Finite Element processing core was developed in MATLAB. However, at the time the initial development started, MATLAB did not have a robust set of tools to develop a complex GUI. Over time, and because of the large demand for MATLAB, different types of toolboxes have been added to make it easier to develop feature-rich GUI's.

3.1.1 Graphical User Interface

A graphical user interface (GUI) is a software-based system designed to create a visual interactive tool for users to enter the necessary information for the underlying tool to execute and display the expected results. As previously mentioned, RPAS was originally developed with a GUI developed in C++ but in the research discussed here a new user interface developed in MATLAB was presented. MATLAB has many built-in tools that help to code and develop a graphical user interface without having any extensive programming knowledge.

3.1.1.1 Components, Callback Functions and User Functions

To develop the RPAS user interface several components of the default MATLAB GUI development app have been used. Common features like numeric edit field, text edit filed, button, check box, dropdown list box, axes, radio button group, slider etc. have been used extensively. The user can interact with each numeric text filed, except for fixed values, making them editable.

In scenarios where there are multiple choices available, a drop-down option with callback function is used mostly. This user-friendly approach enables users to select their desired option from a list, and upon selection, the system responds accordingly.

Radio buttons and check boxes have been used throughout the interface to facilitate the user selections in a controlled and straightforward way. For radio buttons, users can select only one option at a time. This helps the users to limit their choice of options. On the other hand, the checkbox allows users to select multiple options simultaneously, where multiple choices are applicable. Users can tick or untick checkboxes to record their responses for the corresponding options.

Slider is another essential component integrated into the GUI. This component enables the user to select a value from a continuous wide range of values. This slider has been used primarily to select specific sections of the roadway for result plotting. A callback function has been implemented to respond to the changes in the slider value. As the slider value changes, the result plot dynamically updates to display corresponding results.

One of the extensive components of the interface to be used here is the axes object to display 2D or 3D plots of the results. First, the axes component is used to visually present the arrangement and geometry of the slabs in the longitudinal and transverse directions. Furthermore, these are used to display the truck load picture, it helps user to comprehend load's position. However, the most significant application of this component is plotting the analysis results such as deflection, stress and strain.

Callback functions have been used extensively in this user interface to make it user friendly and interactive. When the user selects a particular option or performs an action, the callback function is triggered allowing the application to respond appropriately. This mechanism ensures that the tools behavior is responsive to the user's input.

User defined functions help to perform a specific task or set of tasks. These are functions that perform specific computations, but they are not directly associated with handling user interaction or events like callback function. These functions can handle complex computations, data manipulations, or other tasks that may be used repeatedly. It helps with code reusability. Also, using functions makes the code more organized and reduces redundancy and makes the code more efficient. It enhances the overall structure of the code, making it easier to read, update, and manage.

In the MATLAB GUI, two types of properties can be utilized: private properties and public properties. Private properties are accessible and relevant only within the specific GUI where they are defined. These properties are limited to the GUI in which they are declared, making them suitable for storing and managing data specific to that particular GUI. On the other hand, public properties are accessible within different parts of the program, including multiple GUIs. Public properties have a broader scope and can be shared and accessed by different components of the MATLAB application. However, in the current implementation, private properties have been chosen to restrict the use and scope of variables, allowing for better organization and encapsulation.

3.2 RPAS GRAPHICAL USER INTERFACE

3.2.1 Material properties Tab

Pavement material properties are captured through the "Materials" tab in the user interface (See Figure 3-**1**). At most 7 pavement layers can be entered and analyzed with different types of slab and soil layers. Of these up to 3 can be PCC layers. This tab consists mostly of dropdown menus, text edit box, numeric text field, checkbox and radio buttons. With the dropdown option, the user can select the number of layers and the response will be generated accordingly. After selecting a particular material type for a layer, automatically a material type (i.e. Winkler, Vlasov and 3-D foundations) and corresponding properties edit boxes will pop up. These are editable boxes and users can modify the values as needed. Along with the change in the material properties, users can select the slab bonding type. They can choose to make a bonded slab or non-bonded slab. The overall view for the material properties tab is shown in Error! Reference source not found..

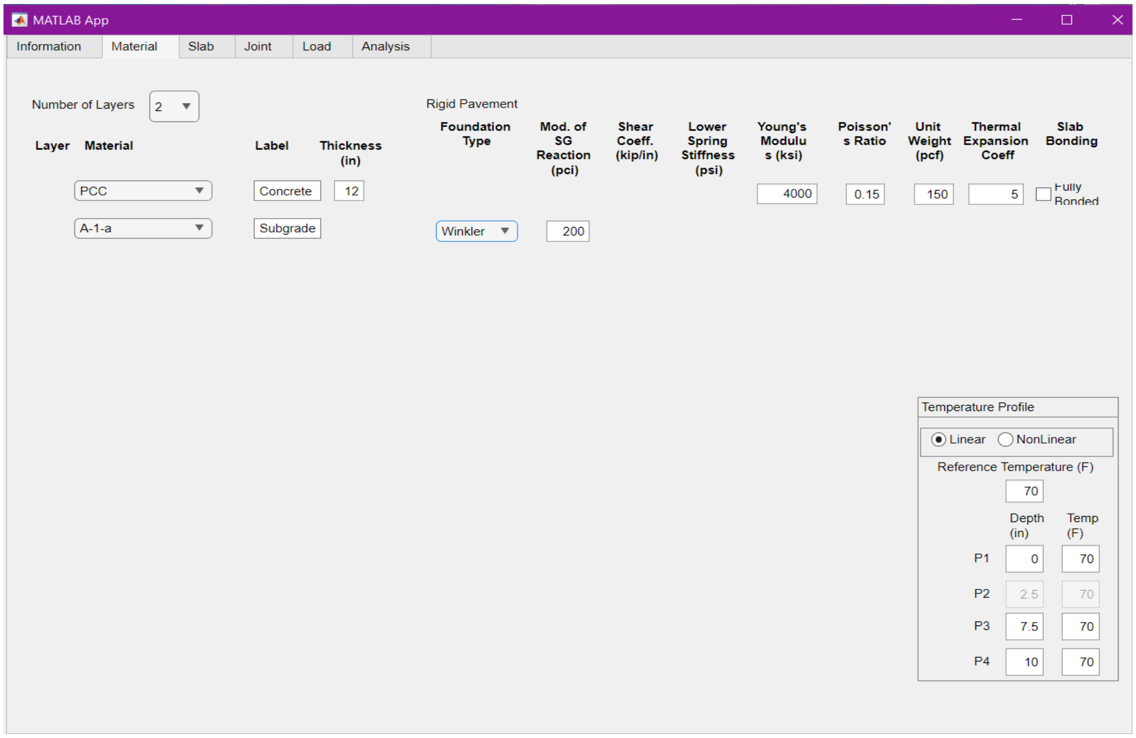


Figure 3-1: Material properties in user interface

In addition to material properties, users can also enter a linear or non-linear temperature profile through the slab thickness. For the linear profile, users must choose only the top and bottom layer temperature and the rest will be calculated linearly. On the other hand, users must select all the layer temperatures to create a non-linear temperature profile.

3.2.2 Slab Tab

In this tab the user can enter the number of slabs in the longitudinal (x) and lateral (y) directions. To make it more user-friendly, users are provided with a drop-down menu. Through this menu, users can choose the desired number of slabs in both directions. Furthermore, users have the flexibility to specify the length and width of the individual slabs based on the analysis requirements.

