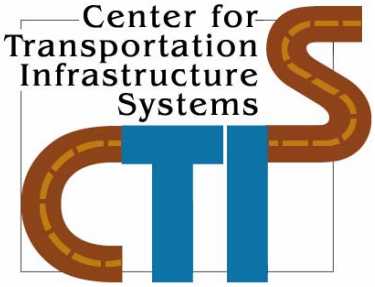

INTEGRATED PAVEMENT DAMAGE ANALYZER (INTPAVE) GRAPHICAL USER INTERFACE MANUAL PRODUCT 9-1502-01-P10

Conducted for:

Texas Department of Transportation,
Louisiana Department of Transportation and Development
New York Department of Transportation and
Federal Highway Administration

October 2006

Center for Transportation Infrastructure Systems
The University of Texas at El Paso
El Paso, TX 79968

This page replaces an intentionally blank page in the original.
-- CTR Library Digitization Team

INTEGRATED PAVEMENT DAMAGE ANALYZER (INTPAVE) GRAPHICAL USER INTERFACE MANUAL

The software to analyze pavement damage was developed into a stand alone executable program which has the following characteristics:

- Easy-to-use interface.
- Convenient data input.
- Integrated FEA model that calculates rutting and stresses for flexible pavements, developed in Matlab®.
- Newer and simpler graphical user interface that sends to and receives data from an external concrete pavement analysis program (JSLAB-2004).
- Graphical display of results.

The software may be used for the following purposes:

- Rutting calculation for a given flexible pavement structure and traffic loading scenario by means of finite element analysis.
- Fatigue cracking determination for a given pavement structure by means of Asphalt Institute procedure as described by the Guide for Mechanistic-Empirical Design (2004).
- Damage factor determination based on comparison of a truck against a standard truck defined by user based on equivalent axle load factors, rutting and fatigue cracking for flexible pavements, and equivalent axle load factors, fatigue cracking, faulting, and pumping index for rigid pavements.

MAIN WINDOW OVERVIEW

Executing the software prompts a main window that displays the main menu with a list of options, as shown in Figure 1. Below the main menu lies a set of buttons that correspond to basic items linked to actions that are also within the main menu. Putting the mouse over the action buttons will display a message showing the function of the button. The action of these buttons is as follows: New File, Open File, Save File, Analysis Type, Run FEA Analysis for a Single Truck, and Exit. Figure 2 presents the list of each button and the corresponding action.

Figure 1 – Main window.

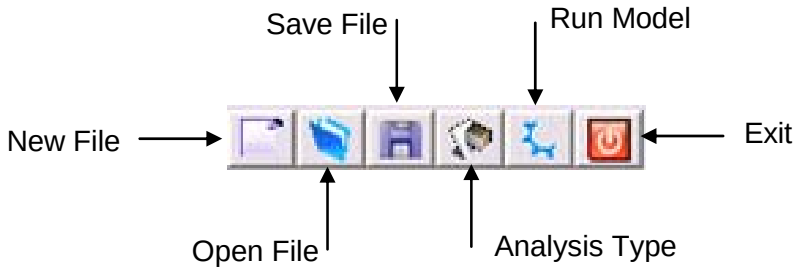

Figure 2 – Button activities.

The selection of any of the main menu options will open another list below them displaying related actions to their corresponding title. As an example, selecting the **File** menu will display a list consisting of **New**, **Open**, **Save** and **Exit** options. Creating a new file will automatically send the user to the next menu, **Input**, specifically to **Pavement Type**. The user will have to continue through the rest of the steps by filling the empty forms displayed. On the other hand, opening a file will load a previously saved file with information regarding the pavement structure, loading conditions and traffic rate. Such data may be edited by the user and saved into another file at any moment.

FLEXIBLE PAVEMENTS

Selecting the Input option from the main window will display a menu with options related to the pavement properties of the program. Such list is shown on Figure 3.

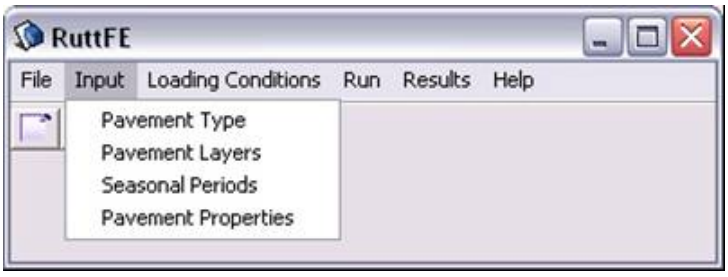

Figure 3 – Input menu for pavement structural properties.

The first item on the list is the **Pavement Type**. This item lets the user decide what kind of pavement to analyze, which can be either a flexible or a rigid pavement. A screenshot of the displayed window is shown in Figure 4.

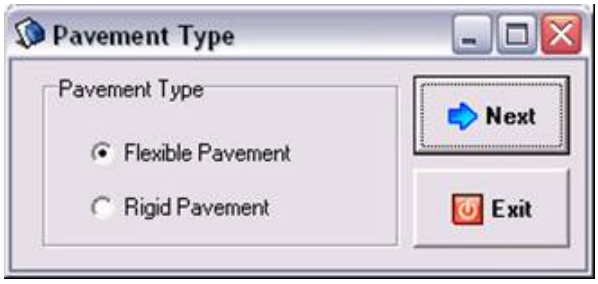

Figure 4 – Pavement type selection.

As it may be seen, each window has the option of going to the next screen, that is the next item on the menu, or closing the current active window letting the user skip menu items to select whatever item the user wants to edit if a file was previously loaded into the program. If the user selects a "Flexible Pavement" and clicks on the "Next" button, a new window titled **Pavement Layers** is displayed asking the user to enter the number of layers the pavement has in its structure, and the thickness and Poisson ratio of each layer. Figure 5 shows the displayed window.

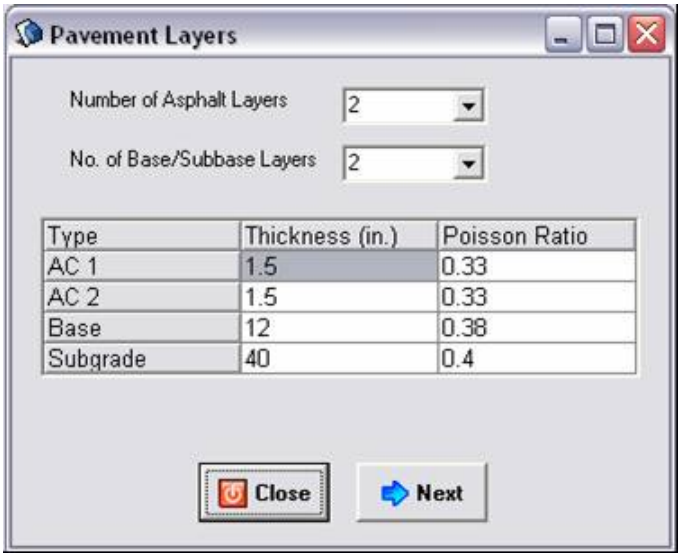

Figure 5 – Number of pavement layers and structural properties.

Other pavement properties have seasonal variation, and these were differentiated from the previous properties, as they vary according to the number of seasons specified by the user. The window prompted by the **Seasonal Periods** option allows the user to indicate the number of seasons within the analysis period and the season duration in days, as shown in Fig. 6.

Figure 6 – Seasonal period analysis.

The last item within the pavement structural properties input menu, **Pavement Properties**, lets the user enter into the program time dependent properties, such as the modulus of elasticity, α, μ, and nonlinear parameters k_2 and k_3. The window on Figure 7 shows sample properties on a selected layer.

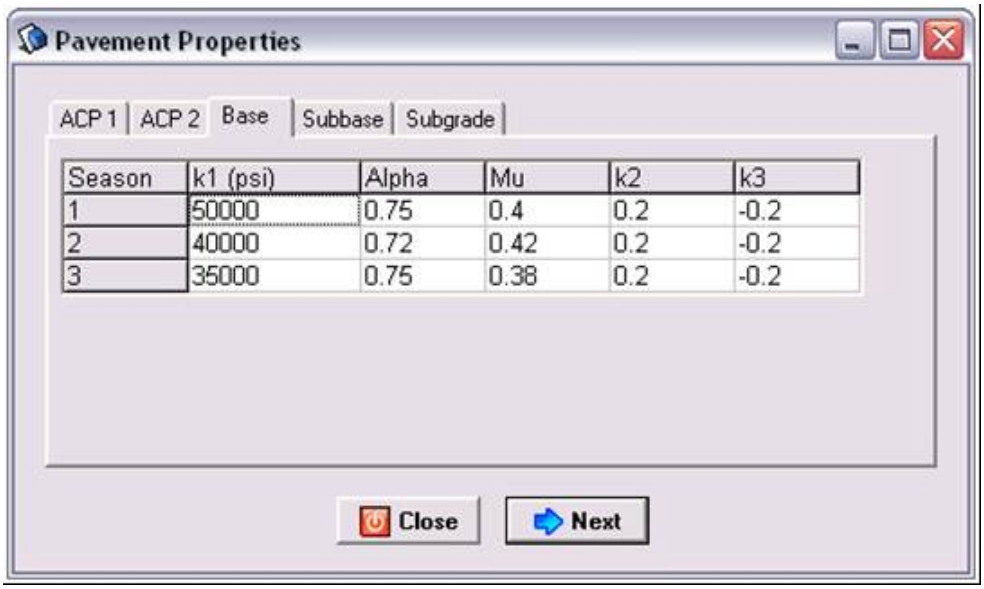

Figure 7 – Seasonal pavement properties

LOADING CONDITIONS

Selecting the **Loading Conditions** option from the main menu will display a submenu with options related to the traffic loading conditions. The first displayed item in the submenu is the Traffic option that prompts a window as shown on Figure 8.

