HIGHWAY PROGRAMMING

Forth Stage

(2016-2017)

Ghada Gh. Masued

Highway programming

Syllabus

1) Introduction to Package Programs for Materials
2) Use of KENPAVE Main Screen
3) LAYERINP and SLABSINP
4) Steps to Design by KENPAVE Software
5) Application Example on KENLAYER Software
6) Use of Dowel Main Screen
7) Dowel Spacing Analysis Methodology
8) Corner Dowel Spacing
9) Centerline Dowel Removal
10) Alternative Dowel Bar Spacings
11) Mitigating Changes in Pavement Response
12) Introduction of Package Program Civil 3D for Material
13) Steps of Design the corridor object
14) Steps of design assembily
15) Steps of design alignments
16) Application Example on civil 3d Software
17) Methods of superelevation attainment
18) Steps of the corridor profile
19) Steps of design assemblies and subassemblies

20) Application Example on civil 3d Software

Textbooks

1. Dowel CAD Guidelines.
2. KENPAVE Design Guidelines.
3. Pavement Analysis and Design book.

(Huang, Y. H. (2004): ***"Pavement Analysis and Design"***, 2nd edition, Prentice Hall, Englewood Cliffs, New Jersey).

4. Auto CAD Civil 3D.
5. Handouts.

Introduction to Package Programs for Materials

KENPAVE: is a computer package for pavement analysis and design it applies to calculate stresses, strains and deformations in flexible and rigid pavements. The software KENPAVE which is developed in 1993 by YANG H. HUANG, the professor emeritus of civil engineering in the University of Kentucky, is used to model typical flexible and rigid pavement structures. The packages that form KENPAVE software are LAYERINP, KENLAYER, SLABSINP and KENSLABS. KENPAVE can use either English or SI unit.

The KENLAYER computer program applies only to flexible pavements with no joints or rigid layers. For pavements with rigid layers, such as PCC and composite pavements, the KENSLABS program should be used. The backbone of KENLAYER is the solution for an elastic multilayer system under a circular loaded area. The solutions are superimposed for multiple wheels, applied iteratively for non - linear layers, and collocated at various times for viscoelastic layers. As a result, KENLAYER can be applied to layered systems under single, dual, dual-tandem, or dual-tridem wheels with each layer behaving differently, either linear elastic, nonlinear elastic, or viscoelastic. Damage analysis can be made by dividing each year into a maximum of 12 periods, each with a different set of material properties. Each period can have a maximum of 12 load groups, either single or multiple. The damage caused by fatigue cracking and permanent deformation in each period over all load groups is summed up to evaluate the design life.

Use of KENPAVE Main Screen

MAIN SCREEN

Figure (1) shows the Main Screen of KENPAVE, consisting of two input boxes at the top and 11 command buttons at the bottom. The left three buttons are used for flexible or asphalt pavements, the right five for rigid or concrete pavements, and the remaining three for general purposes. Details on the use of these boxes and buttons can be viewed by clicking the Help button.

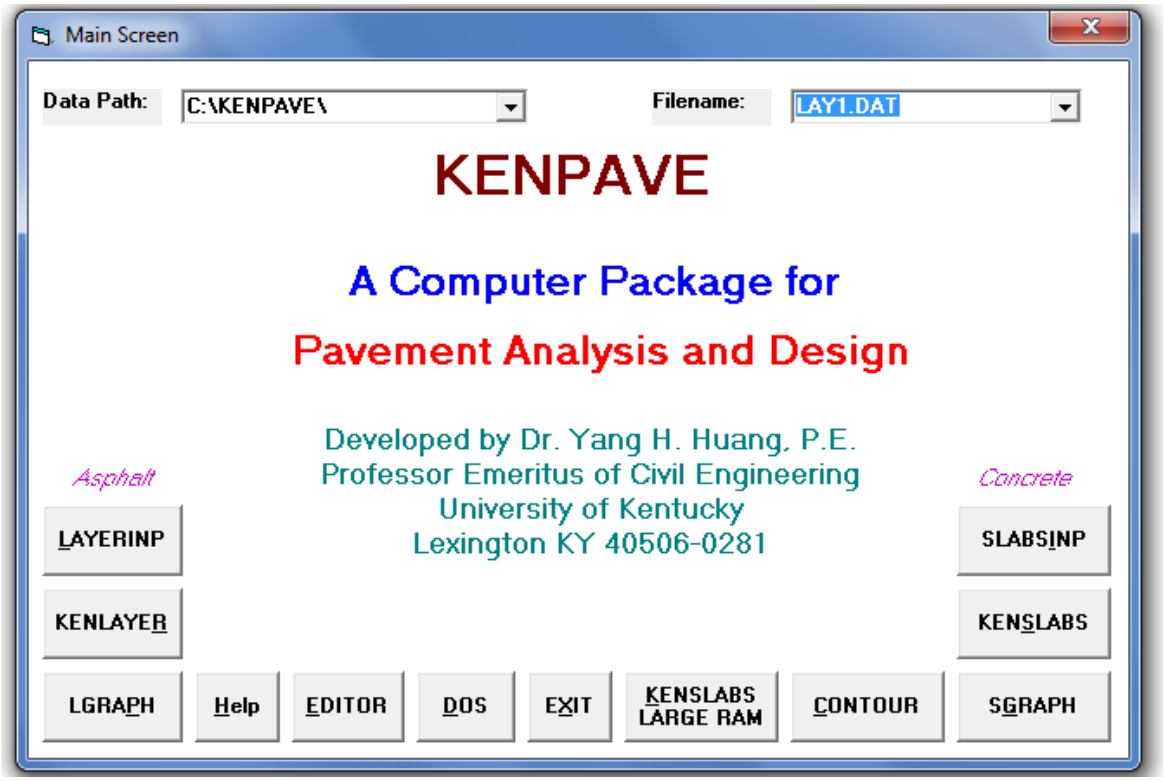


Figure (1): The Main Screen Capture of KENPAVE

USE OF MAIN SCREEN

The describing of the using of Main Screen is explained below:

a) Data Path and Filename

The default **Data Path** is shown in a drop-down list box on the upper left. Note the down arrow on the left of the box. When you use KENPAVE for the first time and

click the arrow, only the default data directory C:\KENPAVE\ is listed and highlighted. If your data file is in a different directory, you can type in the name of the directory. After the last \ is typed, all the data files in that directory with extension .DAT will be shown in the **Filename** drop-down list box on the right.

Simply click the data path you want and the content in the **Filename** box will also be changed.

For a new file to be created by LAYERINP or SLABSINP, you don't have to type a name in the **Filename** box because the file you created will be automatically given a name **Untitled.DAT** and you can later save it in the name you want. All data files must have an extension of DAT. The filename shown in the box will also be used in other files generated during the execution of KENLAYER or KENSLABS. Instead of the extension DAT, these files will have different extensions, such as TXT for the text file, LAY for running LGRAPH, and SLA for running SGRAPH and SLAB CONTOUR.

Note: If the same **Data Path** is used for asphalt and concrete be sure that the correct file is shown on the **Filename** box. If file LAY1.DAT is used to run KENSLABS, or SLA1.DAT to run KENLAYER, the program will not run or an error message will appear.

b) Command Buttons

1. HELP

Most of the screens have a Help textbox which explains the input parameters and the use of the form. However, some of the screens have a 'HELP' button or 'Help' menu the clicking of which will show a textbox on a separate screen.

2. EDITOR

EDITOR can be used to inspect, edit, and print data files. For users not familiar with the arrangement of data file, the use of LAYERINP or SLABINP as an editor is highly recommended. If you are an experienced user, you may like to make some simple changes in the data file by EDITOR. You must be careful that no mistakes be made in editing.

3. DOS and EXIT

DOS can be used to exit temporarily from Windows to MS-DOS. To return from MS-DOS to Windows, type EXIT. After all the desired analyses have been made, you can click EXIT to close KENPAVE.

4. LAYERINP or SLABSINP

LAYERINP or SLABSINP is used to create a data file before KENLAYER or KENSLABS can be run. The data are arranged in groups on a series of forms.

5. KENLAYER or KENSLABS

KENLAYER or KENSLABS is the main program for pavement analysis and can be run only after a data file has been established. The program will read from the data file and begin execution. During execution, some results will appear on the screen to let you know that the program is running.

If KENLAYER or KENSLABS stops unexpectedly during a run, the most probable cause is due to data error. You have to find out the location of the error and then correct the data in the data file. You cannot locate the error from the message on the screen. The easiest way is to open the .TXT file by using EDITOR. You can read down the file until you find out which input is wrong and make the necessary correction.

6. LGRAPH or SGRAPH

LGRAPH or SGRAPH can be used to graph the plan and cross section of the pavement together with some information on input and output.

7. Contour

This menu is useful to plot the contours of stresses or moments in the x or y direction.

LAYERINP and SLABSINP

Detailed instructions on the use LAYERINP and SLABSINP can be viewed on the computer screens and later repeated it. Figure (2) shows the location of input parameters on various screens in LAYERINP. All input parameters are in upper case; the menus in which the input parameters are located are in lower case, except for the first letter of each menu. The letter X indicates the existence of an auxiliary screen, which will appear automatically when a certain input parameter in the previous screen is typed. Figure (3) shows the location of input parameters on various screens in SLABSINP. There are 10 menus on the Main Screen of SLABSINP and seven auxiliary screens, as indicated by the seven X's.