Once the user selects the desired number and the dimensions of slabs in each direction, the GUI promptly generates corresponding responses with a callback function. The callback function is programmed to generate a graphical plot according to the chosen slab configuration. This plotting callback function is activated for both number of slab selection and the dimension changes. This plot serves the purpose of enabling users to have an idea about the anticipated appearance of the slabs. **Figure 3-2** shows the final view after plotting the slabs based on the user provided information.

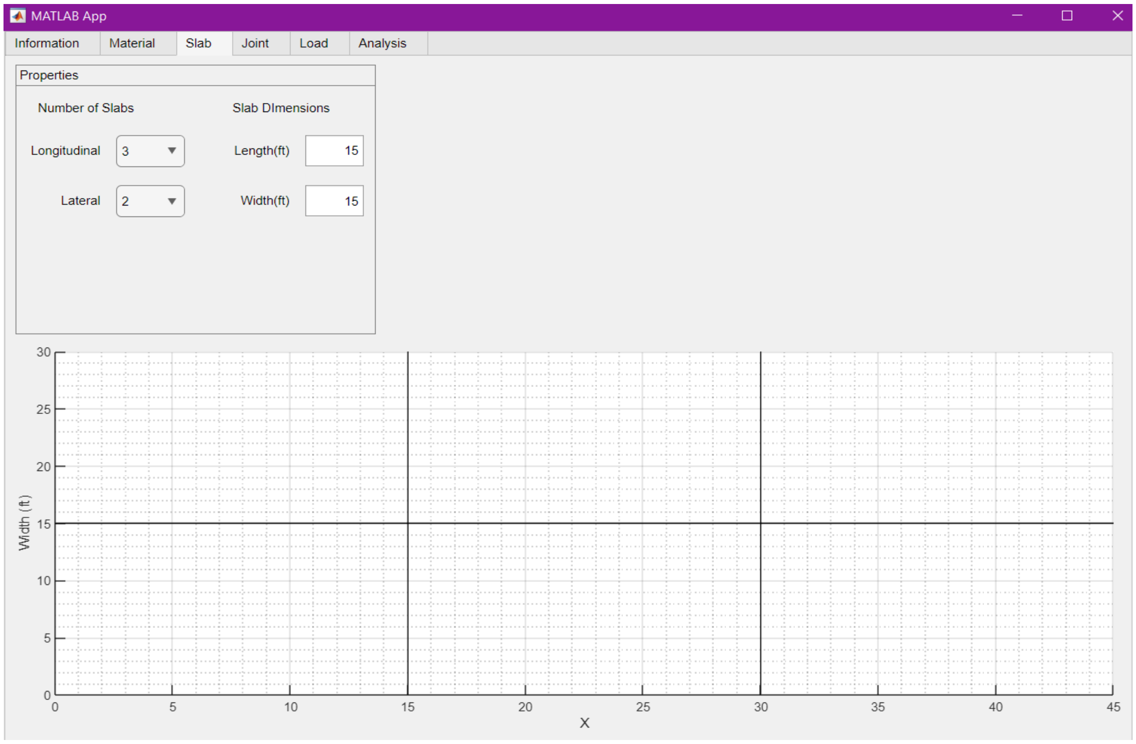


Figure 3-2: Slab properties in MATLAB GUI

3.2.3 Joint Tab

In this tab the type of PCC slab joints and their properties are defined. The joint configurations consist of various options such as interlock, dowel and dowel and interlock for transverse joints and for the longitudinal joints there will be only interlock and tie bar. These joint options are defined using radio-buttons.

Similar to the slab dimensions in the previous tab the slab joints are plotted based on the properties defined by the user. As the user enters or modifies the type of joint a callback function that updates the plot is executed. It is important to note that the dowel bar properties are automatically selected according to the dowel bar propertied from TxDOT specifications. This ensures the properties comply with the established standards. **Figure 3-3** shows a view of the joint tab.

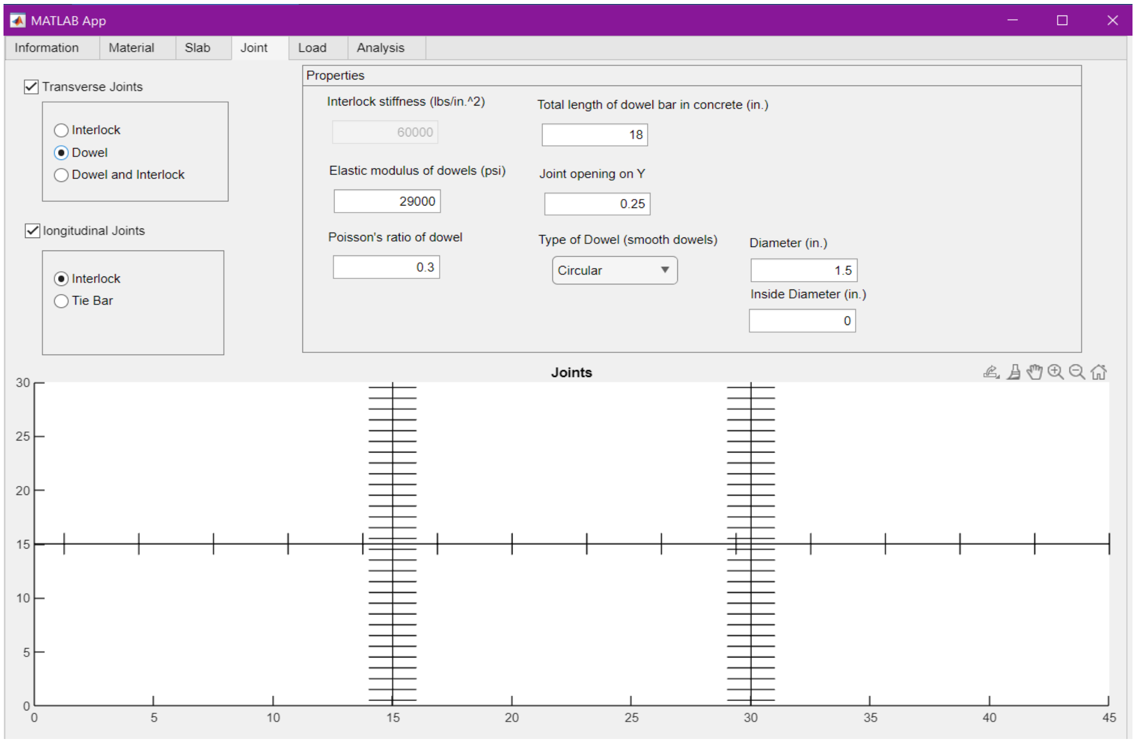


Figure 3-3: Joint tab properties and response

3.2.4 Loading Tab

In this tab users can define the truck configuration to be used in the analysis of the pavement. Users can select an "existing configuration" for trucks already saved or "create configuration" to define a new truck.

The existing configuration serves as the default option and offers a selection of common truck loads. Several common standard truck loads have been added to the interface. Among these loads, the most common is tridem truck, which has three axel groups. An automated selection of this common type of truck has been implemented whenever the user selects this type of truck.

There are several ". trk" files stored in the truck folder. Whenever user selects the load list menu, a callback function reads all the names of the truck load files located in the truck folder and displays the list in the name of vehicle list. By selecting a vehicle type from the vehicle library, the corresponding truck load information is extracted from the associated text file and used to fill all the input components in this tab. This tab has a subtab group which will appear or disappear according to the associated axle group number. And if there is any image assigned with a similar name provided in the document, it will appear on the right side of the tab. This will help users to visualize how the truck is going to look. **Figure 3-4** shows the typical standard truck type and a sample of how the load tab will appear.