The prompted window asks the user the required traffic information in terms of daily number of repetitions. Furthermore, it asks the user to specify if the analysis has to be done for a specific truck or if a comparative analysis is required in order to compare damage generated by a truck relative to the one generated by a standard truck. From this point, the analysis may follow two different paths and results are displayed in a different manner.

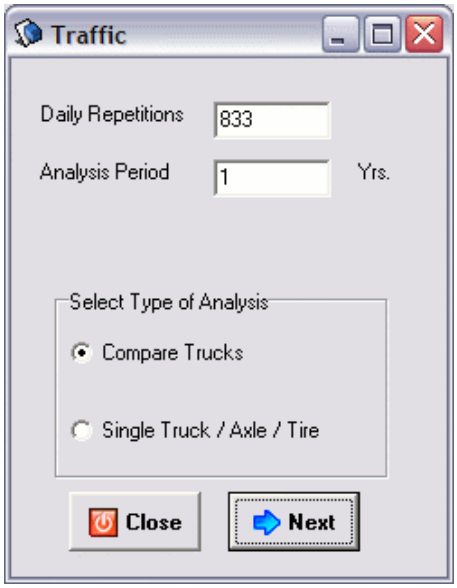

Figure 8 – Seasonal pavement properties.

SINGLE TRUCK, AXLE OR TIRE

The option **Single Truck** sends the user to the window shown in Figure 9. This window asks the user for information regarding the axle loading and other properties such as tire radius, pressure, axle spacing and tire spacing depending on the number of axles within the axle group.

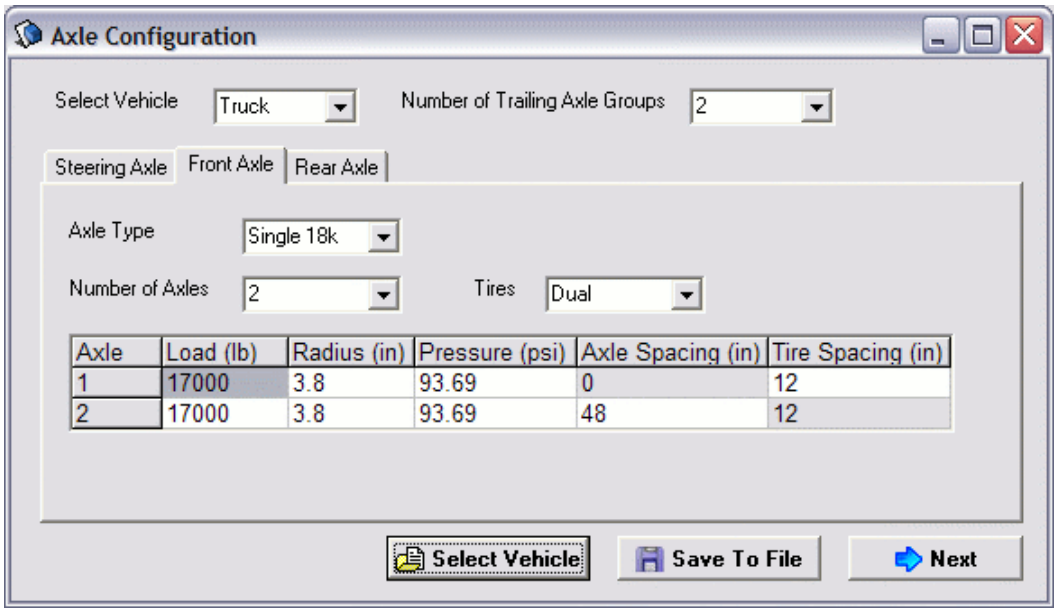

Figure 9 – Axle loading and properties

Furthermore, the user may analyze a single tire if that option is selected. The following window, shown on Figure 10, prompts the user to select an **Analysis Type** which can either be a linear or nonlinear analysis and each can be two- or three-dimensional. Two-dimensional modeling is

only available for single tires; if an axle or truck is analyzed, the 2D option is hidden. If the 3D option is selected, the user will get the option of saving the mesh and stresses for further reference.

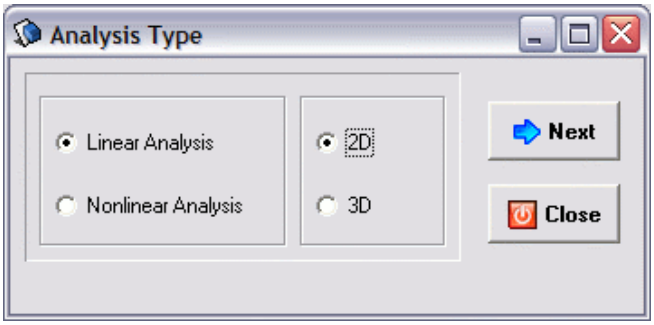

Figure 10 – Analysis type, selecting a 2D model.

2-D Model

Selecting a 2D model analysis, as shown in Figure 10, will lead to an axisymmetric finite element analysis, for this reason two-dimensional modeling is only available for single tires. Though the mesh may be viewable using Gmsh by opening Geometry2D.msh file, the analysis will not produce an output file with the stresses obtained from the analysis. Clicking on the **Next** button leads the user to the same steps that will be explained for the 3-D Model.

Gmsh is a three-dimensional finite element mesh generator with built-in pre- and post-processing facilities. This GNU General Public License program is used as a meshing tool called from the finite element analysis program and is used as a post-processor for viewing stress contours as well as the resulting mesh for multiple axles for three dimensional modeling.

3-D Model

Selecting a 3-D model analysis from the **Analysis Type** window, as shown in Figure 11, will lead to a three-dimensional finite element analysis. Though Figure 11 presents all options, the 2D option will not be available if an axle or a truck was selected when the configuration was defined.

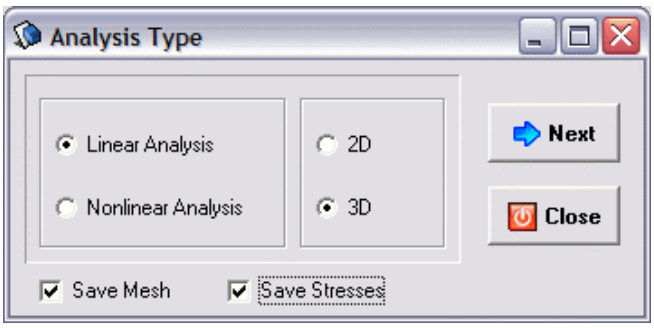

Figure 11 – Analysis type, selecting a 3D model.

Once an analysis type has been selected, another form titled **Summary of Input** prior to the analysis execution is displayed presenting a summary of all input information, as shown in Figure 12. All information previously input will be displayed containing the pavement seasonal properties, traffic information, as well as the axle configuration. In Figure 12, the information for a truck having a steering axle carrying 12,000 lbs, and two tandem axles with 17,000 lbs are displayed, similar to the information displayed on Figure 9.

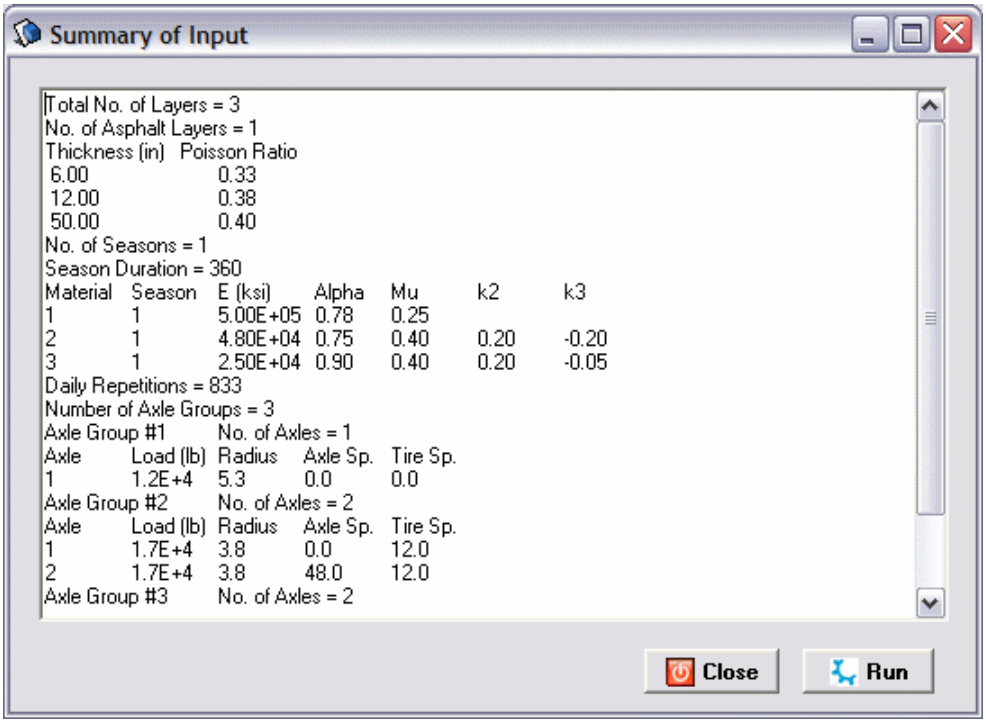

Figure 12 – Summary of input

Clicking the **Run** button will execute the finite element program based on the decision taken on the analysis type. The analysis will generate a mesh and stresses files for each axle and stored at the *Mesh* and *PosGen* folders created within the root folder, respectively. Files will be numbered with respect to the axle number accordingly, overwriting files from previous executions.