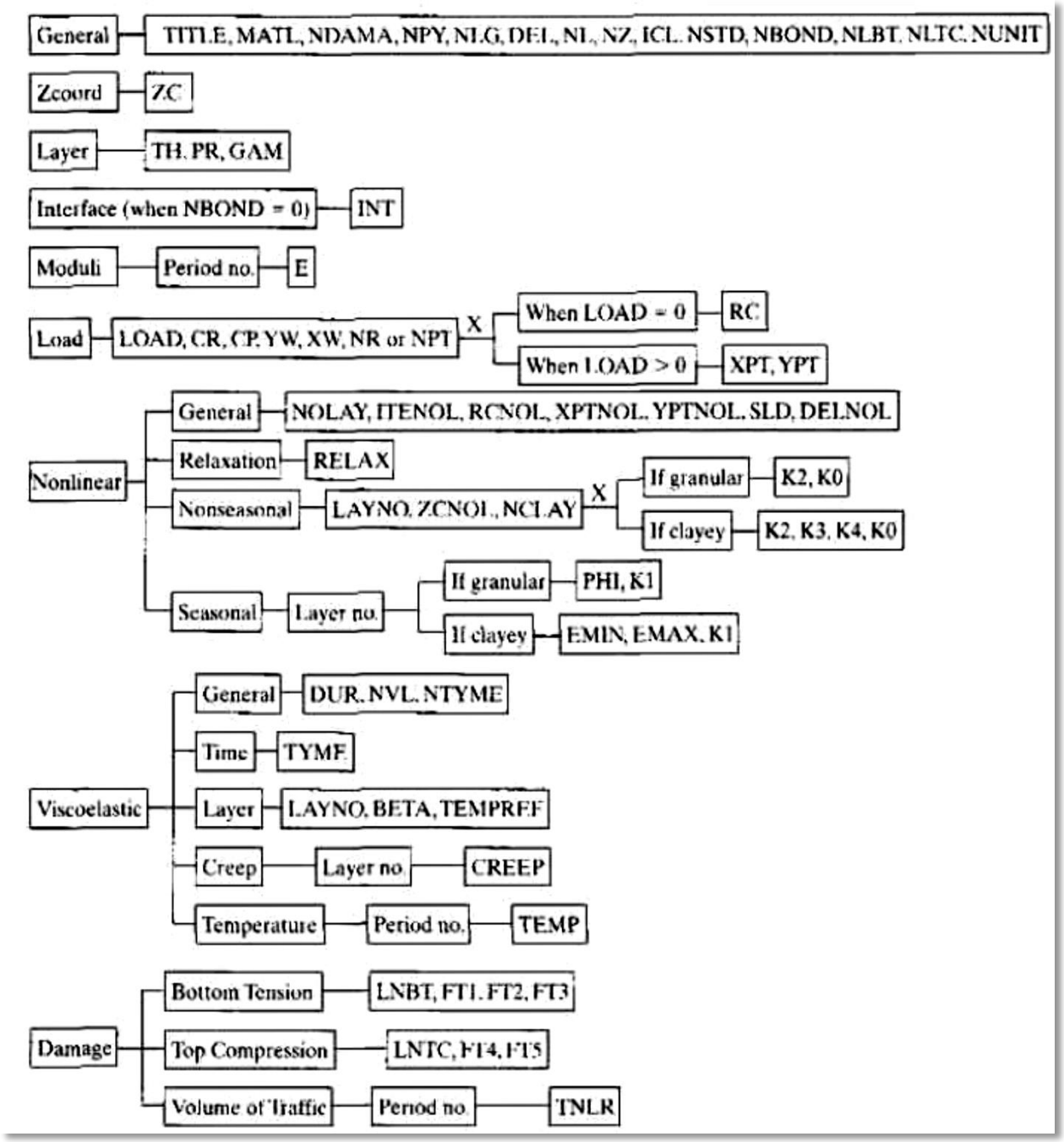


Figure (2): The Location of Input Parameters on Various Screens in LAYERINP

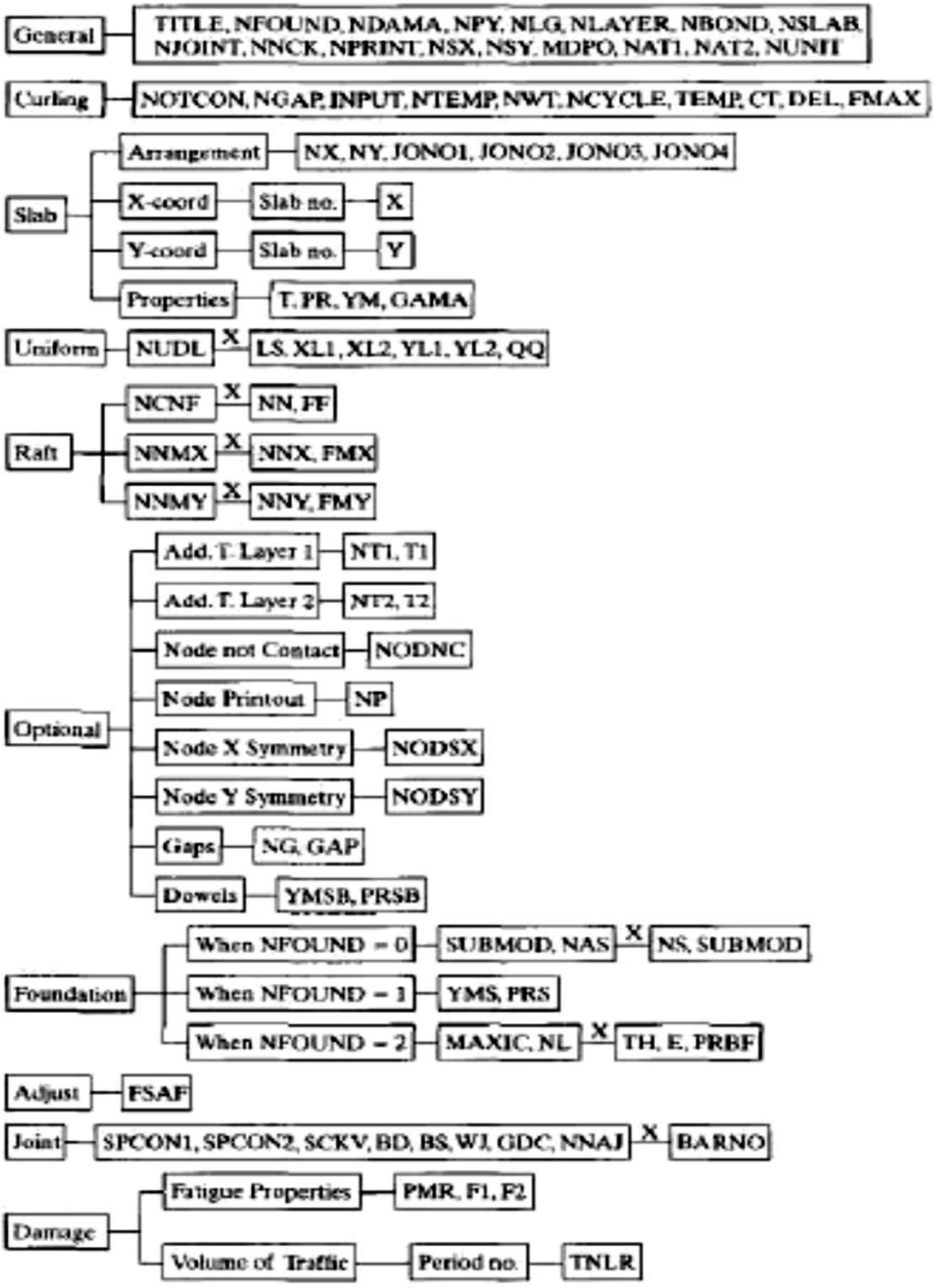


Figure (3): The Location of Input Parameters on Various Screens in SLAPSINP

STEPS TO DESIGN BY KENPAVE SOFTWARE

To learn using KENPAVE software to modeling flexible and rigid pavement structures, below general steps to describe this usage. The figures in each step is taken for KENLAYER software are example of using KENPAVE software for illustration only and the SLABSINP details found in explain of each step.

Step 1: Launch KENPAVE

This is the first step which concludes start operating the KENPAVE program. Main screen will appear and from this screen all buttons can be use easily and **Data path** & **Filename** appear clearly at the top of screen as show in Figure (4).

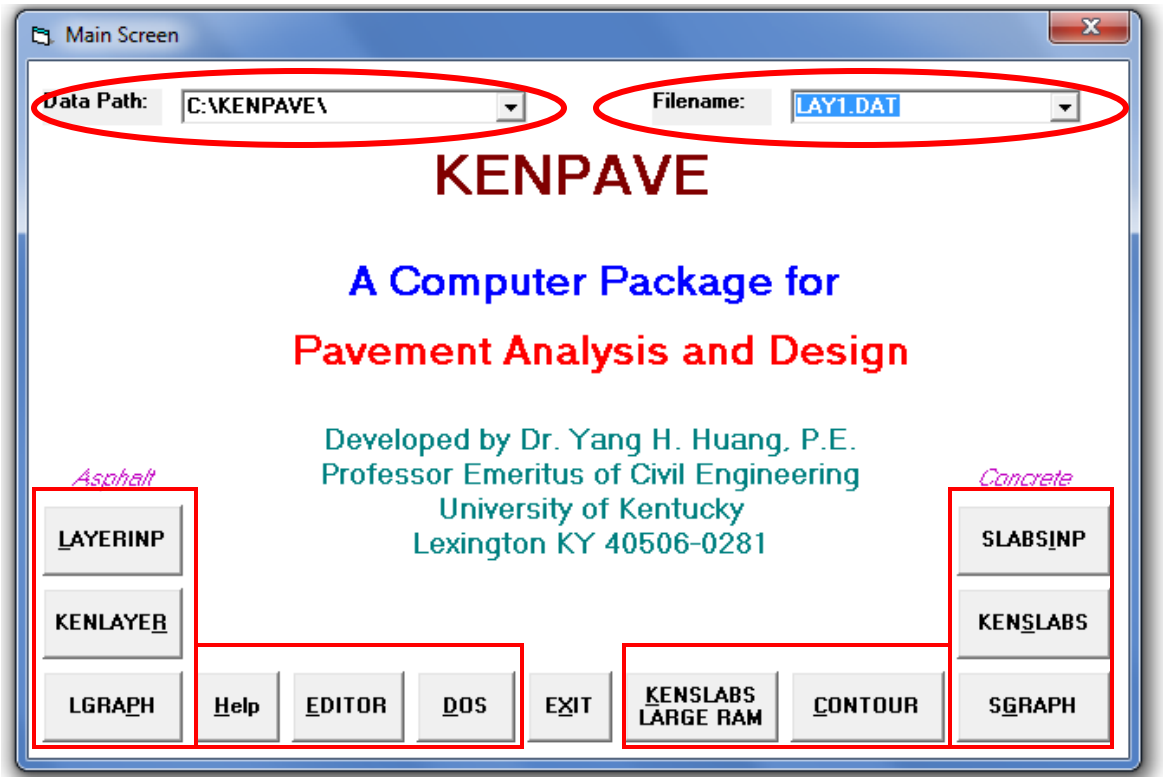


Figure (4): Details for Main Screen of KENPAVE Software

Step 2: Select LAYERINP or SLABSINP

Main menu of LAYERINP is shown in Figure (5). Main menu for LAYERINP and SLABSINP are used for creating and editing data file. This menu appears when the LAYERINP button on the Main Screen of KENPAVE is clicked. The data is divided into groups and can be found by clicking the appropriate menu.

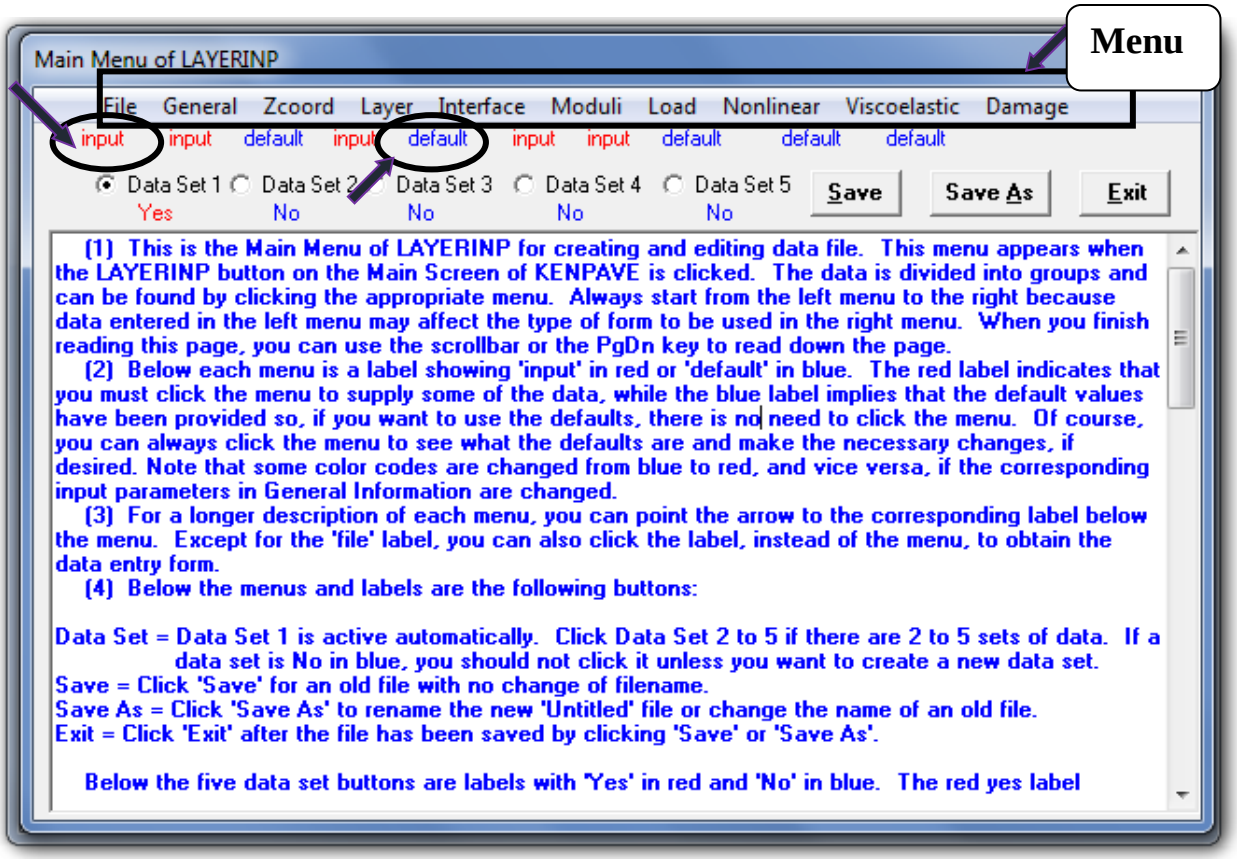


Figure (5): Main Menu of LAYERINP

Below each menu is a label showing **'input'** in red or **'default'** in blue. The red label indicates that you must click the menu to supply some of the data, while the blue label implies that the default values have been provided so, if you want to use the defaults, there is no need to click the menu. Of course, you can always click the menu to see what the defaults are and make the necessary changes, if desired.