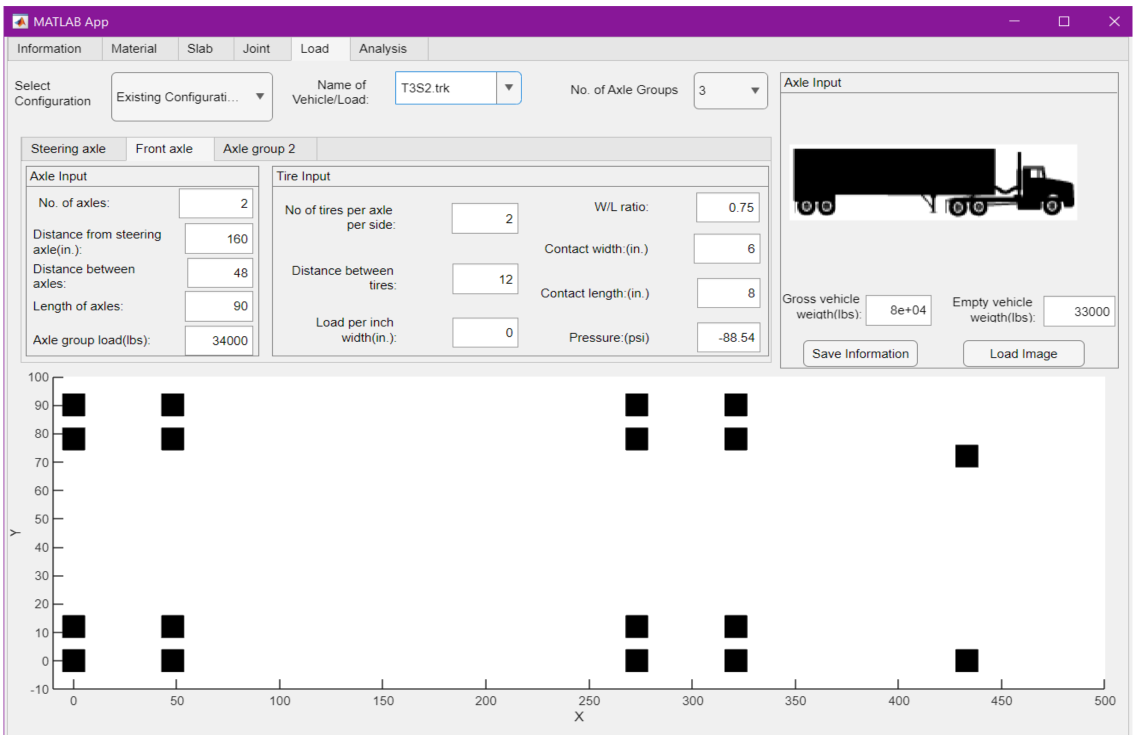


Figure 3-4: Sample of standard truck loading condition

For the 'create configuration' option, users can define their own truck load configurations. This process starts with the user providing a name for the vehicle. Following that, the user selects the number of axle groups from the drop-down menu. With the help of the callback function, whenever the user selects the number of axle groups, the corresponding number of axle sub-groups will appear in the axle group tab. User can have at most nine axel groups for truck creation. Within each axle group, the user can specify the number of trucks axles, axle length, distance between axles, tire pressure, the tire contact patch width and length. These are the main properties of creating a customized vehicular load. It is important to note that the distances are being measured from the consecutive axles and not from the centroid of axle groups. It is optional for the user to include a picture/image of the truck. If the user possesses a relevant picture, it can be associated with the configuration. However, if there is no image available, the system allows the user to proceed without one. The information provided will be saved in the truck folder if the user prefers and a new text file will be created accordingly. After that, anytime user wishes to pick this truck properties, it can be selected from the truck load list.

To visualize how the track is going to look like, a callback function has been set up and being triggered whenever the user modifies any properties of the track such as number of axels within this within excel groups or any other relevant parameter. There is a graph in the load tab that first works within the response of the number of slabs in the x and y directions from the slab tab property. If the user changes the slab type property of the slab type, this part will be changed accordingly. This graph depicts the number of axle groups and the number of axles and tires in each group, serving as a schematic picture of the truck load. By incorporating these features, the system allows users to create and visualize their own tuck loads.

3.2.5 Analysis Tab

In the analysis tab the users can position the truck on the slabs using the "Position" tab. This plays a vital role in the analysis as different positions produce different results. This position can be set through two available options in the position sub tab group. The first option is manually entering the x and y coordinates, enabling precise placement of the truck load and ensuring that the analysis results are obtained at the desired position. The second option allows to select the position of the tire on the slab using the mouse cursor. However, it is crucial to note that regardless of the chosen approach, whether using the coordinate system or mouse click, the specified position always refers to the position of the front right tire. There can be cases where all the tires cannot be fitted on the whole pavement. In these cases, the tires that fall outside of the slabs will not contribute to the analysis.

This tab has several additional options available for analysis, including two different methods: static and influence path. For the sake of simplicity, only the static solution has been implemented in the current system. Furthermore, in the method section, the number of iterations and the spacing between the iterations can be changed too.

Once all the necessary information has been provided, the user can initiate the analysis by clicking the "Analyze" button. This action triggers a button pushed callback function which calls the RPAS finite element analysis core function. The RPAS code executes by taking the stored values from the graphical user interface (GUI), enabling the analysis process to begin. **Figure 3-5** shows a view of this tab.

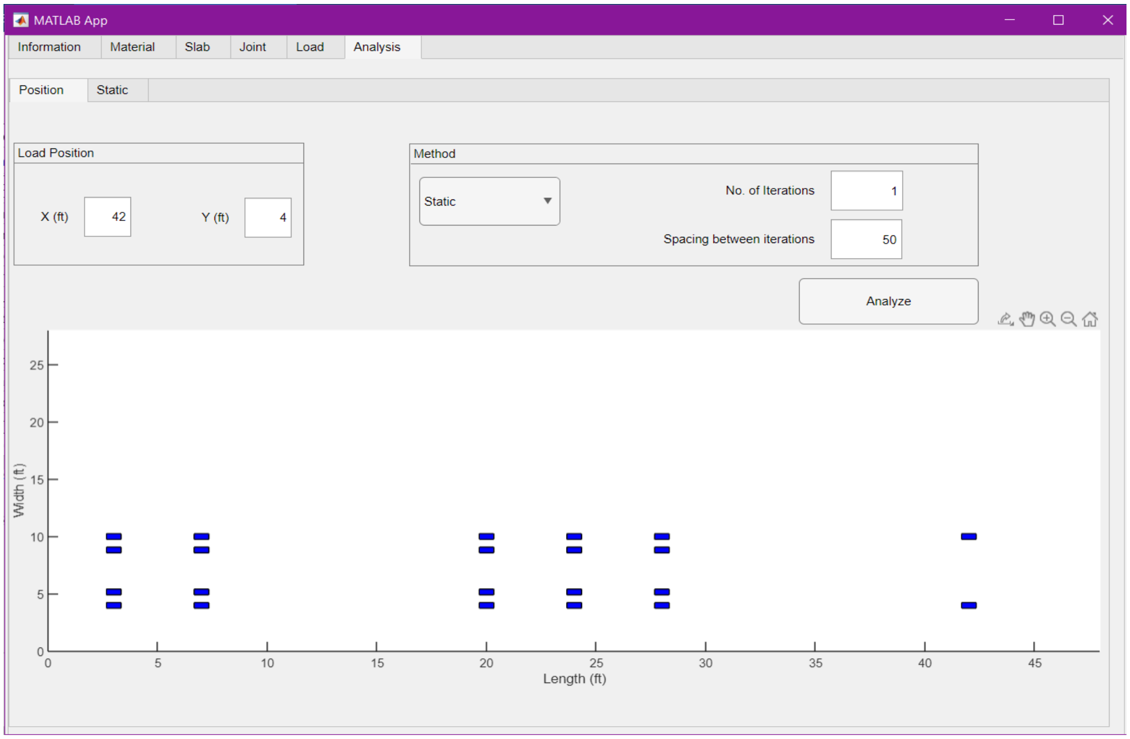


Figure 3-5: Truck load positioning

When the user selects the "Static" subtab the user will be able to plot the results of the analysis. After the completion of analysis, the user may select the desired response from a wide range of options available in the user interface. Various responses including vertical deflection, top longitudinal bending stress, top shear stress, top principal stress, bottom longitudinal bending stress, bottom shear stress, longitudinal rotation, lateral rotation, and more can be derived. A view of this tab is shown in **Figure 3-6**.