As soon as the execution process is finished, the results will be available to the user. Automatically, the **Rutting** window will be presented to the user; a sample rutting graph obtained from the results is depicted on Figure 13, which displays a graph with the total rutting as well as the contribution of each layer compared to the number of load repetitions. The number of repetitions available is the one the user selected for the analysis period.

By selecting **Fatigue Cracking** button a new window is displayed as shown in Figure 14 which displays a graph of fatigue cracking area with respect to load repetitions. The default method of calculating fatigue cracking is based on the Asphalt Institute formulation; however, constant parameters *k* may be changed to any value the user defines and a new graph will be displayed after the user clicks on the **Calculate** button. Similar to the rutting form, the fatigue cracking form has a summary button that displays the summary of results.

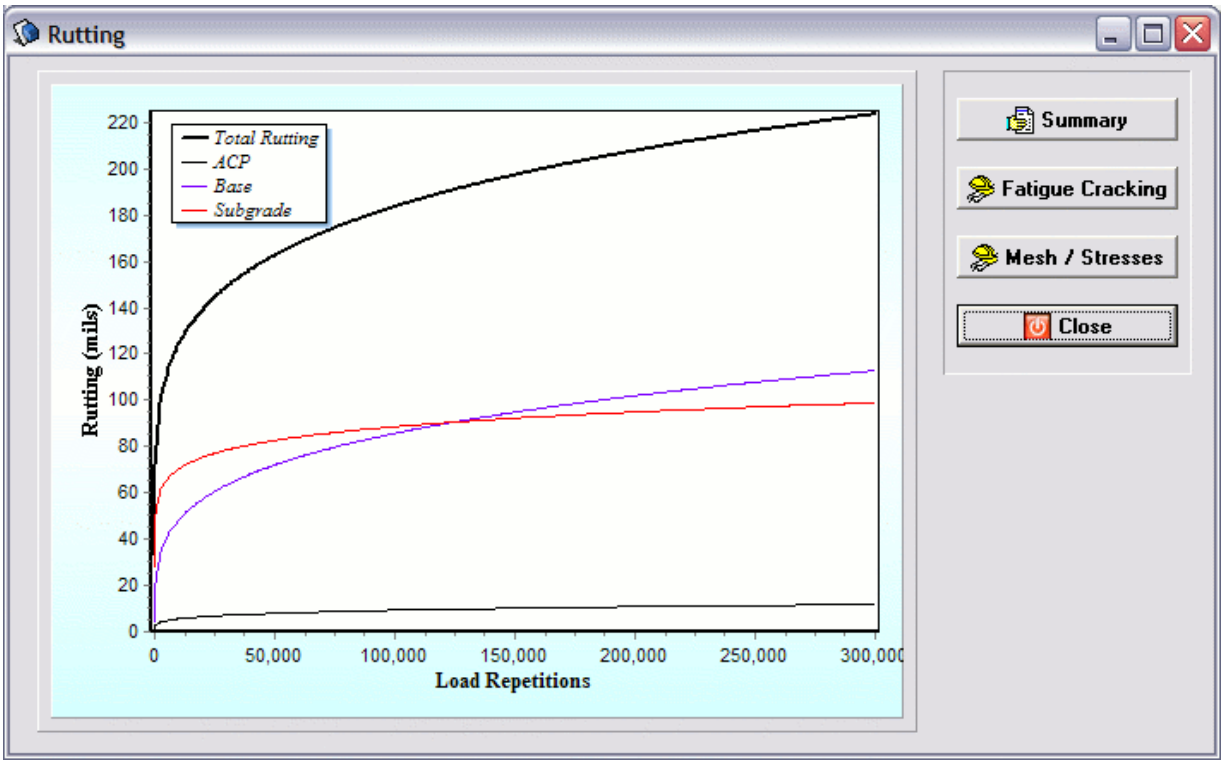

Figure 13 – Rutting vs. load repetitions.

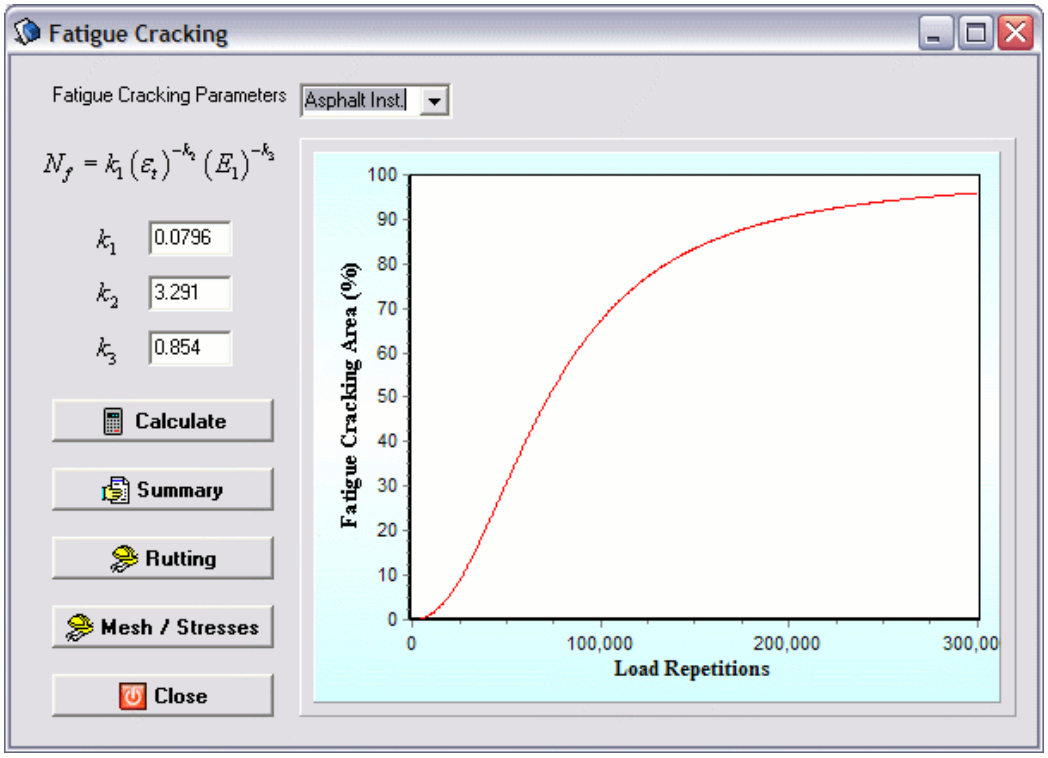

Figure 14 – Fatigue cracking results.

The user will have other information readily available that may be displayed by selecting the options shown in the **Rutting** or the **Fatigue Cracking** windows. The user has further access to view rutting or fatigue cracking results, mesh and stresses' contours or view a summary of the information obtained and displayed by just clicking on the buttons located next to the graph.

By selecting the **Summary** button a new window is opened that displays a field with a summary of the total rutting at the end of each season, as well as the contribution of each layer at the end of the total analysis period in terms of percentage, as shown in Figure 15. This information may be saved into an ASCII text file.

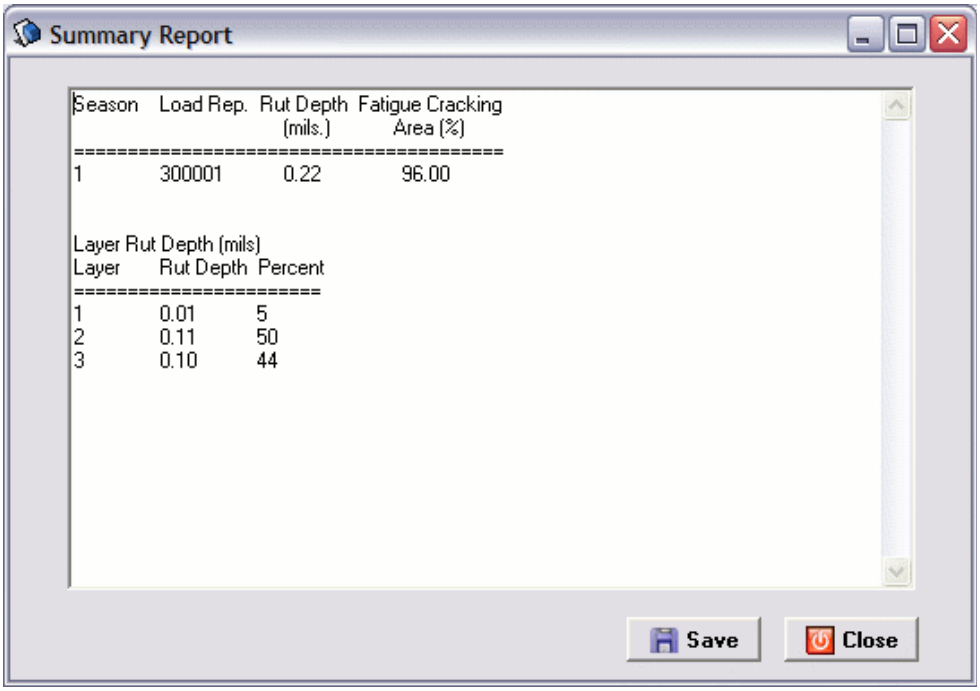

Figure 15 – Summary of results.