For a longer description of each menu, you can point the arrow to the corresponding label below the menu. Except for the ***'file' label***, you can also click the label, instead of the menu, to obtain the data entry form. Below the menus and labels are the following buttons:

- **Data Set** = Data Set 1 is active automatically. Click Data Set 2 to 5 if there are 2 to 5 sets of data. If a data set is No in blue, you should not click it unless you want to create a new data set.
- **Save** = Click 'Save' for an old file with no change of filename.
- **Save As** = Click 'Save As' to rename the new 'Untitled' file or change the name of an old file.
- **Exit** = Click 'Exit' after the file has been saved by clicking 'Save' or 'Save As'.

Below the five data set buttons are labels with ***'Yes' in red*** and ***'No' in blue***. The red yes label indicates that the data set exists or must be provided by the user, while the blue no label indicates that no data set exits. For a ***new file***, the labels under Data Sets 2 to 5 are always No in blue. If these Data Set buttons are clicked, it will be changed to Yes in red and all the data in Data Set 1 will be copied into these new sets. The active data set is indicated by a black dot in the option button.

Step 3: Click on File on the toolbar

To set up a new data file click 'File' and 'New' and the filename 'Untitled' will appear on the label beneath 'File', Figure (6) . You can now proceed to input the necessary data.

To edit an existing file, you can click the filename in the drop-down list box to highlight it in the Filename box on the Main Screen. After clicking the LAYERINP button and then 'File' and 'Old', a dialog box showing a list of data files will appear.

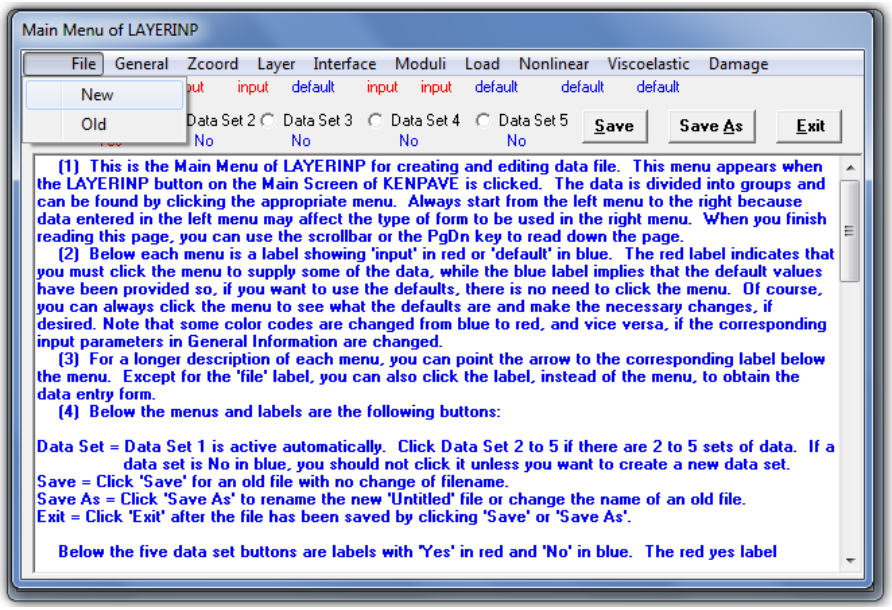


Figure (6): Data File in Main Menu

Step 4: Click on General on the toolbar

When the 'General' on the Main Menu of LAYERINP is clicked the screen as shown in Figure (7) will appears. You can override any of the default values by typing in a new value.

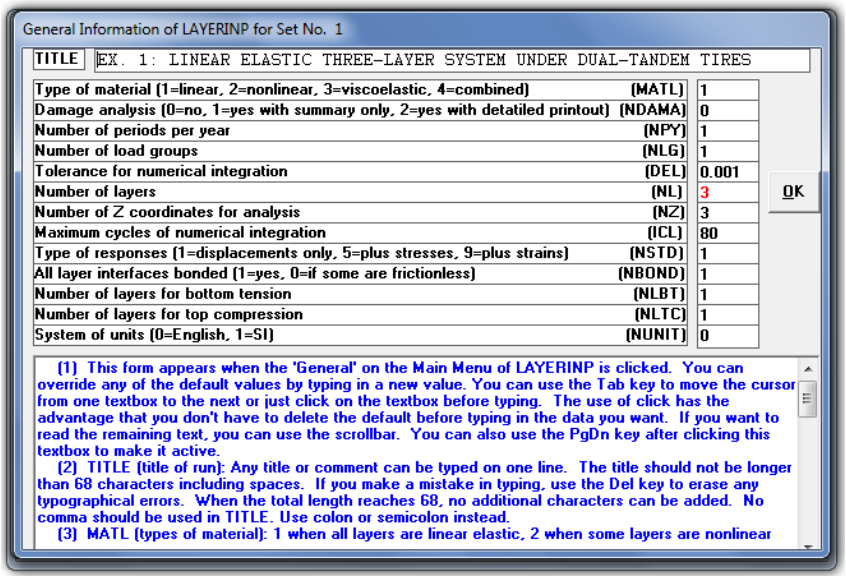


Figure (7): General Screen Capture

Details of 'General' screen are below:

- **TITLE** (title of run): Any title or comment can be typed on one line. The title should not be longer than **68** characters including spaces. No comma should be used in TITLE. Use colon or semicolon instead.
- **MATL** (types of material): 1 when all layers are linear elastic, 2 when some layers are nonlinear elastic and the remaining, if any, are linear elastic, 3 when some layers are viscoelastic and the remaining, if any, are linear elastic, 4 when some layers are nonlinear elastic, some are viscoelastic, and the remaining, if any, are linear elastic.
- **NDAMA** (damage analysis): 0 without damage analysis, 1 damage analysis with summary printout, and 2 damage analysis with more detailed printout.
- **NPY** (number of periods per year): Each year can be divided into a maximum of 12 periods for damage analysis. Even without damage analysis, NPY can be used to find the effect of layer moduli on pavement responses by assigning different moduli for each period.
- **NLG** (number of load groups): Axle loads can be divided into a maximum of 12 groups for damage analysis, each with different wheel loads and configuration.
- **DEL** (tolerance for numerical integration): A default of 0.001 implies an accuracy of 0.1%.
- **NL** (number of layers, maximum 19): The default NL is 3 which you probably would like to change, as indicated in red.
- **NZ** (number of vertical coordinates at which responses are to be computed): When NDAMA = 1, NZ can be left 0 because it will be determined by the program based on the number of locations at which damage analysis are to be made.

- **ICL** (Maximum number of integration cycles, 80 suggested): The actual number of cycles, as shown on the computer screen during execution should be smaller than ICL.
- **NSTD** (number of stresses, strains and displacement): 1 for vertical displacements only, 5 for vertical displacements and four stresses, and 9 for vertical displacement, four stresses and four strains. When damage analysis is performed, NSTD must be assigned 9.
- **NBOND** (types of interface between two layers): 1 when all interfaces are bonded, as is usually the case, and 2 when some interfaces are unbonded or frictionless.
- **NLBT** (number of layers with damage analysis based on the tensile strain at the bottom of asphalt layer).
- **NLTC** (number of layers with damage analysis based on the vertical compressive strain at the top of subgrade or other unbonded layers).
- **NUNIT** (system of units): 1 for SI units and 0 for English units.

English Unit	SI Unit	Conversion
Mile	Kilometer	1 mile = 1.609 Km
Foot	Meter	1 ft = .305 M
Inch	Centimeter	1 inch = 2.54 Cm
Pound	Grams	1 lb = 453.59 G
Ounce	Grams	1 oz = 28.35 G
Gallon	Liter	1 gallon = 3.79 L
Celsius	Kelvin	0 Degree C = 273.15 K

- Upon completion, click OK to return to the Main Menu of LAYERINP.

Step 5: Click on "Zcoord" on the toolbar

When the 'Zcoord' menu on the Main Menu of LAYERINP is clicked the screen as shown in Figure (8) will appears. The number of Z coordinates is equal to NZ, as specified in the 'General' menu.

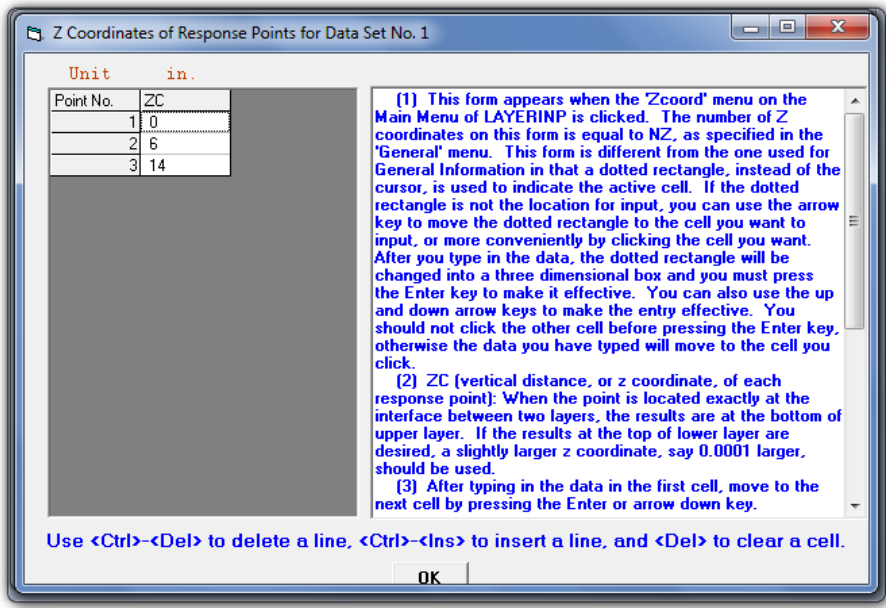


Figure (8): Z Coordinates Screen Capture

Details of 'Z Coordinates' screen are below:

- **ZC** (vertical distance, or z coordinate, of each response point): When the point is located exactly at the interface between two layers, the results are at the bottom of upper layer. If the results at the top of lower layer are desired, a slightly larger z coordinate, say 0.0001 larger, should be used.

You can delete a line, or one coordinate point, by first clicking anywhere on the line to make it active and then press the <Ctrl>-<Del> keys. You can add a new line, or one coordinate point, above any given line by first clicking the cell in the given line to make it active and then press the <Ctrl>-<Ins>. Upon completion, click the OK button at the lower right corner to return to the Main Menu of LAYERINP.

Step 6: Click on "Layer" on the toolbar

When the 'Layer' menu on the Main Menu of LAYERINP is clicked the screen as shown in Figure (9) will appears. The number of layers is equal to NL, as specified in the 'General' menu.