There are some important additional features in this tab. A slider, associated with a value changed callback function is added to focus on specific sections of the plotted data. This slider has a range of data options from where users can select a specific section of the pavement. The range of the slider is limited in the maximum and minimum values of the calculated length and width of the slab system. An interpolation function is applied to the derived values within the specified slider data range to get the interpolated result of the specified section. For the interpolation, MATLAB's built in function named 'griddata' has been used. After that, a 2-D plot has been generated using the interpolated data.

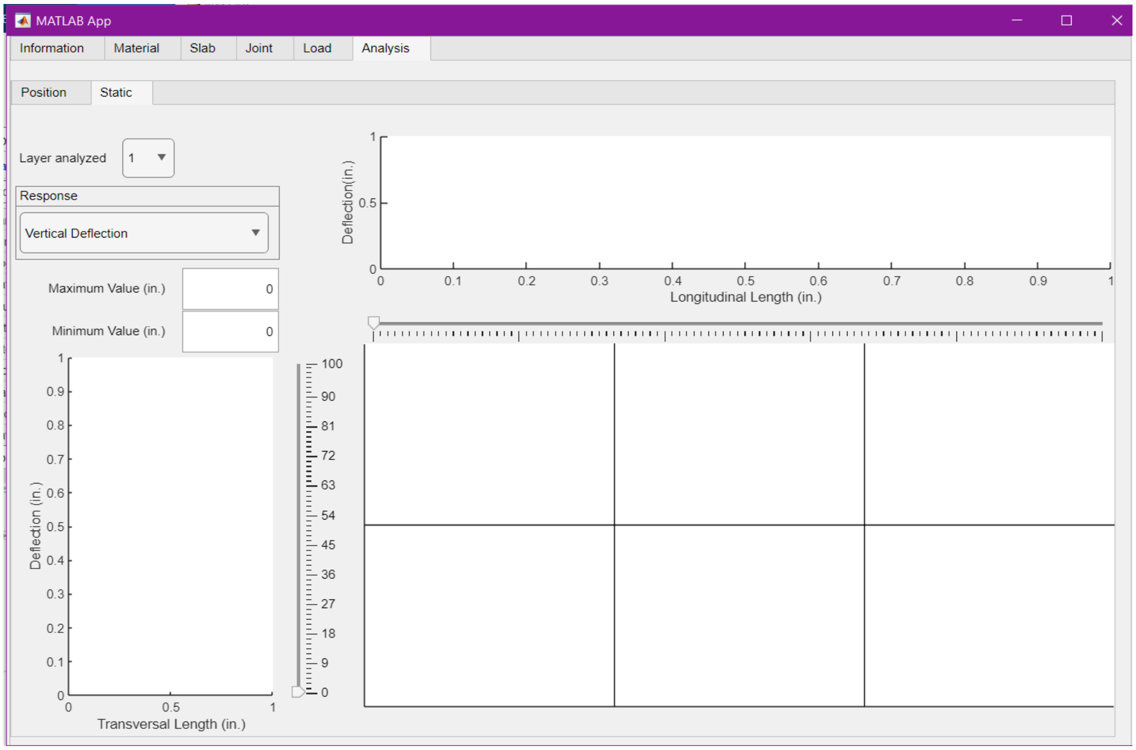


Figure 3-6: Static tab for result visualization

Chapter 4: Evaluation of GUI Performance

To evaluate the functionality of the Graphical User Interface (GUI), a comparison with previously obtained results from a case study was conducted. The goal of this comparison was to assess if the data the user enters in the GUI is properly transferred to the finite element core for analysis.

4.1 DESCRIPTION OF CASE STUDY

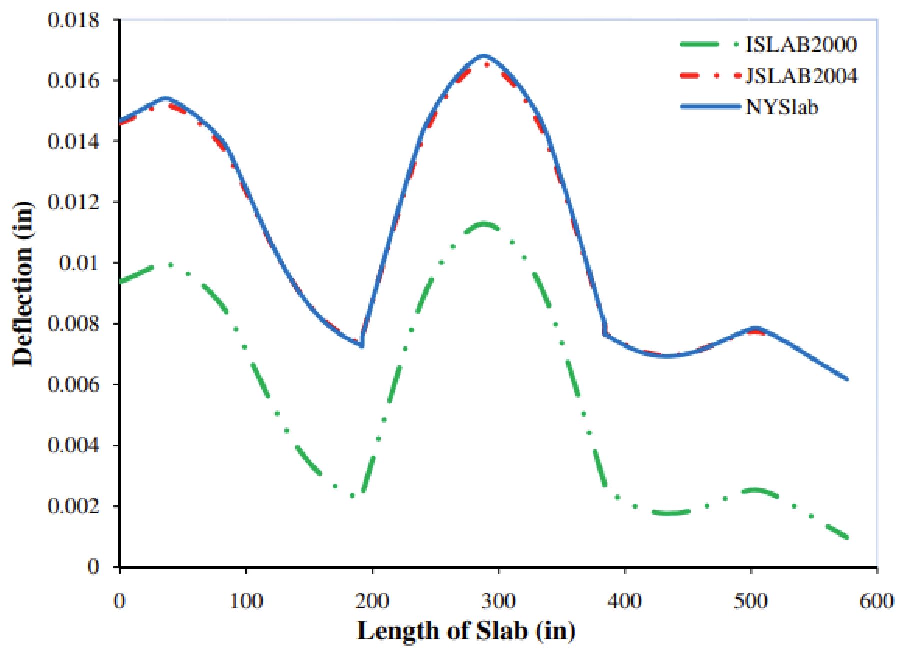
For this case study, a pavement comprising of two by three slabs with one PCC layer and an underlying soil layer has been chosen. This is the same case used by (Limouee, n.d.). The dimensions of the slabs were 16ft by 14ft with a thickness of 12inch. The material properties of the slab are 4000ksi modulus of elasticity, 0.15 Poisson's ratio and 150pcf unit weight. For this case, no irregularities or any temperature profile were added to analyze a simple example. For the joint properties, dowel bars were used for the transverse joints and tie bars were used for longitudinal joints. The transverse joints also had a key-joint with stiffness of 60,000 psi/in. It is important to mention that the dowels are uniformly distributed in all cases. For the foundation a Winkler foundation was selected with a modulus of subgrade reaction of 200 pci.

A truck with three axle groups has been chosen as the traffic load. Each axle group, other than steering axle, has three axles with two tires per side. As previously mentioned, the distances are being measured from the axel to axel. The positioning of the truck is done with the front axle placed at the center of the middle slab. However, due to the length of the three slabs, the last axle was automatically excluded from the analysis. For the tire pressure, a uniform downward value of 100psi is applied to all the tires. To keep the contact area constant, a tire contact area is defined with a width of 8in and a length 6in, resulting in an aspect ratio of 0.75 for each tire. With this kind of configuration, real-world loading conditions are being portrayed as closely as possible.