Clicking on the **Mesh / Stresses** button displays another window form that allows the user view the mesh of all axle groups by launching Gmsh that automatically open the axle group selected by the user from the drop-down menu, as shown in Figure 16. Similarly, the stress contours are available for all axle groups in the vertical, longitudinal and transversal directions by just selecting the desired options from the drop-down menus. Values of stresses are expressed in the same units as they were input into the pavement damage program, that unit consists of lbs/in^2 (psi). If the user did not selected to view this information using the check boxes in the **Analysis Type** window (see Figure 11), then the drop-down menus will display *Not Selected* and won't call these results.

Clicking on the Mesh and Stress **View** buttons will open windows as the ones that were shown in Figure 17(a) and (b), respectively.

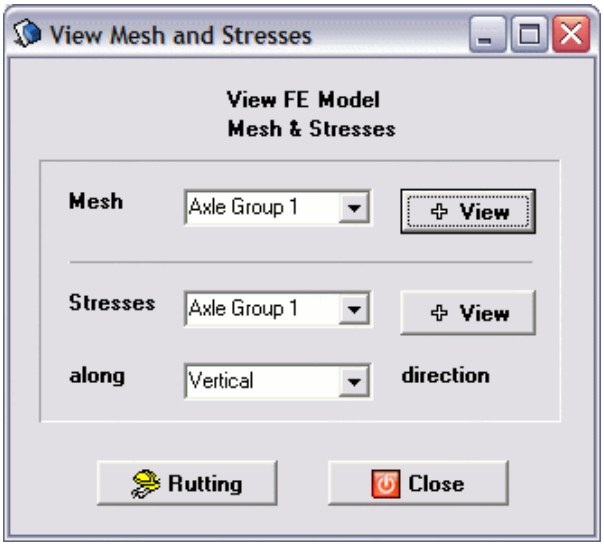

Figure 16 –View mesh and stresses.

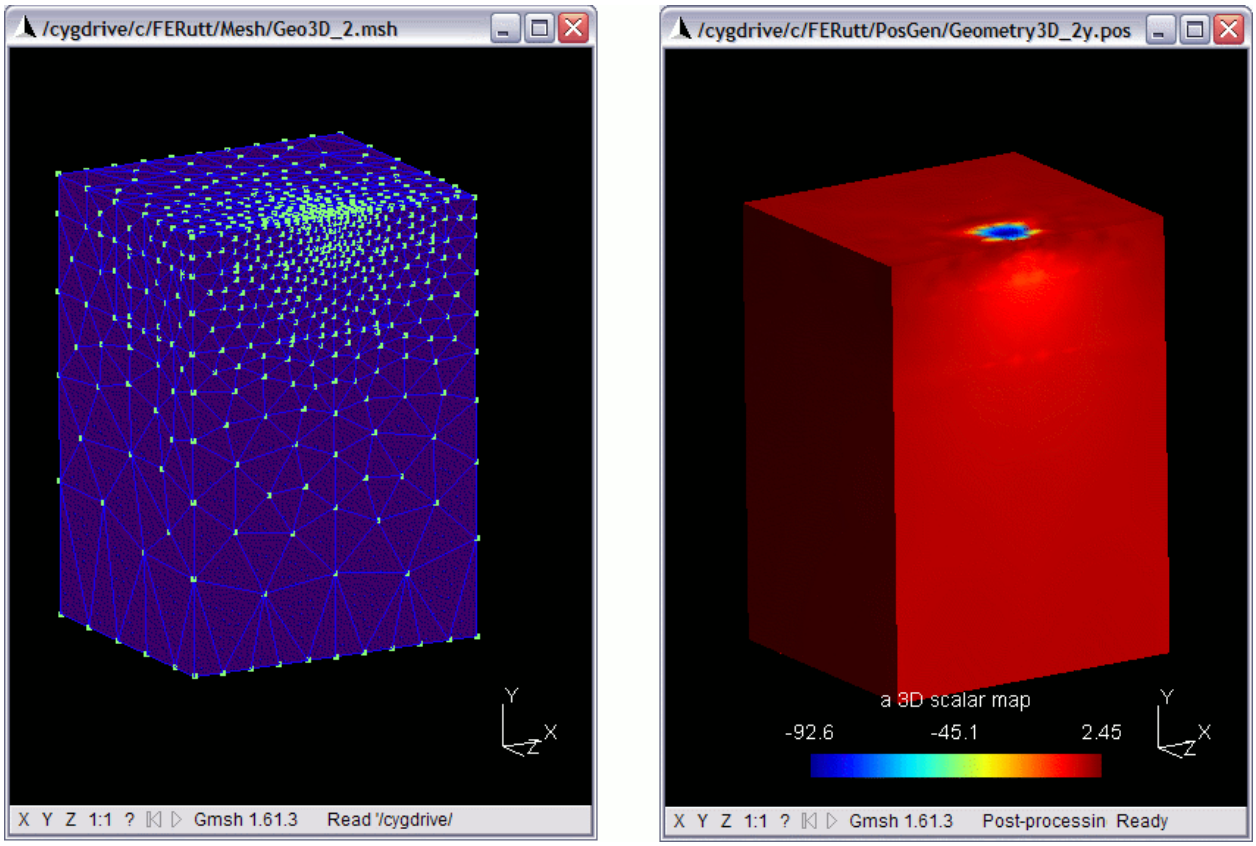

Figure 17 –View mesh (a) and stress contours (b).

COMPARE TRUCKS

The above description applies for the case where the user selects the analysis of a single truck, axle or tire. Now it is explained the other analysis type option, which consists on comparing damage caused by a truck based on the damage caused by a standard truck, both of them user defined by the user. The selection of comparing trucks is done at the traffic form (see Figure 8), clicking on **Compare Trucks** radio button sends the user to a different form titled **Truck Vehicle Selection** where a new and standard vehicle must be defined or selected from previously stored information, as shown in Figure 18. The form allows the user select saved vehicle information, or create and/or edit vehicle information for both trucks. Moreover, it asks the user to enter information regarding the empty vehicle weight of each of the vehicles, and automatically displays the gross vehicle weight (GVW) once the vehicle axle configuration has been created.

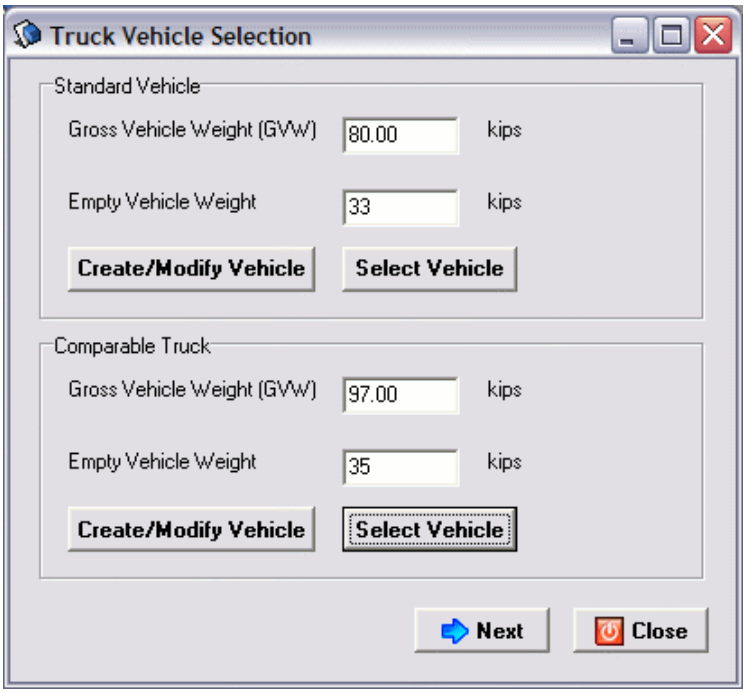

Figure 18 – Truck vehicle selection.

Selecting on the **Create/Modify Vehicle** button opens another window, titled **Truck Characteristics**, Figure 19, where the user can enter the number of axle groups besides the steering axle, the axle type of each group (i.e. single, tandem, tridem, etc.), the number of tires (single or dual), and the corresponding tire properties and distances for each configuration. The user will be able to save new or modified axle groups configurations at any time for later use, once all blank fields are filled. This window form is similar to the **Axle Configuration** form shown for the single truck analysis.

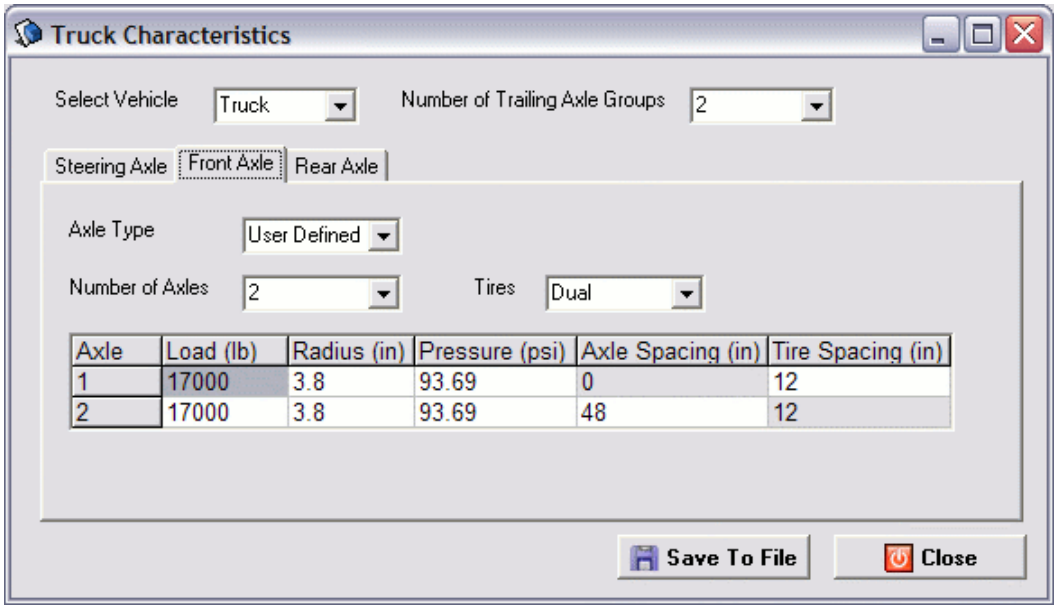

Figure 19 – Truck characteristics.