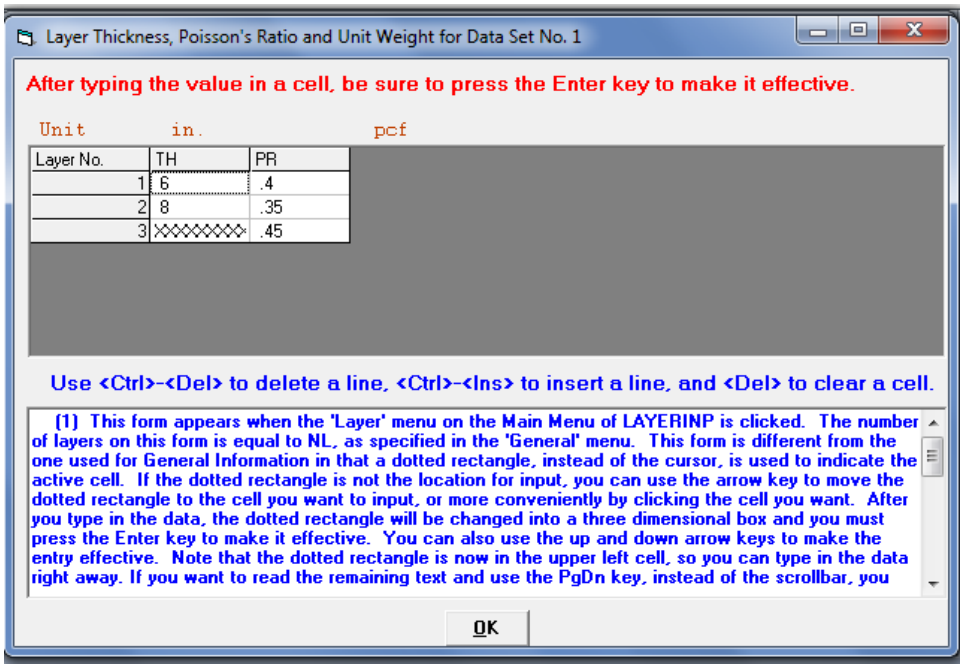


Figure (9): Layer Screen

Details of 'Layer' screen are below:

- TH (thickness of each layer): The last layer is infinite in thickness and need not be inputted.
- PR (Poisson's ratio of each layer): Suggested values are 0.35 for HMA and granular materials and 0.45 for fine-grained soils.
- GAM (unit weight of each layer): Suggested values are 145 pcf (22.8 kN/m^3) for HMA, 135 pcf (21.2 kN/m^3) for granular materials, and 105 pcf (19.6 kN/m^3) for soil. This column disappears when MATL = 1 or 3.
- After typing the data in a cell, be sure to press the Enter key to make it effective.
- If you want to add a line after the last line, you can change NL in the 'General' menu by adding 1 and a blank line will appear as the last line. After completing this form, click OK to return to the Main Menu of LAYERINP.

Step 7: Click on "Moduli" on the toolbar

When the 'Moduli' menu on the Main Menu of LAYERINP is clicked the screen as shown in Figure (10) will appears. The number of periods is equal to NPY, as specified in the 'General' menu. The 12 buttons, as shown in Figure (10), indicates that a maximum of 12 periods may be used. However, only the periods being actually specified are marked with the period number on the button. Below the period button is a label showing 'input' in red, indicating that there are no defaults and you must enter the elastic modulus for each layer. After the data are entered, the letter 'input' will be changed to 'done'. Now you can click the Period1 button to enter the data. After the data for all periods are entered, as indicated by 'done' under each period button, click OK to return to the Main Menu of LAYERINP.

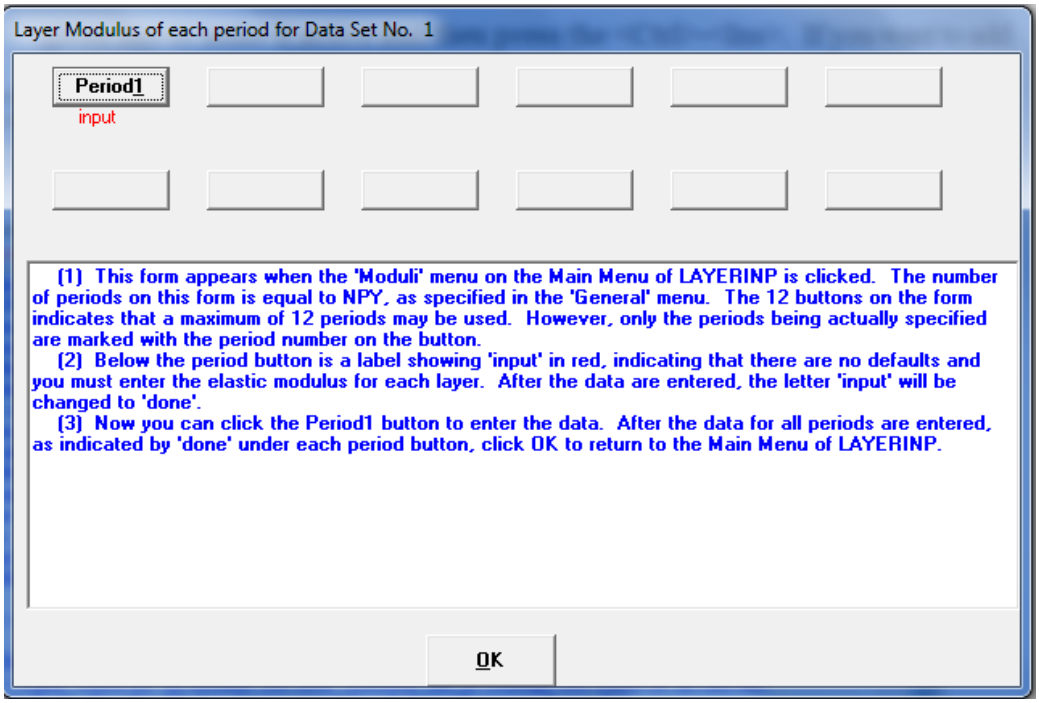


Figure (10): Layer Moduli Screen

Step 8: Click on the "Period1" button

When the period button on the Layer Modulus of Each Period is clicked the screen as shown in Figure (11) will appears. Details of 'Layer Moduli for Period' screen are below:

- **E** (elastic modulus of each layer): Use as the assumed modulus for the first iteration when the layer is nonlinear. If more convenient, you can enter the modulus in exponential form such as 1.234E5. Assign 0 or any value for viscoelastic layer. Upon completion, click the OK button to return to the Layer Modulus of Each Period.

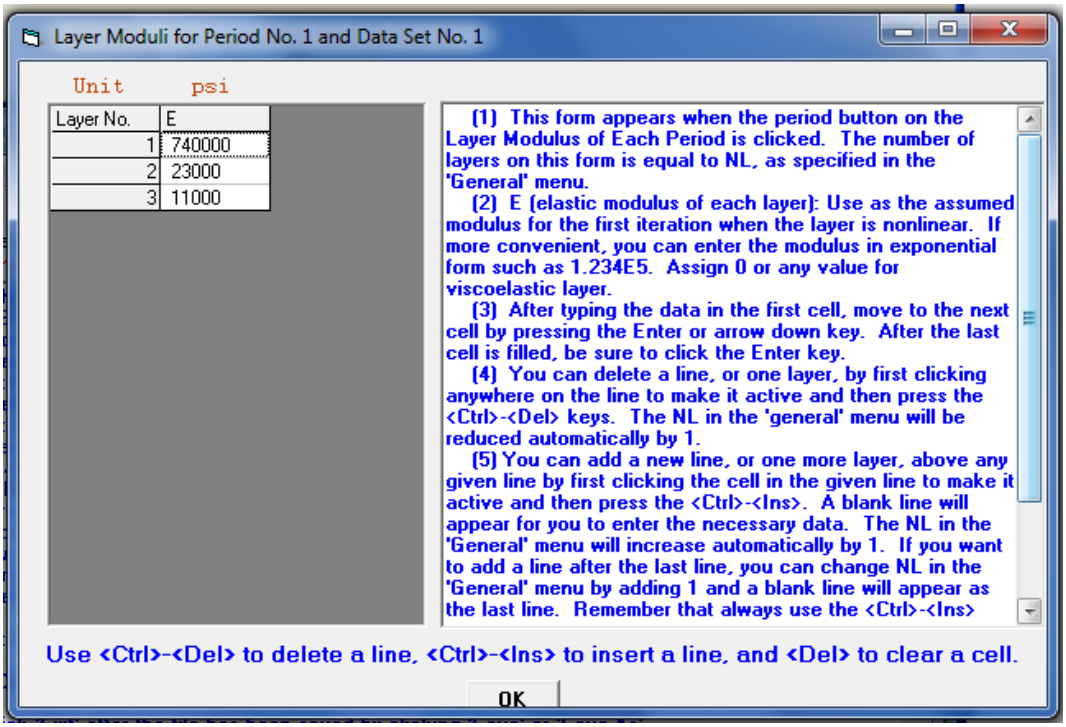


Figure (11): Layer Moduli for Period Screen

Step 9: Click on "Load" on the toolbar

When the 'Load' menu on the Main Menu of LAYERINP is clicked the screen as shown in Figure (12) will appears. The number of lines, or load groups, is equal to NLG, as specified in the 'General' menu. Details of 'Load' screen are below:

- **LOAD** (type of loading): Assign 0 for single axle with single tire, 1 for single axle with dual tires, 2 for tandem axles, and 3 for tridem axles.
- **CR** (contact radius of circular loaded ares).
- **CP** (contact pressure on circular loaded ares).
- **YW** (center to center spacing between two dual wheels along the y axis): Assign 0 if there is only one wheel or LOAD = 0.
- **XW** (center to center spacing between two axles along the x axis): Assign 0 if only one axle exists, i.e. LOAD = 0 or 1.
- **NR** (number of radial coordinates to be analyzed under a single wheel, maximum 25): A single wheel with LOAD = 0 is a case of axisymmetry so the location of response points is expressed in terms of radial coordinates. This column must be entered and cannot be left blank.
- **NPT** (number of points in x and y coordinates to be analyzed under multiple wheels, maximum 25): If LOAD > 0, the location of response points are expressed in terms of Cartesian coordinates x and y. This column must be entered and cannot be left blank.

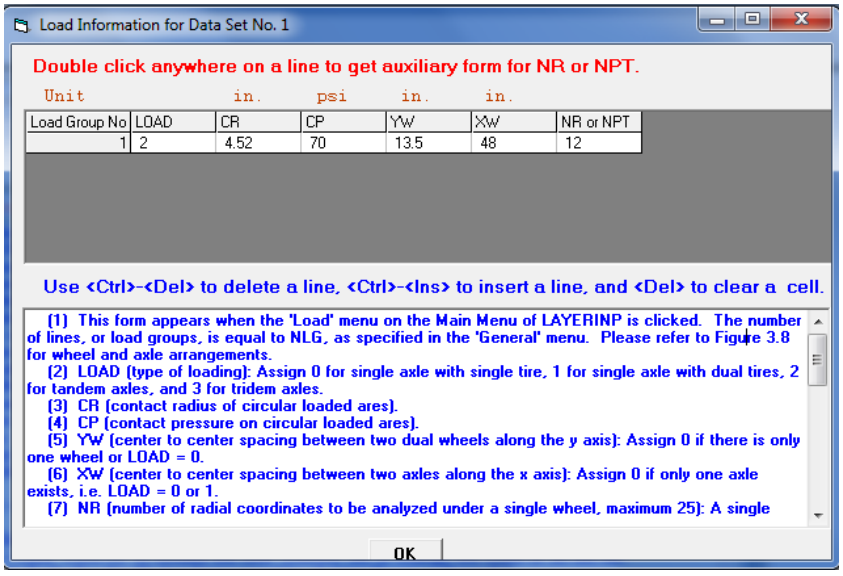


Figure (12): Load Screen

You can enter the auxiliary form of a given line without retyping the data for NR or NPT by double clicking that line. Instead of double clicking, you can also enter the auxiliary form by pressing the <Esc> key, but be sure to move the dotted rectangle to the line before pressing the <Esc> key. Because of the existence of an auxilary form, it is necessary to fill out the form line by line from top to bottom by pressing the Enter key. Do not use the arrow key to move to the next row because, without using the Enter key, these entries are not saved when the auxiliary form is entered. Arrow key can be used only when there is no auxiliary form. After completing this form and all the necessary auxiliary forms, click OK to return to the Main Menu of LAYERINP.