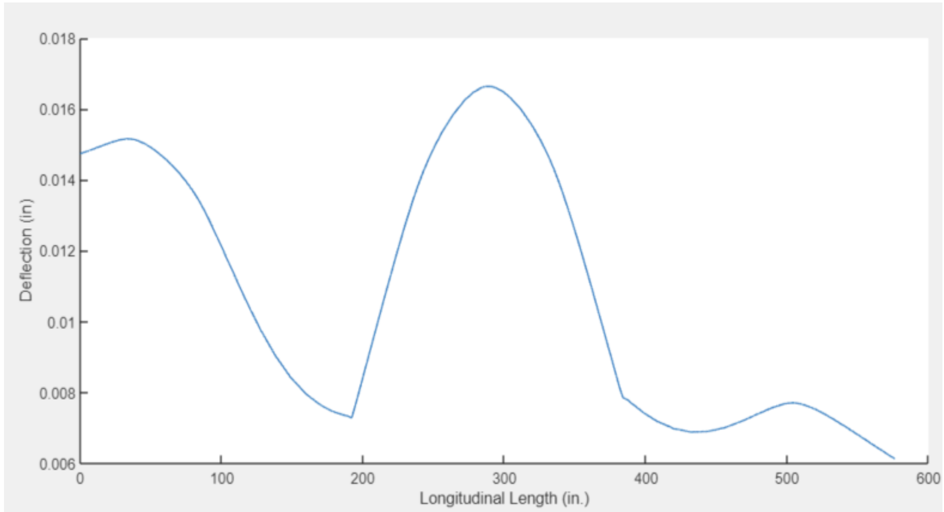
This loading condition with the specified tire dimensions ratio is shown in **Figure 3-5.**

4.2 COMPARISON OF RESULTS

The pavement and loading conditions described in the previous section were entered in the GUI and the analysis conducted produced the following results. **Figure 4-1** (a) shows the slab's deflection profile under the outside tires produced by ISLAB2000, JSLAB2004 and NYSLAB and (b) shows the deflection produced by the new MATLAB user interface. It can be seen that the deflection results match the previous ones; specially the ones for NYSLAB as expected.



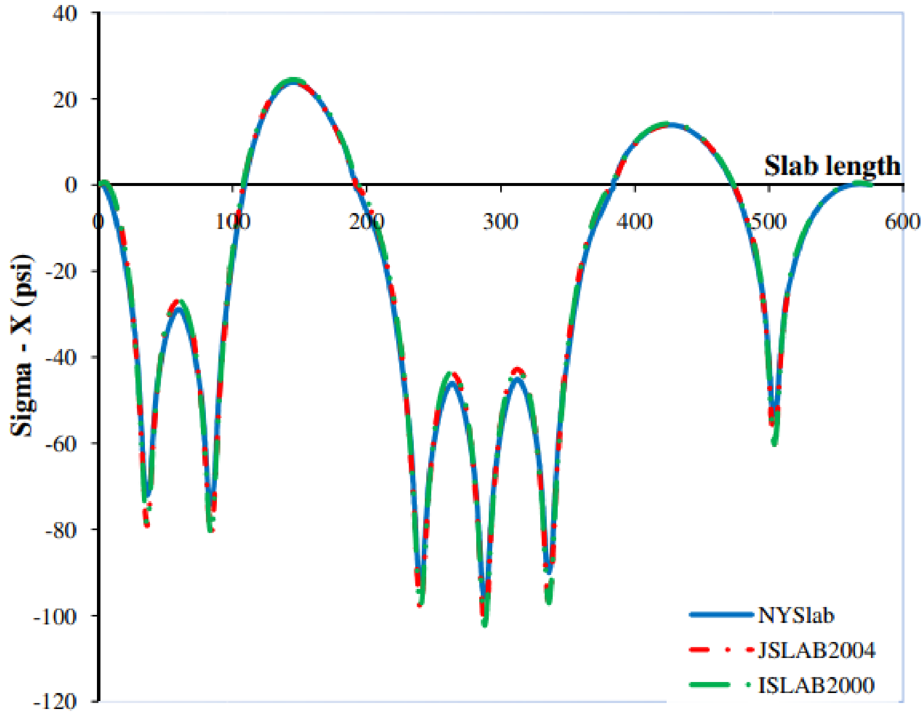
(a)



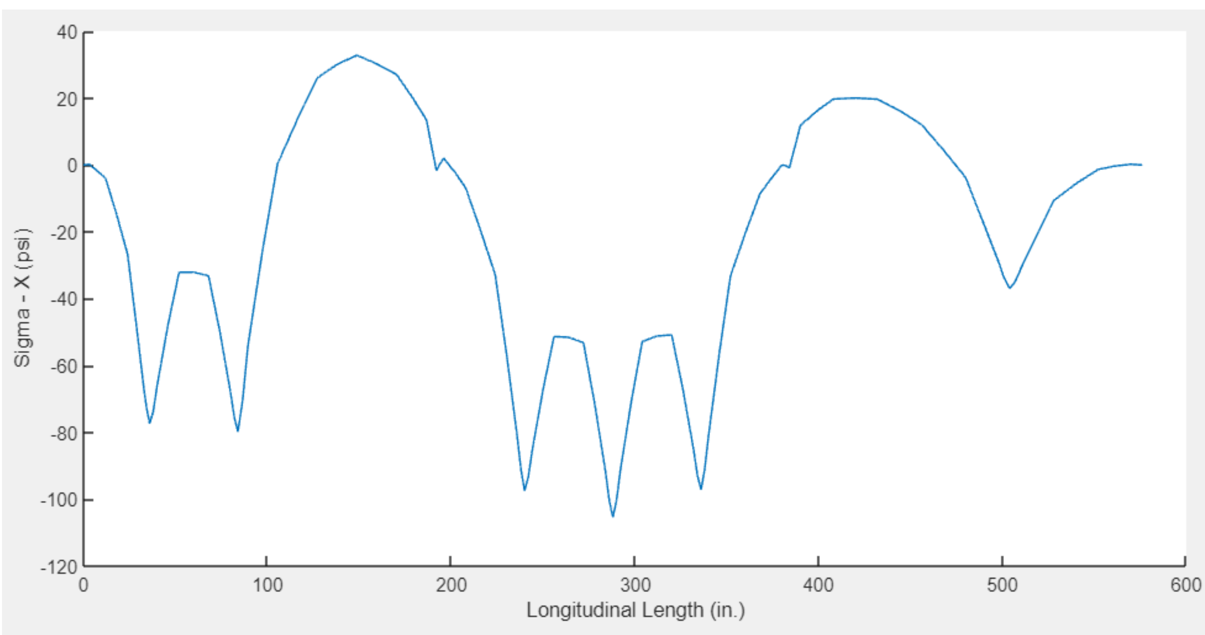
(b)

Figure 4-1: Comparison of slab deflections (a) results from ISLAB2000, JSLAB 2004 and NYSLAB (b) result from new MATLAB User interface

The stress results on the slabs obtained from the new user interface and previously used software provides similar results too. **Figure 4-2** shows the plots of the normal stress in the X direction (longitudinal) produced by a) ISLAB2000, JSLAB2004 and NYSLAB and b) the new user interface. It can be seen that they provide the same pattern, but the results of the new user interface are not as smooth. This is because they were obtained using linear interpolation. Also, we can see that the maximum values and the minimum values are the same in all cases. Error! Reference source not found. shows the surface plot of the normal stresses in this same X direction.