Once both vehicles are selected, the procedure requires the user to select an analysis type to compare vehicles, as shown in Figure 20. There are three analyses available, the first one consisting on a comparison based on the equivalent axle load factor for each truck (AASHTO), the second one based on rutting and the third one based on fatigue cracking. Each selection provides a relative damage based on the standard truck. In the case of rutting, it is necessary to provide what type of analysis the finite element code must perform, either a linear or nonlinear analysis.

Figure 20 – Compare vehicles form.

If the comparison based on the equivalent axle load factor for each truck is selected by clicking in the **AASHTO** button, a new window will be displayed as shown in Figure 21. This window

will provide the ratio between the equivalent axle load factors of the analyzed case truck divided by the selected standard truck.

Figure 21 – Damage factor based on AASHTO formulation.

On the other hand, a finite element analysis will be executed if a **Rutting** or **Fatigue Cracking** analysis is selected. For both cases, a three-dimensional model will be implemented. The window selecting the analysis type appears before the finite element code execution, as shown in Figure 22. In this comparison analysis, the user won't have the save mesh and stresses options.

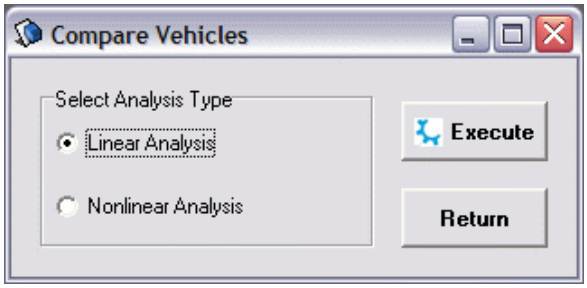

Figure 22 – Analysis type for comparison.

Selection of the rutting comparison will open a new graph displaying a tab control generating a graph depending on the selected tab. The first of the four displayed tabs offers the results of damage factors obtained from a ratio of load repetitions for both trucks required to attain a given amount of rutting, taking also into consideration the payload carried by the trucks too. The damage factor obtained is shown in a graph as the one in Figure 23.

The next two tabs display the rutting generated by the case truck and the selected standard truck based on the number of repetitions, as shown in Figure 24. This information is similar to the one obtained from a single truck analysis (see Figure 13).

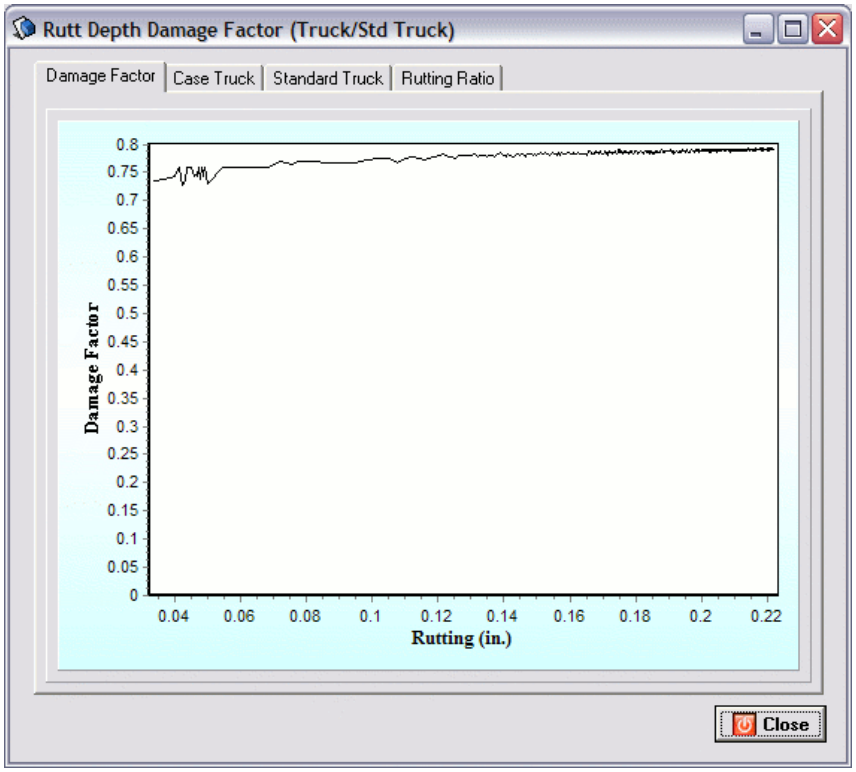

Figure 23 – Damage ratio based on rutting.

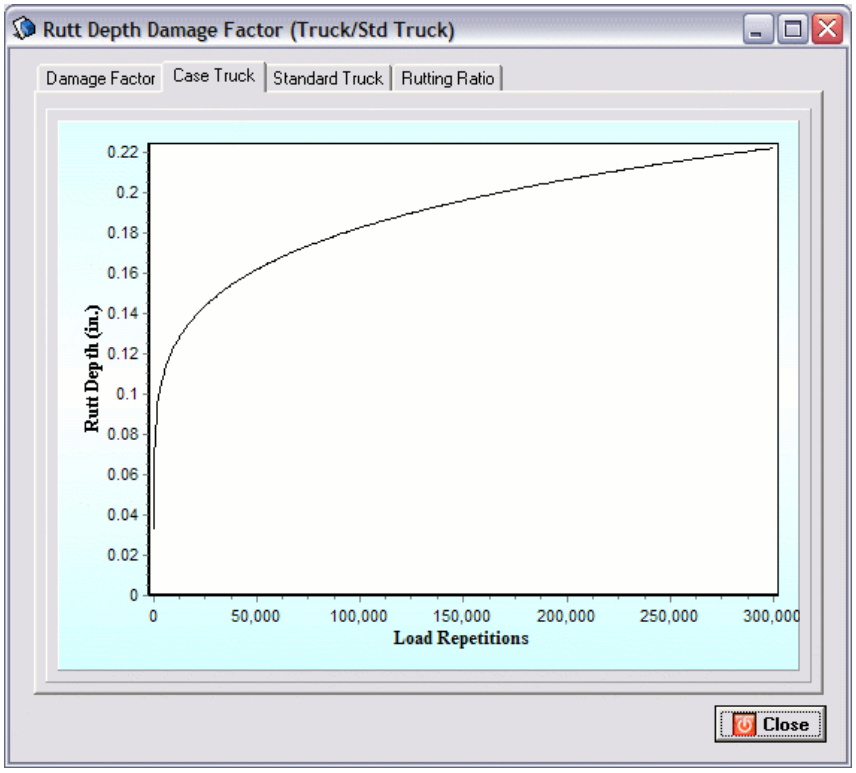

Figure 24 – Rutting vs. load repetitions for each truck.

Finally, the ratio of damage factor based on rutting of the new truck to the standard truck obtained for the first repetitions is displayed if the fourth tab is selected. This information directly compares the rutting generated for each truck pass, which is more suitable for a superheavy truck pass. A sample graph is shown in Figure 25.

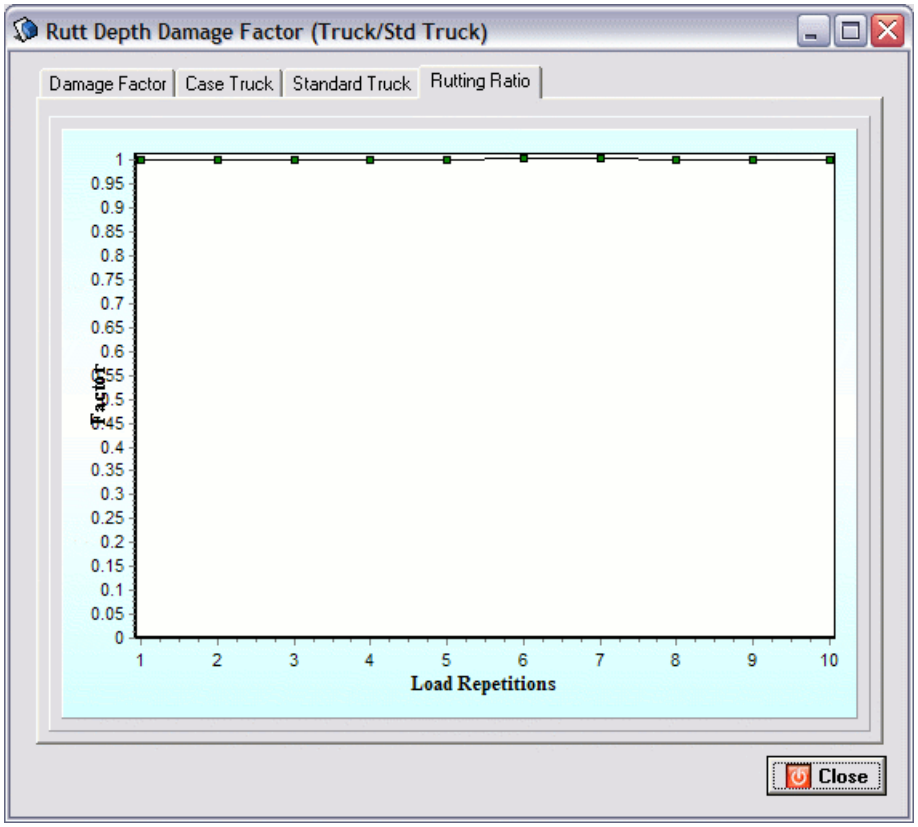

Figure 25 – Ratio based on rutting for the first truck passes.