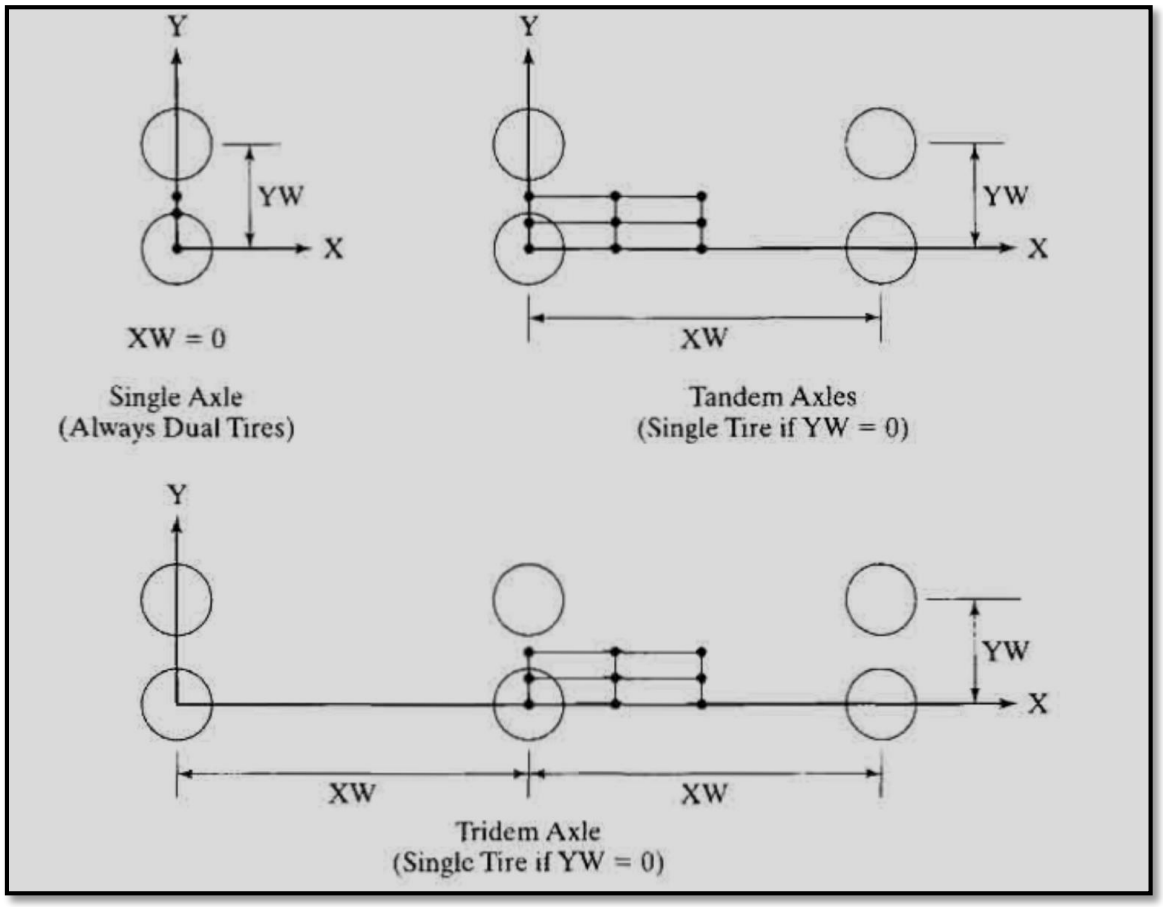


Figure (13): Wheel and Axle Arrangements.

Step 10: Fill in the auxiliary form

Figure (14) show auxiliary form for number of points in x and y coordinates (NPT) and Figure (15) for number of radial coordinates (NR).

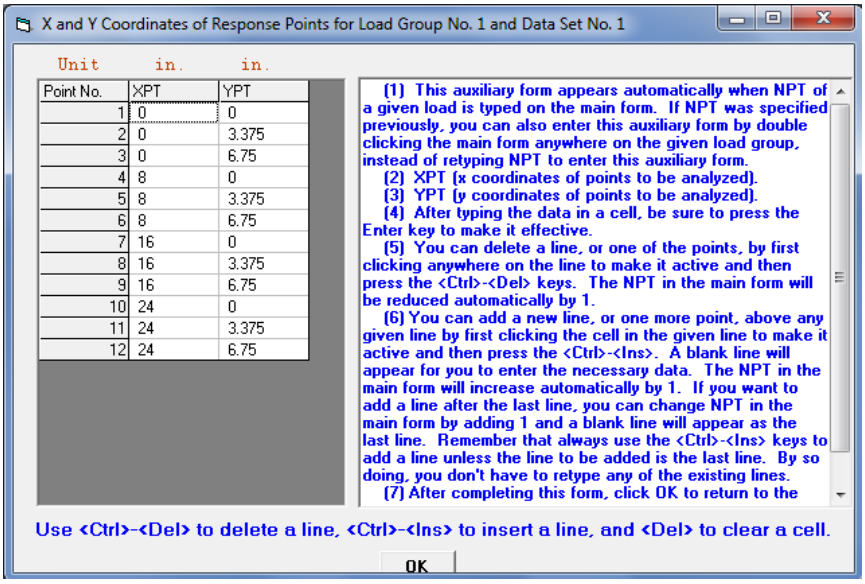


Figure (14): Auxiliary Form for Number of points in x and y coordinates (NPT)

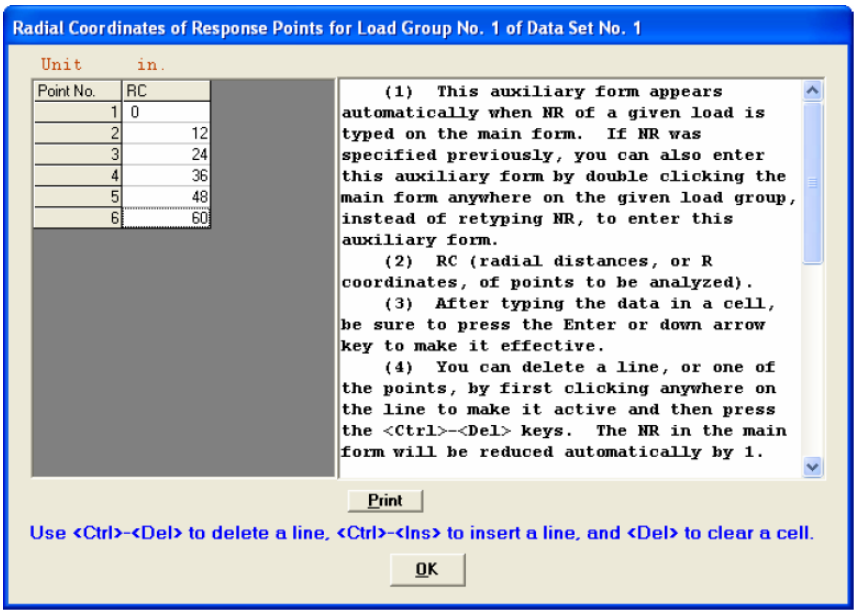


Figure (15): Auxiliary Form for Number of Radial coordinates (NR)

Step 11: Click on "Nonlinear" on the toolbar

When the 'Nonlinear' menu on Main Menu of LAYERINP is clicked the screen as shown in Figure (16) will appears. If there is a nonlinear clay subgrade, you have to decide whether you want to use the defaulted parameter values or not. Defaulted parameter values suggested in Figure (17) for Resilient-modulus–deviator-stress for four types of subgrade. When creating a new file, it is suggested that these default values be used, as indicated by the black dot in the option button. However, if you don't want to use these suggested values or you want to keep the values you input previously, you should click the 'Do not use the default input for clay' option button so when you enter the Seasonal Input Parameters screen, these existing values will not be replaced by the suggested values again.

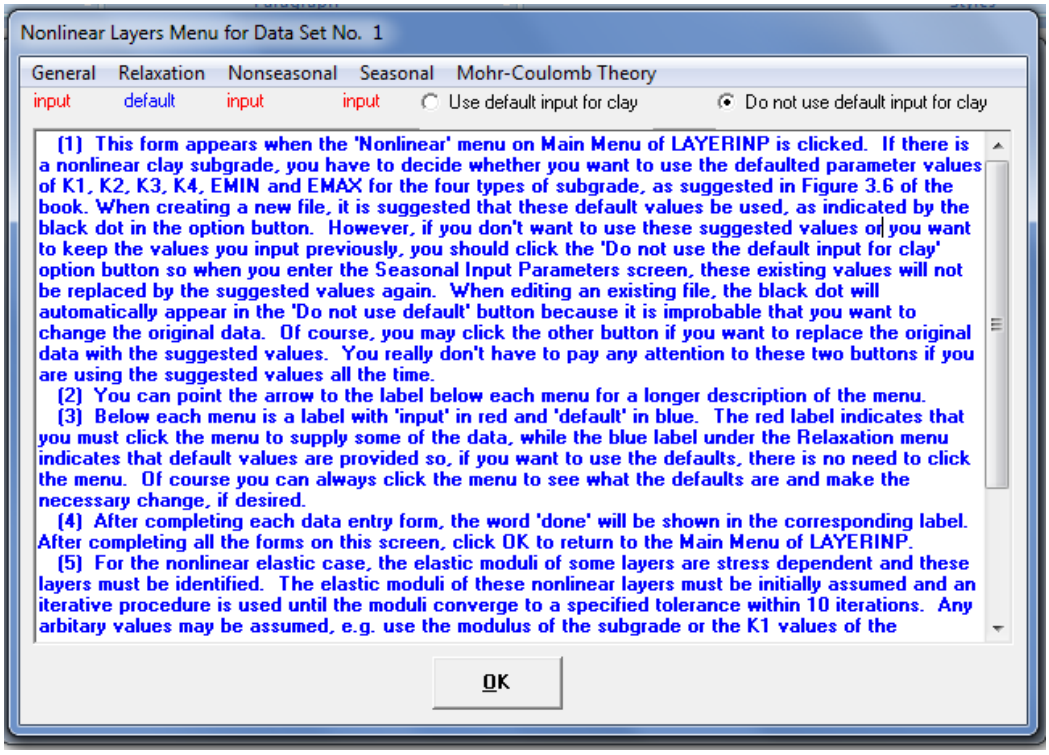


Figure (16): Nonlinear Screen

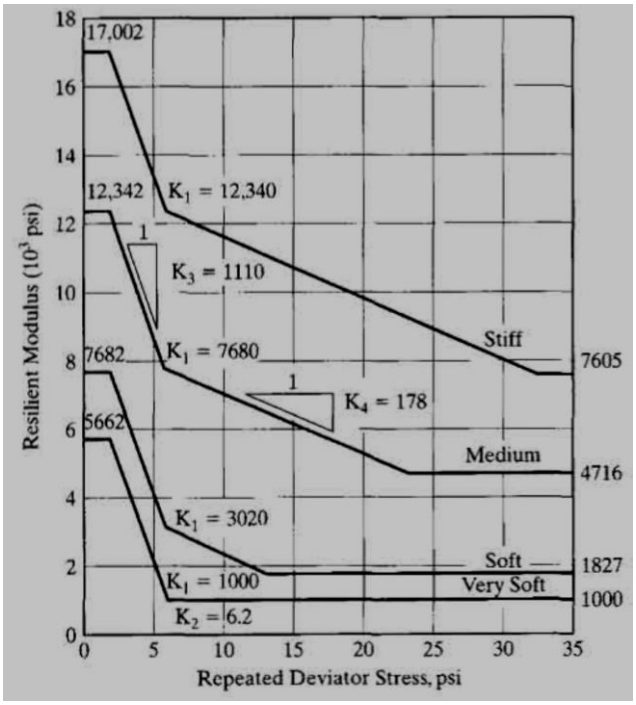


Figure (17): Resilient-Modulus–Deviator-Stress Relationship for Four Types of Subgrade.

You can point the arrow to the label below each menu for a longer description of the menu. Below each menu is a label with 'input' in red and 'default' in blue. You can always click the menu to see what the defaults are and make the necessary change, if desired. After completing each data entry form, the word 'done' will be shown in the corresponding label, Figure (18). After completing all the forms on this screen, click OK to return to the Main Menu of LAYERINP.