(a)



(b)

Figure 4-2: Top normal stress in the X direction (a) result from previous studies (b) result from new user interface

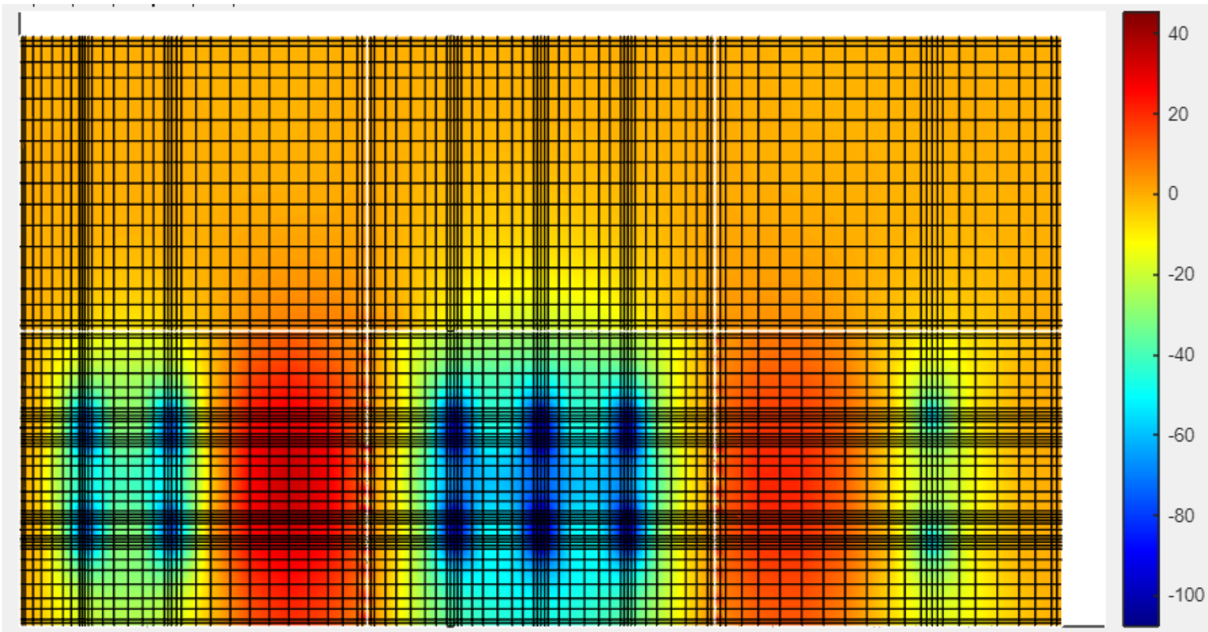
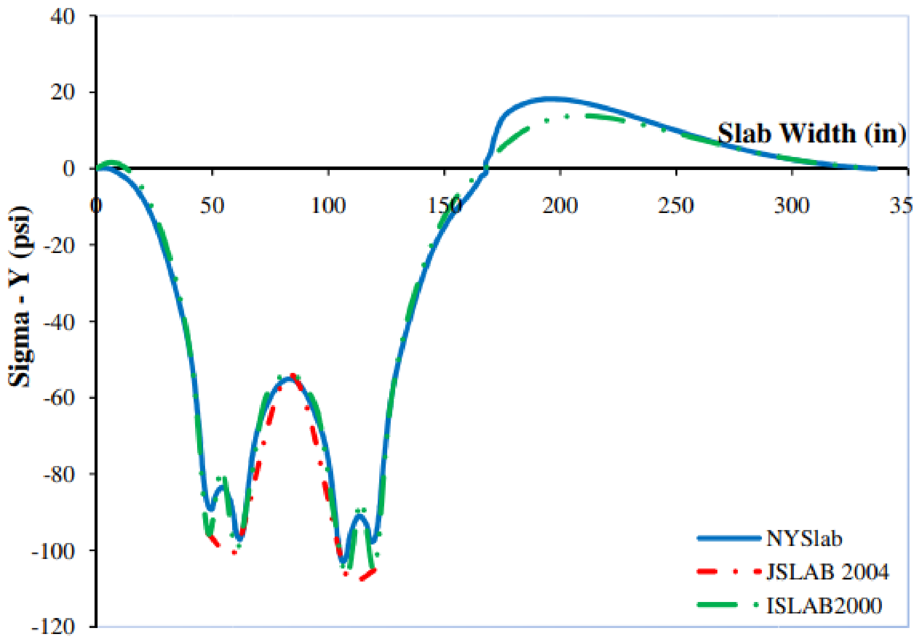
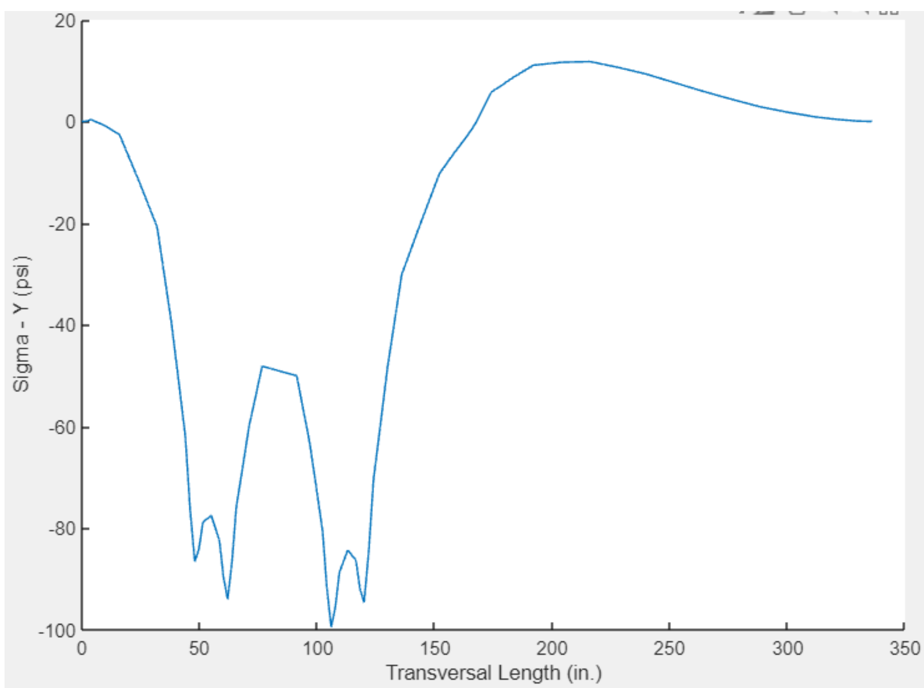


Figure 4-3: Surface plot of top normal stress in the X direction on the PCC slab

The result of normal stresses in the Y direction (transverse) are shown in Error! Reference source not found.. These plots are under the second axle of the second axle group where the maximum stresses occur. It is evident that the stress distribution in the Y direction exhibits similar characteristics to those observed in the X direction. There is the same discrepancy with the previous studies that the plot smoothness does not match (because of the linear interpolation).