Similarly, fatigue cracking comparison displays similar charts, with the same tab control options from the rutting window. Figure 26 presents a sample graph of the rutting comparison. Again, the first of the four displayed tabs offers the results of damage factors obtained from a ratio of load repetitions for both trucks required to attain a given amount of fatigued cracked area, the payload carried by the trucks is again considered. The damage factor based on fatigue cracking is shown in a graph as the one in Figure 26. Changing the *k* parameters and clicking on the **Calculate** button will generate a new graph.

Similarly to the rutting results, the fatigue cracking curves are available for each truck as shown in Figure 27. A direct ratio of case to standard truck based on fatigue cracking for the first load repetitions is also available.

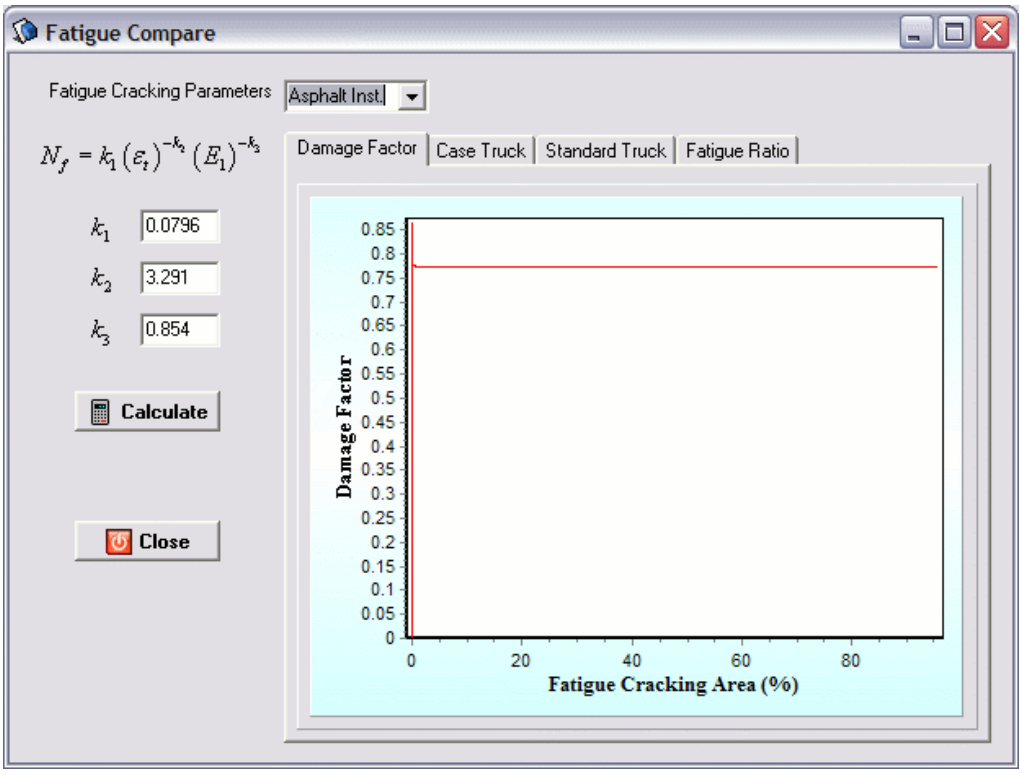

Figure 26 – Damage ratio based on fatigue cracking.

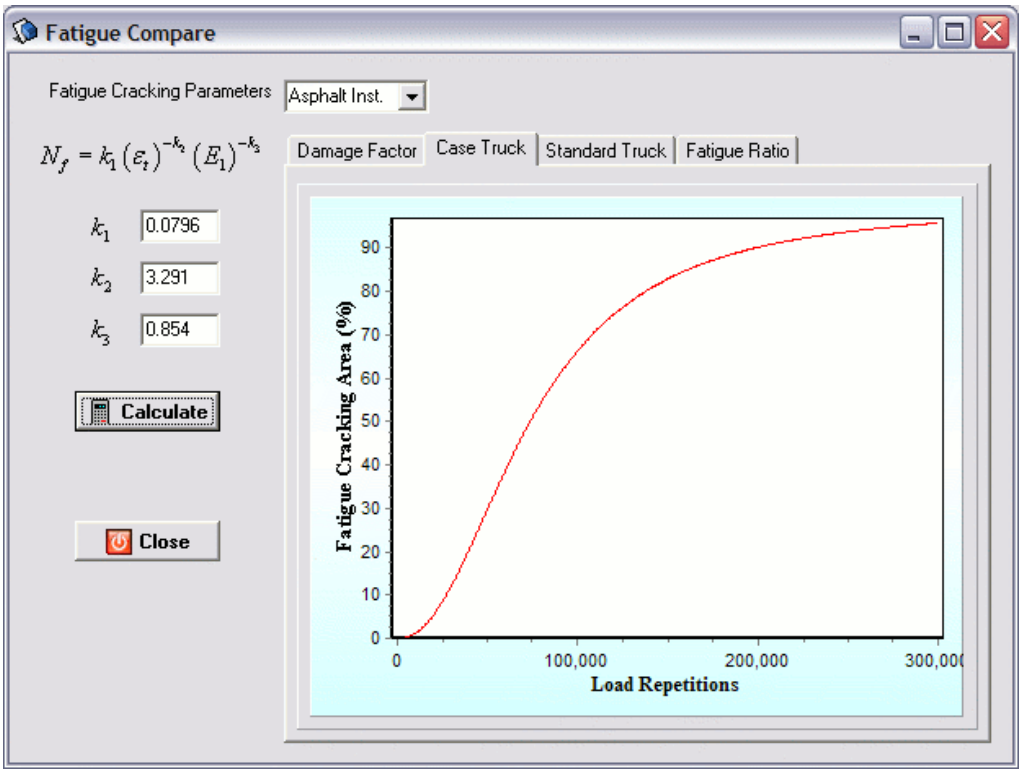

Figure 27 – Fatigue cracking based on load repetitions results per truck.

RIGID PAVEMENTS

The Integrated Pavement Damage Analyzer (IntPave) includes a section for distress analysis of rigid pavements. A graphical user interface was created to send pavement structure and traffic information to an external concrete pavement analysis program called JSLAB-2004, this program is capable of analyzing jointed pavement responses under a moving load from one end of a multiple-slab pavement to the other. The pavement damage analyzer incorporates distress models that make use of the resulting stresses obtained by JSLAB-2004 which are read by the pavement damage analyzer program to assess damage.

Selection of **Rigid Pavement** from the **Pavement Type** window (see Figure 4) leads the user through different windows. Starting with the **Rigid Pavement Properties** window, the user needs to input information related to the slab material (i.e. concrete's elastic modulus, Poisson's ratio, unit weight, equivalent temperature gradient and coefficient of thermal expansion), its thickness, and subgrade modulus. Default values are already displayed for the user to modify, as shown in Figure 28.

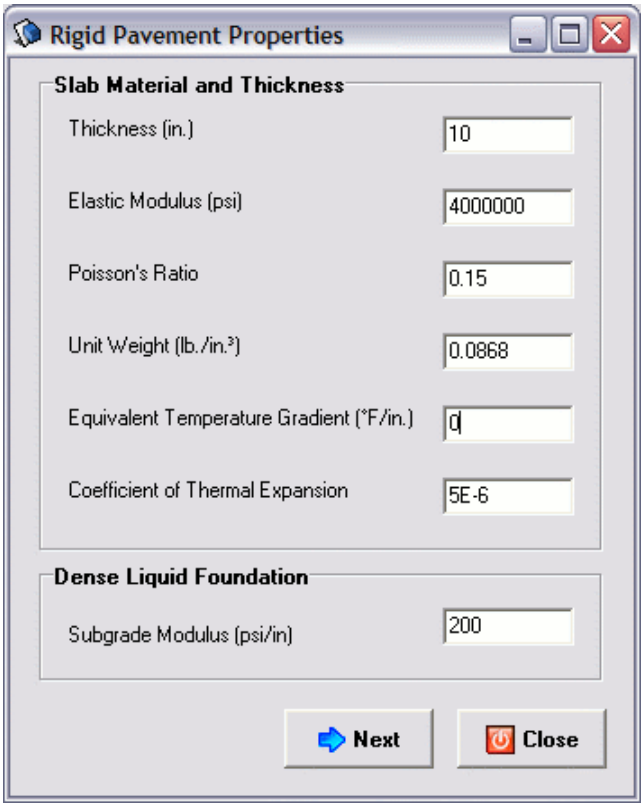

Figure 28 – Rigid pavement properties.

Next, slab properties must be input such as slab dimensions; this information is input into a window form as shown in Figure 29. It further requires to input desired wheel path distance to edge of slab. The program locates a second wheel path 6 ft above the selected wheel path. Moreover, a point within the slab must be defined; this point will be used if stresses want to be found at that specific point. Its location is based on the distance from the edge of slab

(perpendicular to wheel path direction) and the distance from the edge of slab where a joint is located, that is parallel the wheel path (see Figure 30).