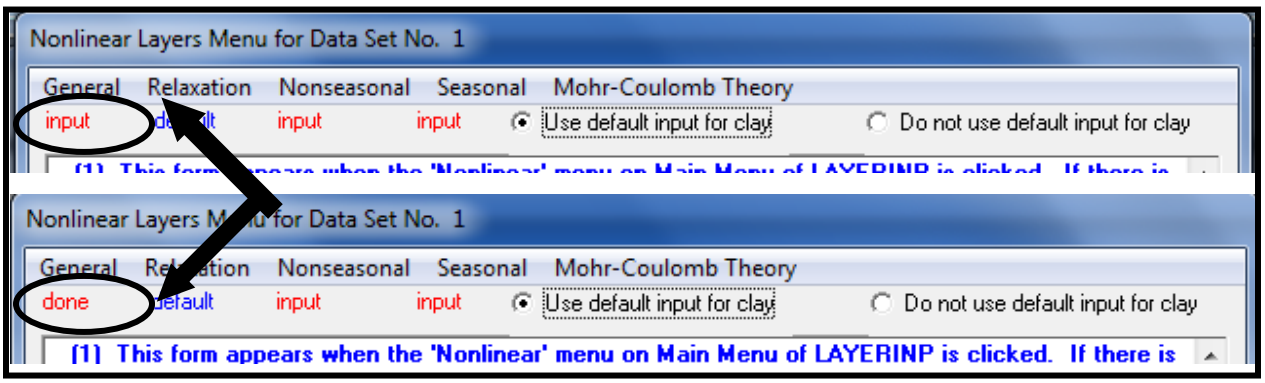


Figure (18): Changing in Label after Completing Data Entry

For longer description of the **Nonlinear Layers Menu** items (General, Relaxation, Nonlinear, Seasonal and Mohr-Coulomb Theory) click on each one you want according to your available data that want to entry and after completing click OK to return to the nonlinear layers menu.

Step 12: Click on "Viscoelastic" on the toolbar

When the ' Viscoelastic ' menu on Main Menu of LAYERINP is clicked the screen as shown in Figure (19) will appears. You can point the arrow to the label below each menu for a longer description of the menu. The red label indicates that you must click the menu to supply some of the data, while the blue label under the Time menu indicates that default values are provided so, if you want to use the defaults, there is no need to click the menu. Of course you can always click the menu to see what the defaults are and make the necessary change, if desired. After completing each data entry form, the word 'done' will be shown in the corresponding label. After completing all the forms on this screen, click OK to return to the Main Menu of LAYERINP.

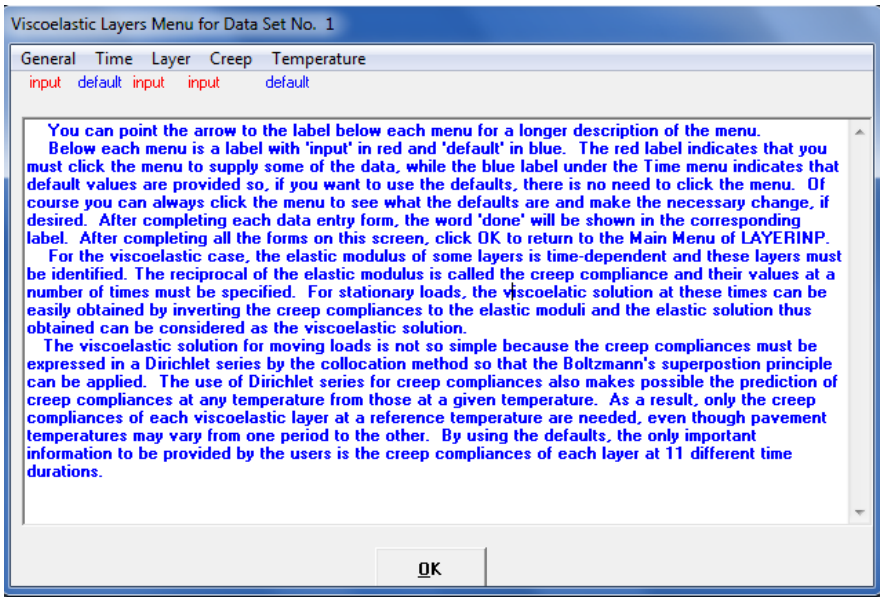


Figure (19): Viscoelastic Screen

For the viscoelastic case, the elastic modulus of some layers is time-dependent and these layers must be identified. The reciprocal of the elastic modulus is called the creep compliance and their values at a number of times must be specified. For stationary loads, the viscoelatic solution at these times can be easily obtained by inverting the creep compliances to the elastic moduli and the elastic solution thus obtained can be considered as the viscoelastic solution.

The viscoelastic solution for moving loads is not so simple because the creep compliances must be expressed in a Dirichlet series by the collocation method so that the Boltzmann's superpostion principle can be applied. The use of Dirichlet series for creep compliances also makes possible the prediction of creep compliances at any temperature from those at a given temperature. As a result, only the creep compliances of each viscoelastic layer at a reference temperature are needed, even though pavement temperatures may vary from one period to the other. By using the defaults, the only important information to be provided by the users is the creep compliances of each layer at 11 different time durations.

Step 13: Click on "Damage" on the toolbar

When the ' Damage ' menu on Main Menu of LAYERINP is clicked the screen as shown in Figure (20) will appears. You can point the arrow to the label below each menu for a longer description of the menu.

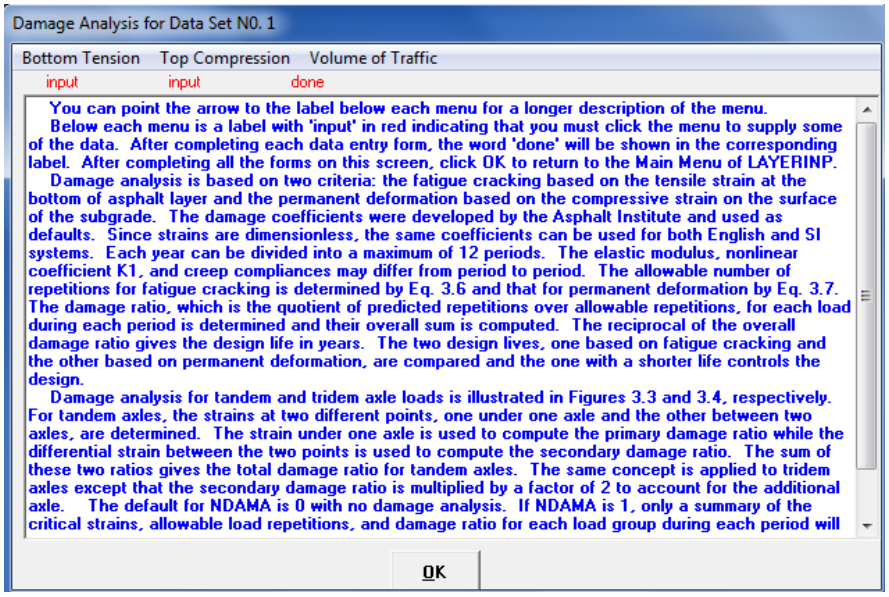


Figure (20): Damage Screen

Below each menu is a label with 'input' in red indicating that you must click the menu to supply some of the data. After completing each data entry form, the word 'done' will be shown in the corresponding label. After completing all the forms on this screen, click OK to return to the Main Menu of LAYERINP.

Damage analysis is based on two criteria: the fatigue cracking based on the tensile strain at the bottom of asphalt layer and the permanent deformation based on the compressive strain on the surface of the subgrade. The damage coefficients were developed by the Asphalt Institute and used as defaults. Since strains are dimensionless, the same coefficients can be used for both English and SI systems. Each year can be divided into a maximum of 12 periods. The elastic modulus, nonlinear coefficient K1, and creep compliances may differ from period to period. The allowable number of repetitions for fatigue cracking is determined by Eq. 1 and that for permanent deformation by Eq. 2.

$$N_f = f_1 \, (\epsilon_t)^{-f_2} \, (E_1)^{-f_3} \qquad \ldots\ldots\ldots\ldots\ldots \text{ Eq. 1}$$

Where:

N_f is the allowable number of load repetitions to prevent fatigue cracking ;

$\in_t$ is the tensile strain at the bottom of asphalt layer ;

E_1 is the elastic modulus of asphalt layer;

and f_1, f_2, and f_3 are constants determined from laboratory fatigue tests, with f_1 modified to correlate with field performance observations. The Asphalt Institute used 0 .0796, 3 .291, and 0 .854 for *f*l, *f2,* and f_3, respectively, in their analytically based design procedure; the corresponding values used by Shell are 0 .0685, 5 .671, and 2 .363 (Shook *et al.*, 1982).

$$N_d = f_4\ (\in_c)^{-f_5} \qquad \text{..... Eq. 2}$$

Where:

N_d is the allowable number of load repetitions to limit permanent deformation,

$\in_c$ is the compressive strain on the top of subgrade,

and f_4 and f_5 are constants determined from road tests or field performance . Values of f_4 and f_5 are suggested as 1.365 *x* 10^{-9} and 4.477 by the Asphalt Institute (AI, 1982), 6 .15 *x* 10^{-7} and 4.0 by Shell (Claussen *et al.*, 1977), and 1 .13 *x* 10^{-6} and 3.571 by the University of Notting -ham (Brown *et al.*, 1977).

The damage ratio, which is the quotient of predicted repetitions over allowable repetitions, for each load during each period is determined and their overall sum is computed. The reciprocal of the overall damage ratio gives the design life in years. The two design lives, one based on fatigue cracking and the other based on permanent deformation, are compared and the one with a shorter life controls the design.Damage analysis for tandem and tridem axle loads is illustrated in Figures (21) and (22), respectively.