(a)



(b)

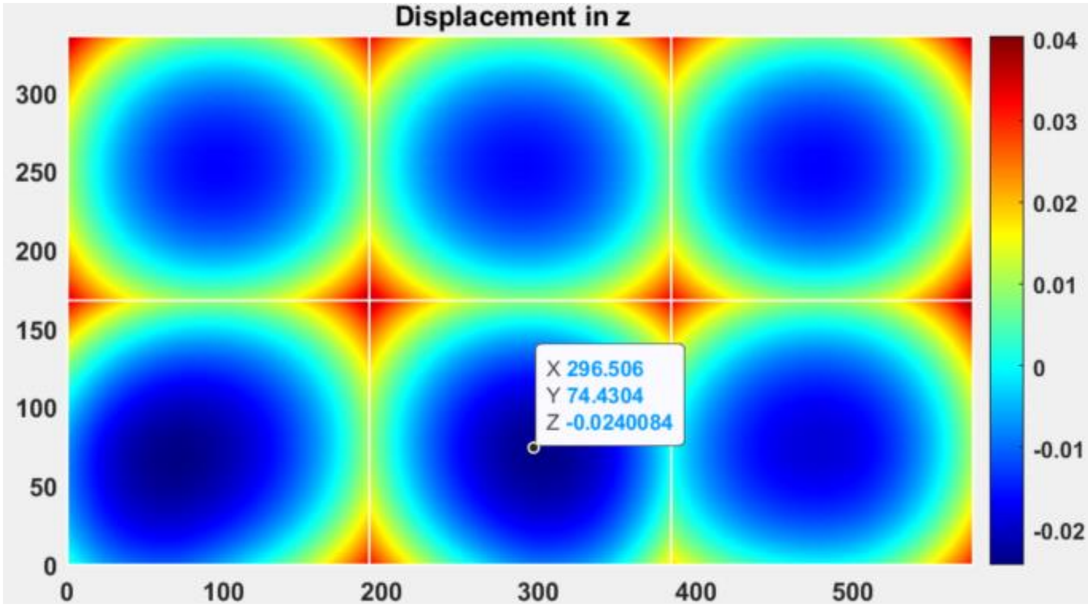
Figure 4-4: Normal stresses in the Y direction (a) results from previous studies, (b) result from new user interface

Chapter 5: Thermal Load Study

In the previous study, one of the simplest foundation models, the Winkler foundation, was used. The present study utilizes the Vlasov foundation model along with the consideration of a temperature profile. The goal of this study is to also verify that the user interface properly transfers the data entered by the user to the core finite element code.

For this study a PCC slab layer and one subgrade layer having 200pci modulus of subgrade reaction and 3 kip/in shear coefficient are used. The temperature profile is considered as non-linear, with the temperature values of 70F, 70F, 80F, 85F in depth of 0in, 0.25in, 7.5in, 10in respectively with 70F as the reference temperature.

A truck with three axle group and tire contact dimensions of 8in length and 6 in width and 100psi. All other parameters are the same as the previous study including the location of the truck on the 3 by 2 slab system. **Figure 5-1**a shows the deflection plots for the case study when the analysis is run directly through the MATLAB finite element code without the use of the user interface while **Figure 5-1** b shows the plots created through the new user interface. From these plots it can be seen that the deflections are the same in both cases indicating that the thermal profile and the soil material properties are properly transferred from the user interface to the finite element code.



(a)

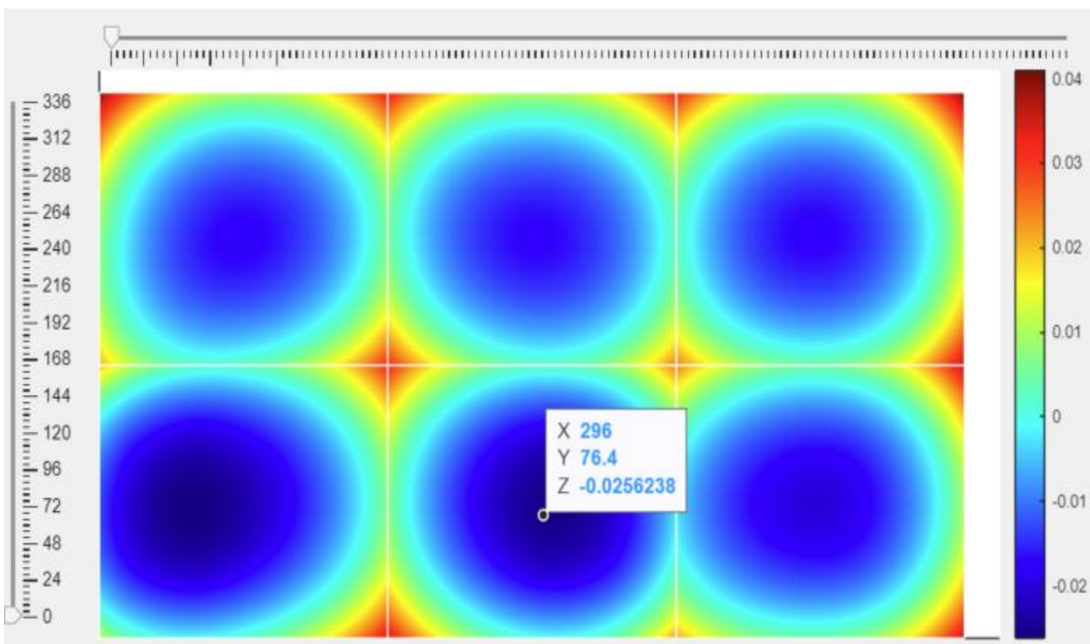


Figure 5-1: Deflection surface plot (a) with RPAS code, (b) with user interface

Chapter 6: Summary and Conclusion

This chapter provides the summary and the conclusion of this research as well as further recommendations.

6.1 SUMMARY

Having the capacity to accurately analyze pavement structures has become a great concern for departments of transportation and consulting firms. For this reason, researchers have made significant efforts to develop improved tools for the analysis of pavements. Over the last decade UTEP developed RPAS as a reliable analysis tool in MATLAB with a user interface developed in C++. The work documented in this thesis has led to the development of a new user interface in MATLAB that will make it easier to implement new capabilities and continue to be user friendly. After the interface was implemented a series of case studies were conducted to verify its operation and the results documented here.

With this new interface users can easily analyze jointed concrete pavement producing results accurately and in a more efficient way than with the previous interface. This tool will help users visualize the stress, strain and deflection of the pavement structure all within MATLAB.

6.2 CONCLUSION AND RECOMMENDATIONS

The results of the case studies conducted as part of this work demonstrate that the new user interface is reliable at providing accurate pavement analysis result. With comparison studies it has been shown that this interface functions effectively under various conditions.

It is recommended that the presentation of the results be done using higher order interpolation instead of the linear interpolation currently implemented. In addition, further development of the analysis tool to be able to be accessed through a cloud-based interface will significantly improve access to many users. Accomplishing this task will remove the requirements for users to install MATLAB and the necessary tool-boxes in their own computer to run the tool. On the cloud-based interface users can be given access credentials and be able to run the tool using the resources in the host server.

References

Aguirre, N., 2020. RPAS: An Enhanced Finite Element Code For The Mechanistic Analysis Of Rigid Pavements 187.

Aguirre, N., TaghaviGhalesari, A., Carrasco, C., 2019. A Comparison of Concrete Pavement Responses Using Finite Element Method with Foundation Springs and 3-D Solid Elements, in: Airfield and Highway Pavements 2019. Presented at the International Airfield and Highway Pavements Conference 2019, American Society of Civil Engineers, Chicago, Illinois, pp. 81–90. https://doi.org/10.1061/9780784482452.009

Aguirre, N., Taghavighalesari, A., Rogers, R., Carrasco, C., Nazarian, S., n.d. Influence of Foundation Layer Properties in a Roller- Compacted Concrete Pavement System Subjected to Heavy Vehicle Loads 13.

Ashtiani, M.A.Z., 2014. Enhanced Finite Element Modeling Of The Thermo-Mechanical Responses Of Jointed PCC Pavements Under Environmental And Traffic Loads 216.