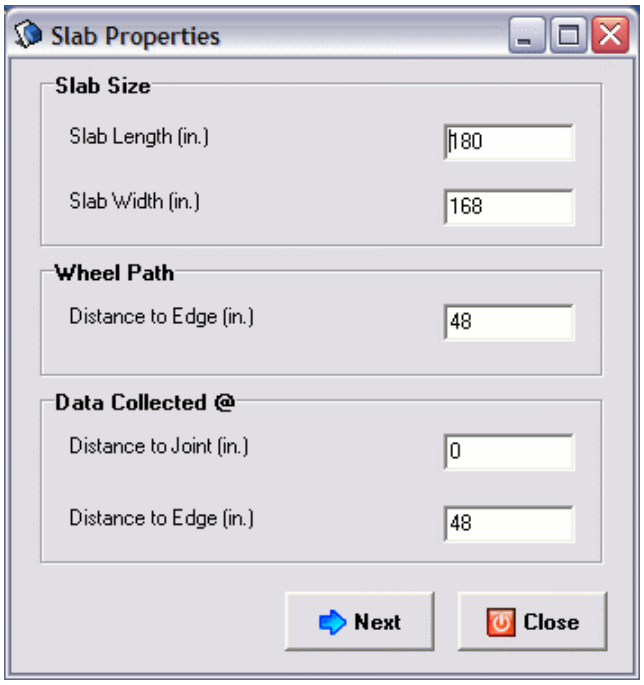

Figure 29 – Rigid pavement properties.

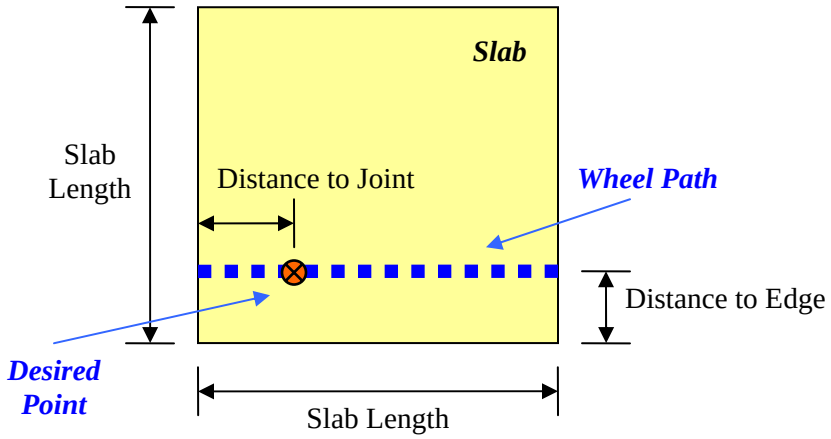

Figure 30 – Wheel path and selected point location within slab.

The following form titled **Joints** asks the user to enter information about the joints on the rigid pavement. Information about dowels and tiebars should be provided if required, as shown in Figure 31. Input fields activate depending if dowels exist or do not.

After this information is input, the form titled **Truck Vehicle Selection** is displayed in order to perform a comparative analysis. As in the case of the flexible pavement, a new and standard vehicle must be selected, as it was shown previously in Figure 18.

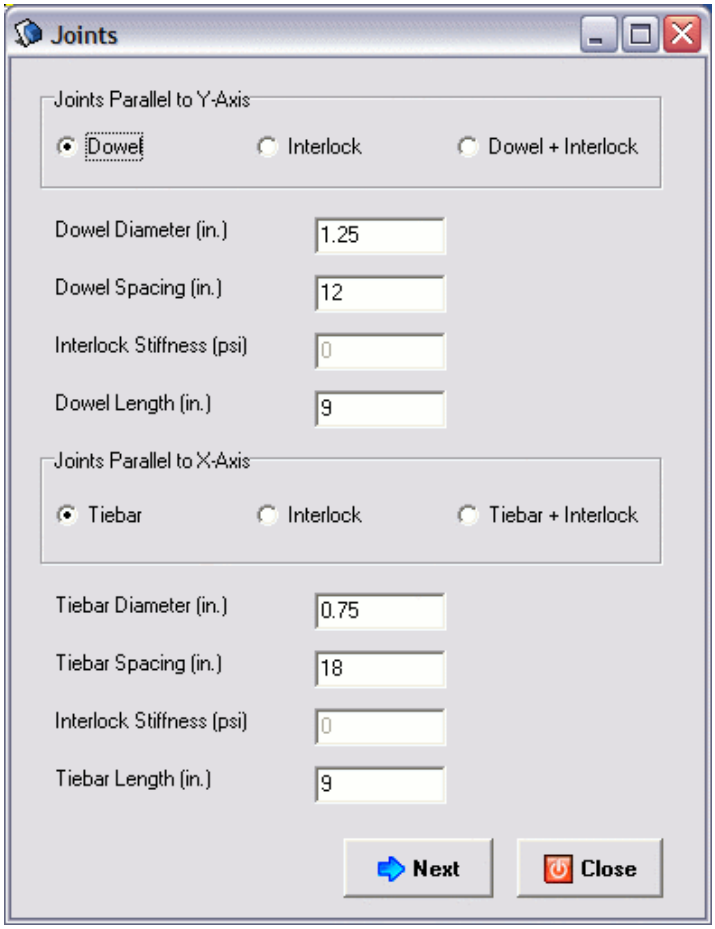

Figure 31 – Joints.

Again the user will have to edit the truck characteristics if necessary, opening a window as shown in Figure 32, which happens to be similar to the one displayed from the flexible pavement; however, a new field activates on the bottom part of window that asks the user to enter the distance between axle groups. This information is necessary in order to dimension the truck.

Once the necessary information is provided, the next window asks the user to select the type of analysis desired, shown in Figure 33. Among the options it is possible to compare damage of the new to the standard truck based on the equivalent axle load factor for each truck (AASHTO), or based on the stresses generated by each truck based on a dynamic analysis of the truck moving through the slab.

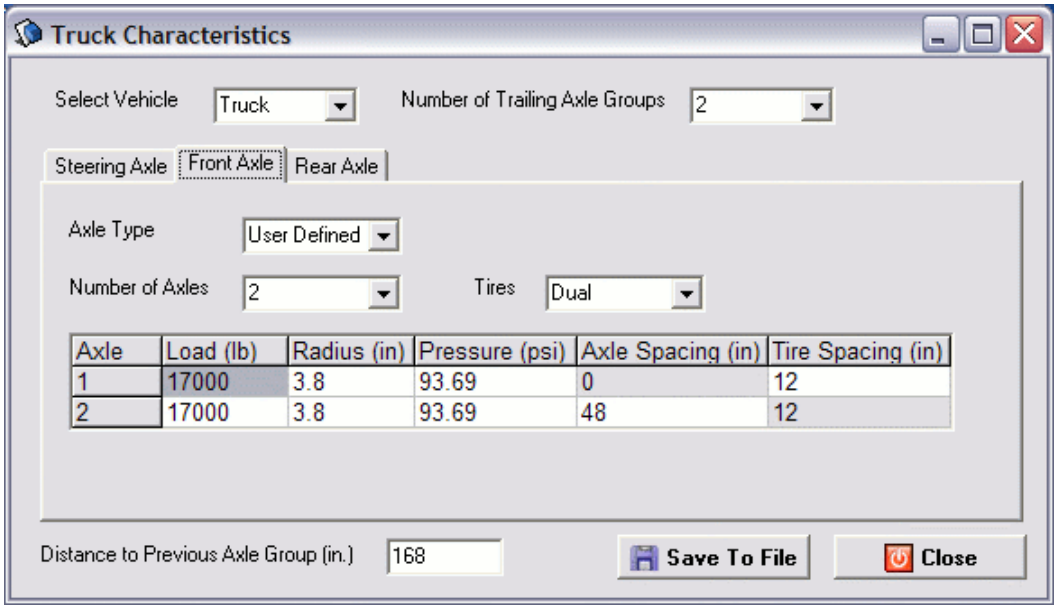

Figure 32 – Truck characteristics.

Figure 33 – Analysis type.

The first option, similarly to the flexible pavement, obtains the damage factor based on the equivalent axle load factor for each truck. By clicking in the **AASHTO** button, a new window will be displayed as shown in Figure 34. This window will provide the ratio between the equivalent axle load factors of the analyzed case truck divided by the selected standard truck.

Figure 34 – Damage factor based on AASHTO formulation.

Selection of the **Fatigue Cracking @ Point** button makes the program to perform a dynamic analysis to obtain the stresses in order to find fatigue cracking at the point that was previously specified by the user on the Slab Properties window (Figure 29). This will provide the damage factor based on fatigue cracking at a unique point as shown in Figure 35.

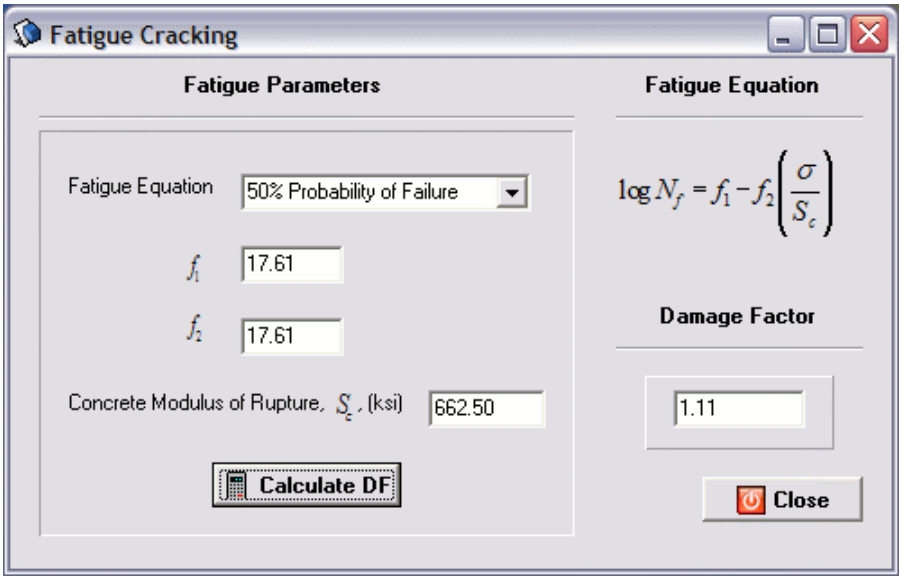

Figure 35 – Damage factor based on fatigue cracking at user defined point.