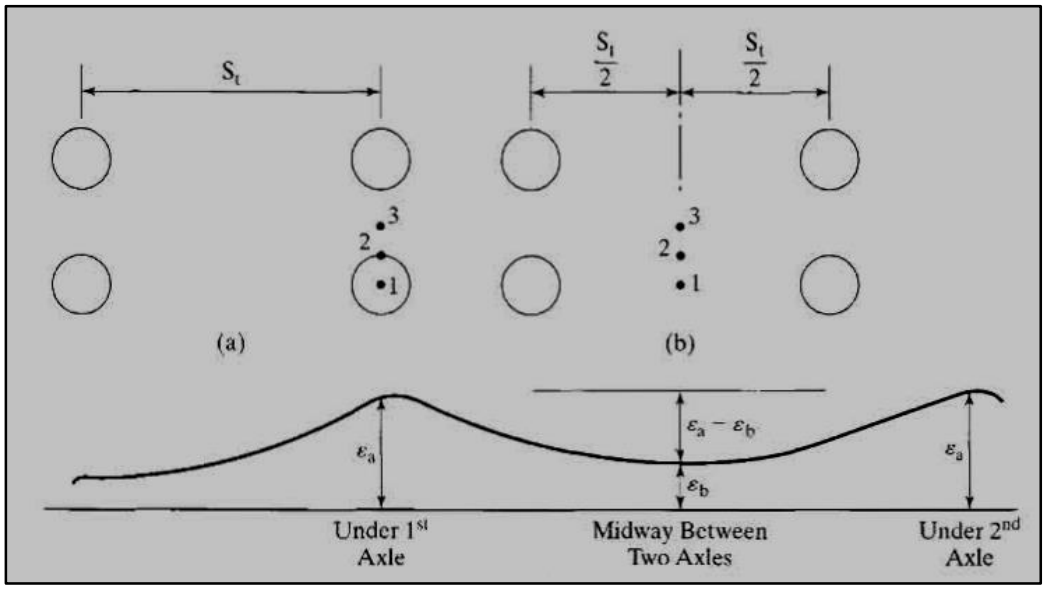


Figure (21): Damage Analysis of Tandem-Axle Loads

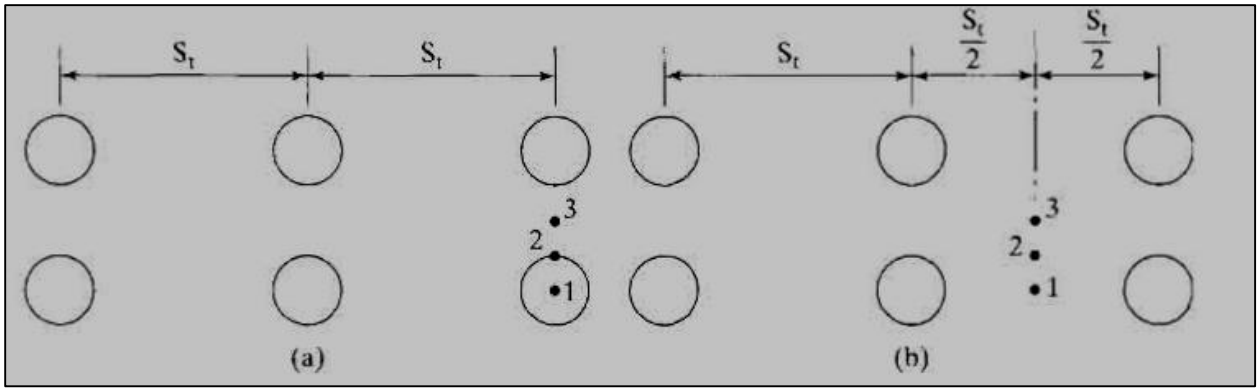


Figure (22): Damage Analysis of Tridem-Axle Loads

For tandem axles, the strains at two different points, one under one axle and the other between two axles, are determined. The strain under one axle is used to compute the primary damage ratio while the differential strain between the two points is used to compute the secondary damage ratio. The sum of these two ratios gives the total damage ratio for tandem axles. The same concept is applied to tridem axles except that the secondary damage ratio is multiplied by a factor of 2 to account for the additional axle. The default for NDAMA is 0 with no damage

analysis. If NDAMA is 1, only a summary of the critical strains, allowable load repetitions, and damage ratio for each load group during each period will be presented. If NDAMA = 2, a more detailed printout including the complete table of displacements, stresses and strains will be presented.

Step 11: Click on Save As to save the data to a new file

Name it like "Problem4" or something like that. Since this program is DOS-based, you probably can't use file names with embedded spaces or filenames with more than 8 characters (not counting the file extension, which will always be ".dat").

Step 12: Click on Exit

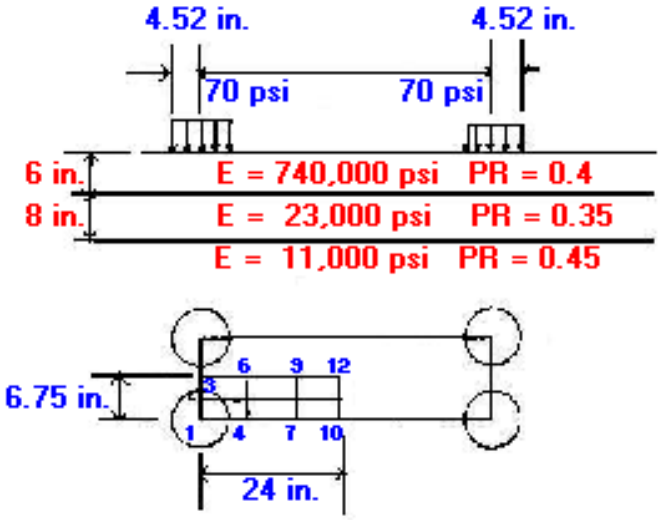
Step 13: Click on KENLAYER to perform the calculations

Step 14: Click on EDITOR to view the output file.

Application Example on KENLAYER Software

(LINEAR ELASTIC THREE-LAYER SYSTEM UNDER DUAL-TANDEM TIRES EXAMPLE)

LINEAR ELASTIC THREE-LAYER SYSTEM UNDER DUAL-TANDEM TIRES

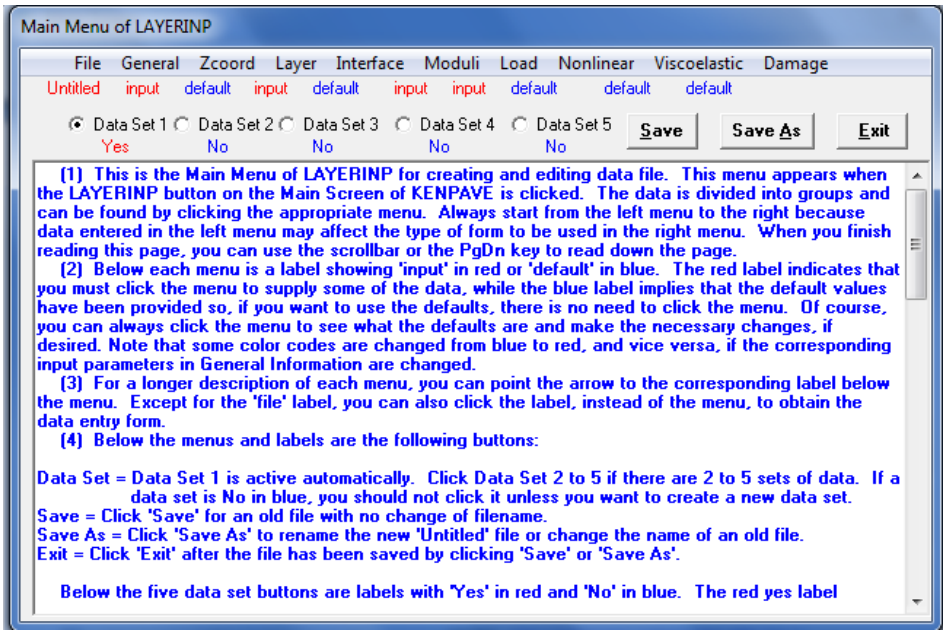


Example: The figure above shows an elastic three layer system under a set of dual tandem tires. Each tire has a contact radius of 4.52 in. (115 mm) and a contact pressure of 70 psi (483 kPa). Layer 1 has an elastic modulus of 740,000 psi (5.1 GPa), a Poisson's ratio of 0.4, and a thickness of 6 in. (152 mm); Layer 2 has an elastic modulus of 23,000 psi (159 MPa), a Poisson's ratio of 0.35, and a thickness of 8 in. (203 mm); and Layer 3 has an elastic modulus of 11,000 psi (77 MPa) and a Poisson's ratio of 0.45. Compute the ***surface*** and ***interface deflections*** at the 12 grid points shown in the figure and determine the ***maximum surface*** and ***interface deflections***.

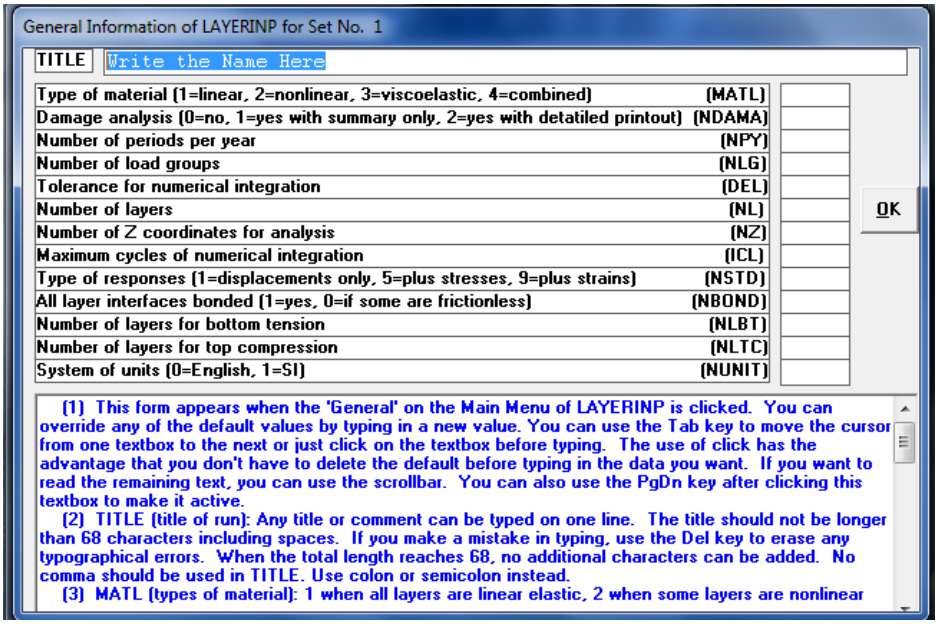
Note: all interfaces are fully bounded

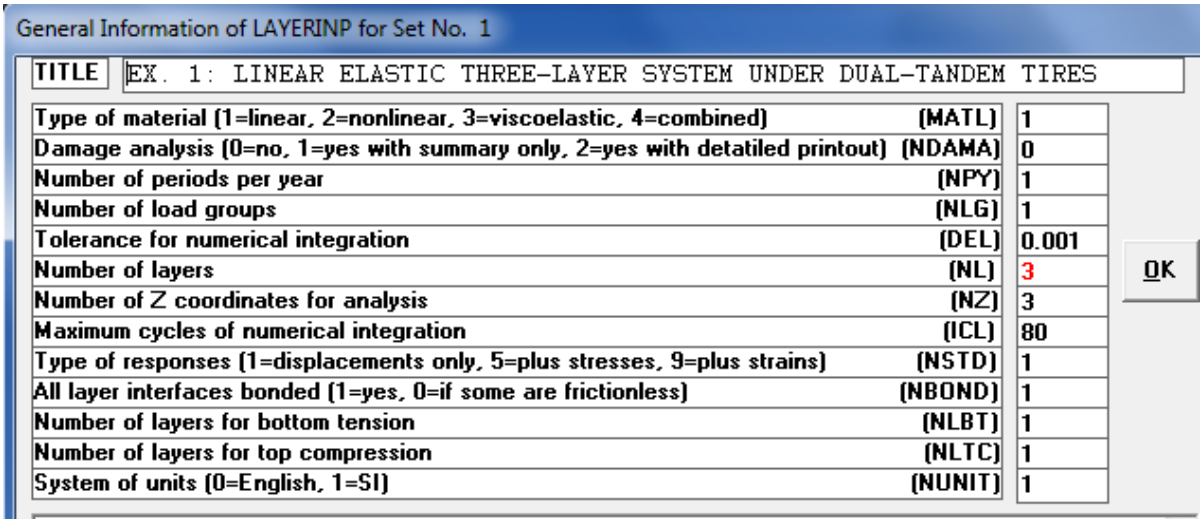
Solution:-

✓ **Step 1** : Select LAYERINP and create new file by click on File on the toolbar To set up a new data file click 'File' and 'New' and the filename 'Untitled' will appear on the label beneath 'File'. You can now proceed to input the necessary data.



✓ **Step 2** : Click on General on the toolbar and input title name and data.





TITLE: Enter any descriptive title like (EX. 1: LINEAR ELASTIC THREE-LAYER SYSTEM UNDER DUAL-TANDEM TIRES).

MATL: Enter "1" because all of the materials we'll be using are linear elastic.

NDAMA: Enter "0" since we don't want to do a damage analysis at this time.

NPY: Enter "1" since we will not be using seasonally-varying layer moduli.

NLG: Enter "1" because we will only have one load group.

DEL: Leave at the default of 0.001

NL: Enter "3" pavement layers.

NZ: Enter "3" because we're going to compute the surface and interface deflections outputs at these depths.

ICL: Leave at 80. If the program doesn't converge on a solution within 80 iteration, it stops.

NSTD: Enter "1" to output only vertical displacements.

NBOND: Leave at "1" to use fully frictional layer interfaces.