Barbero, E.J., Luciano, R., Sacco, E., 1995. Three-dimensional plate and contact/friction elements for laminated composite joints. Computers & Structures 54, 689–703. https://doi.org/10.1016/0045-7949(94)00355-7

Carrasco, C., Limouee, M., Tirado, C., Nazarian, S., Bendaña, J., 2011. Development of NYSLAB: Improved Analysis Tool for Jointed Pavement. Transportation Research Record: Journal of the Transportation Research Board 2227, 107–115. https://doi.org/10.3141/2227-12

Ceylan, H., Tutumluer, E., Barenberg, E.J., 1998. ARTIFICIAL NEURAL NETWORKS AS DESIGN TOOLS IN CONCRETE AIRFIELD PAVEMENT DESIGN. Presented at the Airport Facilities: Innovations for the Next Century. Proceedings of the 25th International Air Transportation Conference.American Society of Civil Engineers.

Davids, W.G., Turkiyyah, G.M., Mahoney, J.P., 1998. EverFE: Rigid Pavement Three-Dimensional Finite Element Analysis Tool. Transportation Research Record 1629, 41–49. https://doi.org/10.3141/1629-06

Davids, W.G., Wang, Z., Turkiyyah, G., Mahoney, J.P., Bush, D., 2003. Three-Dimensional Finite Element Analysis of Jointed Plain Concrete Pavement with EverFE2.2. Transportation Research Record 1853, 92–99. https://doi.org/10.3141/1853-11

Foxworthy, P.T., Darter, M.I., 1989. ILLI-SLAB and FWD Deflection Basins for Characterization of Rigid Pavements. Nondestructive Testing of Pavements and Backcalculation of Moduli. https://doi.org/10.1520/STP19818S

Ioannides, A.M., Thompson, M.R., Barenberg, E.J., 1985. FINITE ELEMENT ANALYSIS OF SLABS-ON-GRADE USING A VARIETY OF SUPPORT MODELS. Presented at the Third International Conference on Concrete Pavement Design and RehabilitationPurdue

University, School of Civil Engineering; Federal Highway Administration; Portland Cement Association; Transportation Research Board; Federal Aviation Administration; and Indiana Department of Highways.

Khazanovich, L., Ioannides, A.M., 1994. STRUCTURAL ANALYSIS OF UNBONDED CONCRETE OVERLAYS UNDER WHEEL AND ENVIRONMENTAL LOADS. Transportation Research Record.

Limouee, M., 2009. Verification Of Nyslab A Software For The Analysis Of Jointed Pavements 78.

Mechanistic-Empirical Pavement Design Guide: A Manual of Practice, 2020.

Ozbeki, M.A., Kilareski, W.P., Anderson, D.A., n.d. Evaluation Methodology for Jointed Concrete Pavements. Transportation Research Record 8.

Smith, K.D., Peshkin, D.G., Darter, M.I., Mueller, A.L., Carpenter, S.H., 1990. PERFORMANCE OF JOINTED CONCRETE PAVEMENTS. VOLUME V - APPENDIX B - DATA COLLECTION AND ANALYSIS PROCEDURES.

Tabatabaie, A.M., Barenberg, E.J., Smith, R.E., 1979. Longitudinal Joint Systems in Slip-Formed Rigid Pavements. Volume II. Analysis of Load Transfer Systems for Concrete Pavements. ILLINOIS UNIV AT URBANA-CHAMPAIGN DEPT OF CIVIL ENGINEERING.

Taghavi Ghalesari, A., Aguirre, N., Carrasco, C.J., Vrtis, M., Garg, N., 2020. Evaluation of the response from the rigid pavement analysis system (RPAS) program for the characterisation of jointed concrete pavements. Road Materials and Pavement Design 1–20. https://doi.org/10.1080/14680629.2020.1747522

Taghavighalesari, A., 2020. Development Of A Framework For The Calibration Of RPAS, A 3D Finite Element Analysis Tool For Rigid Pavements 203.

Aguirre, N., TaghaviGhalesari, A., Carrasco, C., 2019. A Comparison of Concrete Pavement Responses Using Finite Element Method with Foundation Springs and 3-D Solid Elements 81–90. https://doi.org/10.1061/9780784482452.009

Hu, S., Hu, X., Zhou, F., Walubita, L.F., 2008. SA-CrackPro: New Finite Element Analysis Tool for Pavement Crack Propagation. Transp. Res. Rec. 2068, 10–19. https://doi.org/10.3141/2068-02

Limouee, M., n.d. Verification Of Nyslab A Software For The Analysis Of Jointed Pavements.

Olowosulu, A.T., Kaura, J.M., Murana, A.A., Adeke, P.T., 2022. Development of framework for performance prediction of flexible road pavement in Nigeria using Fuzzy logic theory. Int. J. Pavement Eng. 23, 3809–3818. https://doi.org/10.1080/10298436.2021.1922907

Srikanth M R, Sri Jayachamarajendra College of Engineering, 2015. Study on Analysis of Flexible Pavement using Finite Element based Software Tool. Int. J. Eng. Res. V4, IJERTV4IS090865. https://doi.org/10.17577/IJERTV4IS090865

Wu, Z., Hu, S., Zhou, F., 2014. Prediction of stress intensity factors in pavement cracking with neural networks based on semi-analytical FEA. Expert Syst. Appl. 41, 1021–1030. https://doi.org/10.1016/j.eswa.2013.07.063

`TaghaviGhalesari, A., Aguirre, N., Carrasco, C., 2020. Application of Sensitivity Analysis in the Calibration Process of a Concrete Pavement Analysis Software, in: International Conference on Transportation and Development 2020. Presented at the International Conference on Transportation and Development 2020, American Society of Civil Engineers, Seattle, Washington (Conference Cancelled), pp. 51–60. https://doi.org/10.1061/9780784483183.006

Tayabji, S.D., Colley, B.E., 1986. Analysis of jointed concrete pavements.

Tia, M., Armaghani, J.M., Wu, C.-L., Lei, S., n.d. FEACONS III Computer Program for Analysis of Jointed Concrete Pavements 11.

Zokaei-Ashtiani, A., Carrasco, C., Nazarian, S., 2014. Finite element modeling of slab–foundation interaction on rigid pavement applications. Computers and Geotechnics 62, 118–127. https://doi.org/10.1016/j.compgeo.2014.07.003

Zokaei-Ashtiani, M.-A., Tirado, C., Carrasco, C., Nazarian, S., Bendaña, J., 2013. Modeling of Slab–Foundation Friction in Jointed Concrete Pavements under Nonlinear Thermal Gradient or Traffic Loads. Transportation Research Record 2367, 123–131. https://doi.org/10.3141/2367-13

Curriculum Vita

Samina Samrose was born in Dhaka, Bangladesh. Growing up in a third world country had broadened her vision towards necessity and instilled a keen awareness of finding appropriate solutions for the problems. As there were not ample resources for the solution, she started to think about using the ample resources she had back in her country. This curiosity to explore new solutions within the available limited resources led her to research. This passion became real when she was working as a research assistant in the renowned university Bangladesh University of Engineering and Technology (BUET). She earned her Bachelor of Science in Civil Engineering in 2019.

In pursuit of her enthusiasm and diving deeper into her field, she started her Masters in Civil Engineering in the University of Texas at El Paso (UTEP) in June 2021. During her time at UTEP, she embraced working as a teaching assistant and a researcher under Dr. Cesar Carrasco. Throughout her research period, she really enjoyed working under Dr. Carrasco’s supervision. During her masters program, she was devoted to working with MATLAB code and developed pavement analysis tool. Beyond her research endeavors, she has excelled in her academic courses. Currently she is doing her internship in TxDOT and hoping to start her PhD in Auburn University from Fall 2023.

Contact Information: ssamrose@miners.utep.edu