The user has the option of changing the fatigue equation as well as the *f* constants in order to obtain a new damage factor after pressing the **Calculate DF** button, as shown in Figure 36.

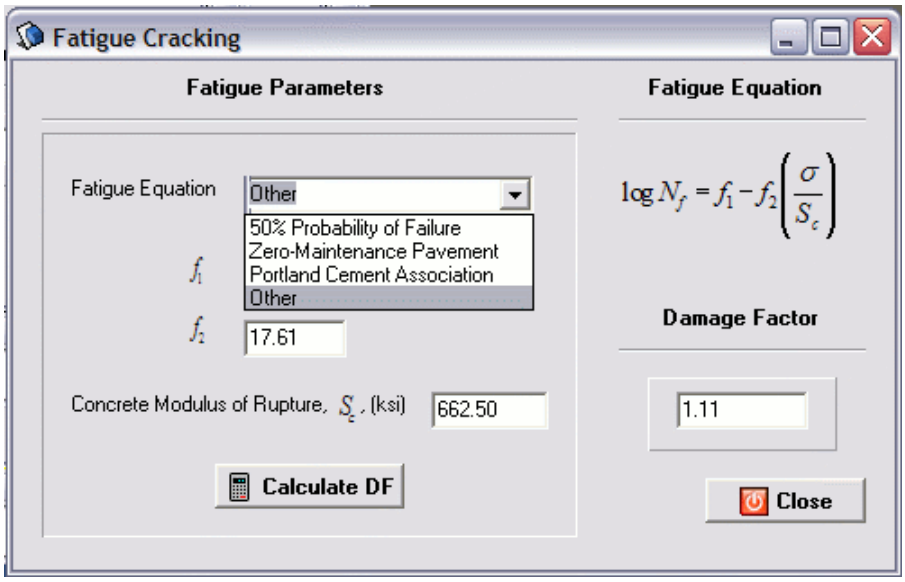

Figure 36 – Available fatigue equations.

On the other hand, it is possible to obtain the damage factor based on fatigue cracking at the point where maximum stress occurred in the slab by selecting the **Fatigue Cracking @ Slab** option. The maximum stress is obtained after each iteration as the truck passes through the slab, and a comparison is made based on the maximum of them. The same form is presented but a slightly different damage factor is obtained from the maximum stresses (Figure 37).

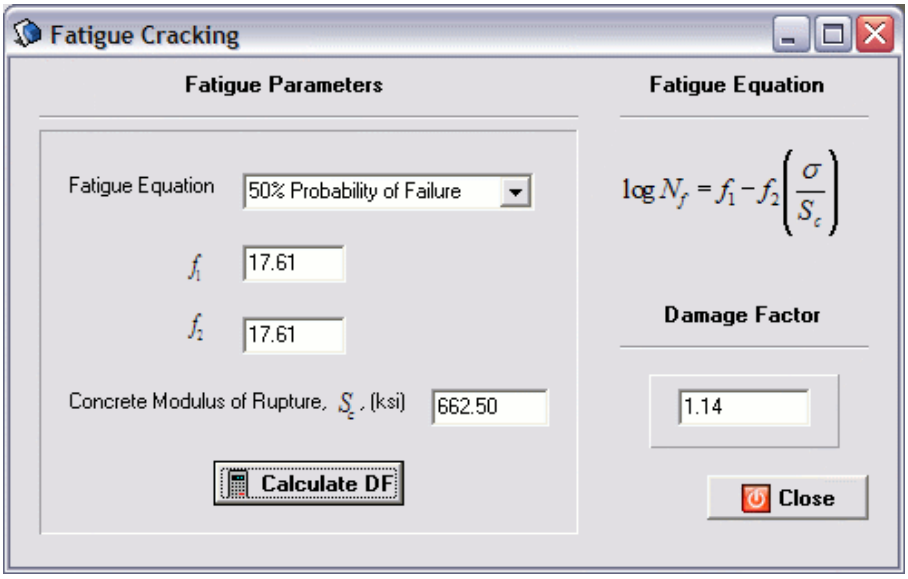

Figure 37 – Available fatigue equations.

Selecting the **Pumping** option presents the user another window that finds the damage factor based on a ratio of pumping indexes. The user is asked to provide the soil type from a drop-down menu, as well as the annual precipitation and freezing index in order to compute the pumping indexes, as shown in Figure 38.

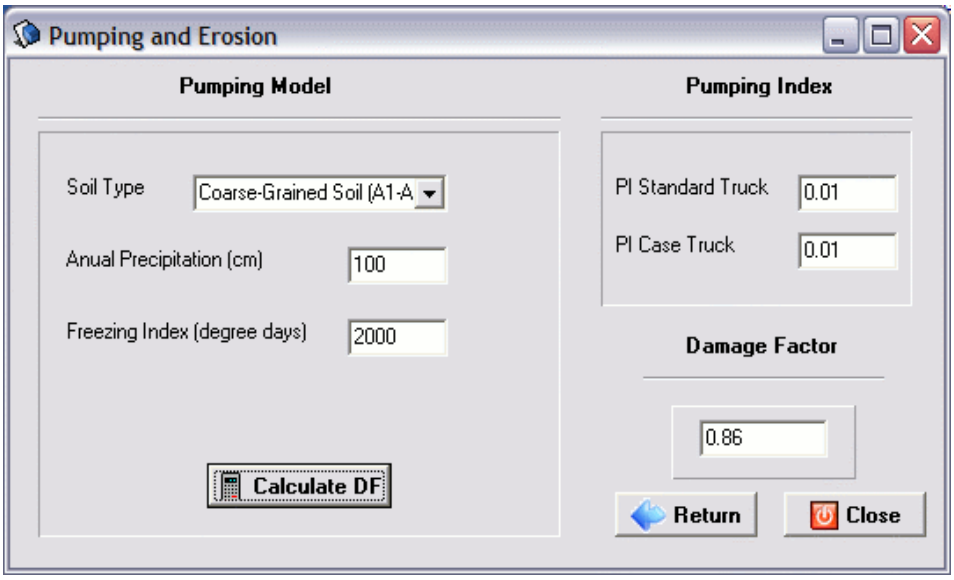

Figure 38 – Pumping model.

Finally, the last available option consists on the **Faulting** model shown in Figure 39. The damage factor is obtained based on the ratio of maximum faulting of the case truck by the standard truck observed for all dowels as the truck moves through the slab. If the pavement joints lacked dowels then the damage ratio is obtained from a faulting model that utilizes the maximum stresses found through the slab from the dynamic analysis of each truck; however, the user still needs to enter more information needed by the faulting model such as the soil type, subbase erodibility factor, edge support, drainage index and freezing index.

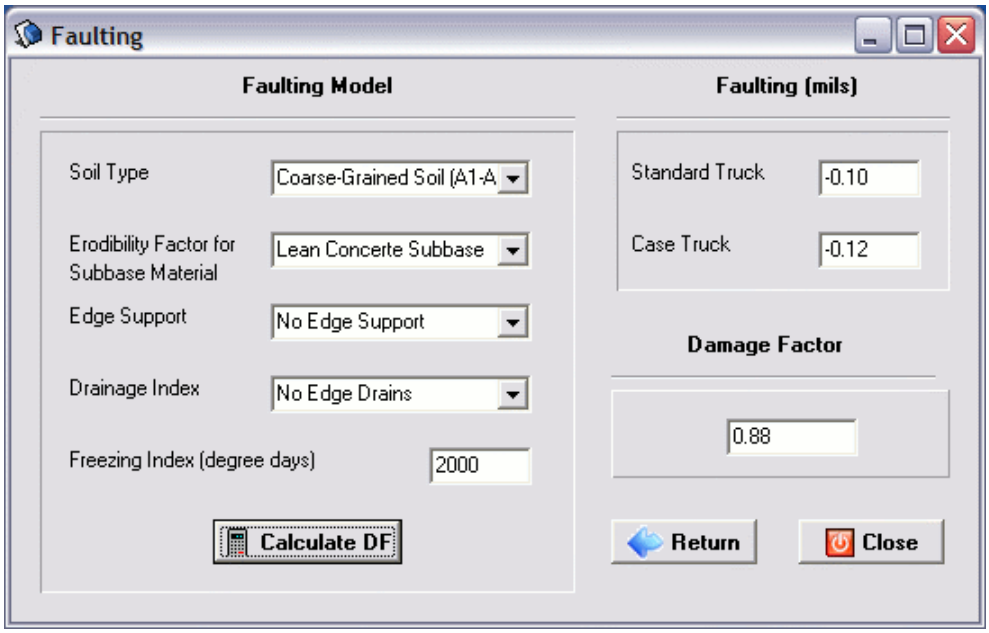

Figure 39 – Faulting model.

All analytical tools and available models used in IntPave, for both flexible and rigid pavements, are described into more detail in the Research Report FHWA/TX-05/9-1502-01-7.