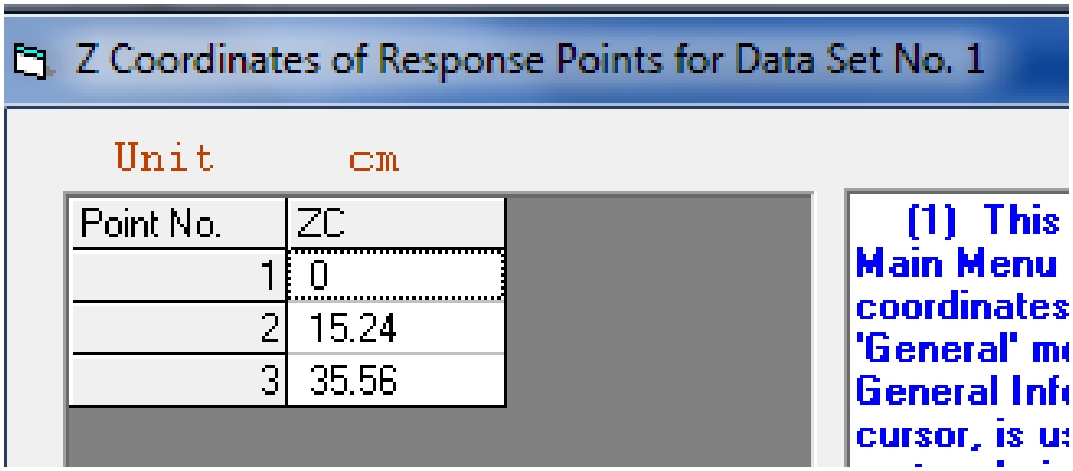
NLBT: Leave at "1" since we're not doing a damage analysis. *This entry specifies the number of layers for which a damage analysis will be performed based on the tensile strain at the bottom of the layer (which contributes to fatigue failure).*

NLBT: Leave at "1" since we're not doing a damage analysis. *This entry specifies the number of layers for which a damage analysis will be performed based on the compressive strain at the top of the layer (which contributes to rutting failure).*

NUNIT: Leave at "1" for SI units.

Upon completion, click 'OK' to return to the Main Menu of LAYERINP.

✓ **Step 3** : Click on Zcoord on the toolbar



ZC: Enter the depth at which you want to obtain deflection output. *When the point is located exactly at the interface between two layers, the results are for the bottom of upper layer. If the results at the top of next lower layer are desired, a slightly larger z coordinate, say 0.0001 larger, should be used.*

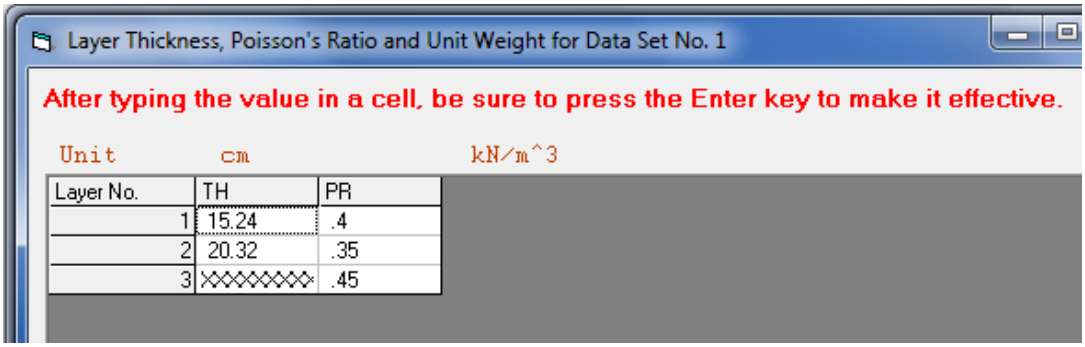
Upon completion, click the 'OK' button to return to the Main Menu of LAYERINP.

✓ **Step 4** : Click on Layer on the toolbar

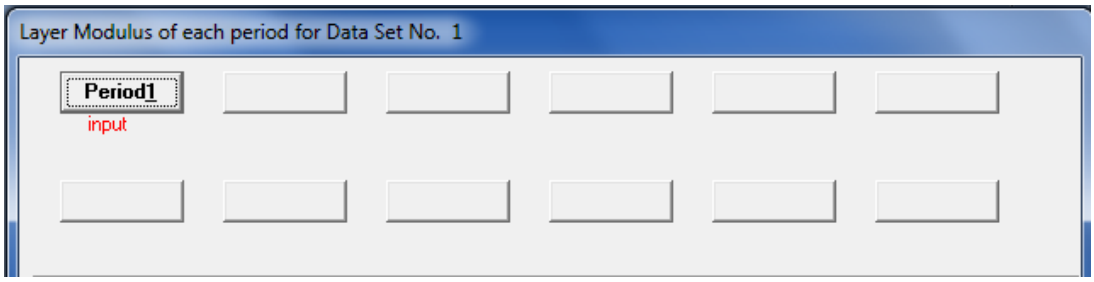
TH: Enter the thickness of each material layer. The last layer is infinite in thickness and need not be specified.

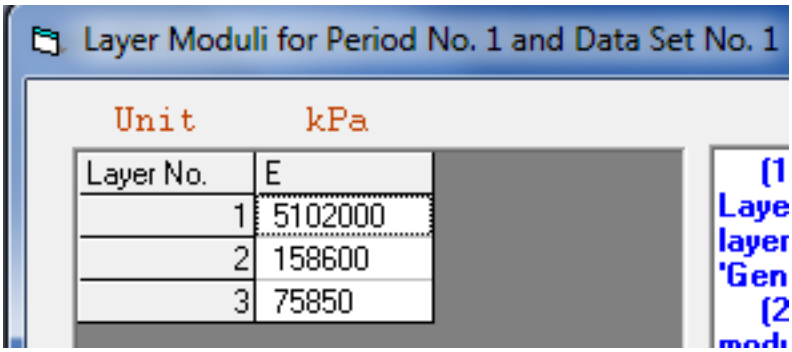
PR: Enter Poisson's ratio for all layers.

After completing this form, click 'OK' to return to the Main Menu of LAYERINP.



✓ **Step 5** : Click on "Moduli" on the toolbar & "Period1" button





✓ **Step 6** : Click on "Load" on the toolbar

LOAD: Enter "2" for tandem axles with dual wheels (four contact patches).

CR: Enter "11.4808" for the radius of the contact patch.

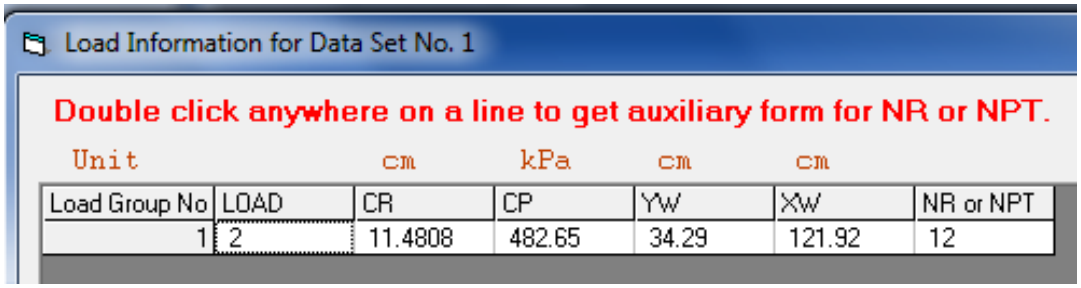
CP: Enter "482.65" for the contact pressure.

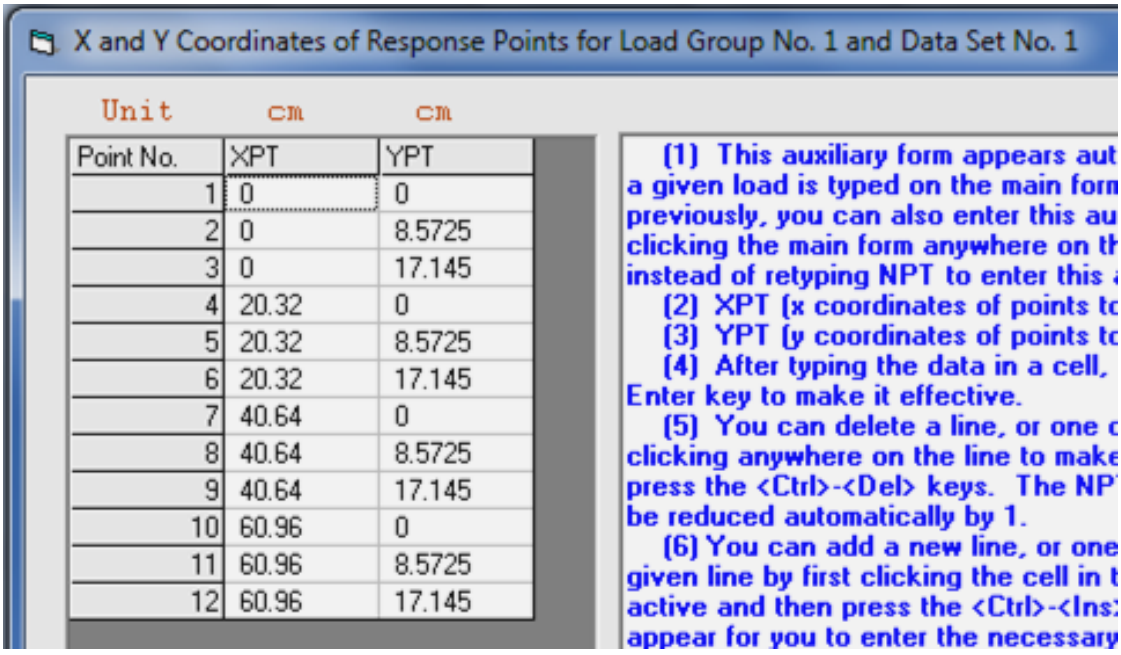
YW: Enter "34.29" center-to-center spacing between the two dual wheels.

XW: Enter "121.92" center-to-center spacing between the axles.

NR or NPT: Enter "12" for the number of points in the horizontal plane at which you want outputs.

After completing this form, click 'OK' to enter the auxiliary form where you enter the coordinates of the output points.





After completing this form, click 'OK' to return to the load information form, then click 'OK' to return to the Main Menu of LAYERINP.

✓ **Step 7** : Click on Save As to save the data to a new file

- ✓ **Step 8** : Click on Exit
- ✓ **Step 9:** Click on KENLAYER to perform the calculations
- ✓ **Step 10:** Click on EDITOR to view the output file

INPUT FILE NAME -F:\temp C\LAY1SI.DAT

NUMBER OF PROBLEMS TO BE SOLVED = 1

TITLE -EX. 1: LINEAR ELASTIC THREE-LAYER SYSTEM UNDER DUAL-TANDEM TIRES

MATL = 1 FOR LINEAR ELASTIC LAYERED SYSTEM
NDAMA = 0, SO DAMAGE ANALYSIS WILL NOT BE PERFORMED
NUMBER OF PERIODS PER YEAR (NPY) = 1
NUMBER OF LOAD GROUPS (NLG) = 1
TOLERANCE FOR INTEGRATION (DEL) -- = 0.001
NUMBER OF LAYERS (NL)---------- = 3
NUMBER OF Z COORDINATES (NZ)------ = 3
LIMIT OF INTEGRATION CYCLES (ICL)- = 80
COMPUTING CODE (NSTD)---------- = 1
SYSTEM OF UNITS (NUNIT)----------= 1

Length and displacement in cm, stress and modulus in kPa unit weight in kN/m^3, and temperature in C

THICKNESSES OF LAYERS (TH) ARE : 15.24 20.32
POISSON'S RATIOS OF LAYERS (PR) ARE : 0.4 0.35 0.45
VERTICAL COORDINATES OF POINTS (ZC) ARE: 0 15.24 35.56
ALL INTERFACES ARE FULLY BONDED

FOR PERIOD NO. 1 LAYER NO. AND MODULUS ARE : 1 5.102E+06 2 1.586E+05
3 7.585E+04

LOAD GROUP NO. 1 HAS 4 CONTACT AREAS
CONTACT RADIUS (CR)---------- = 11.4808
CONTACT PRESSURE (CP)---------- = 482.65
NO. OF POINTS AT WHICH RESULTS ARE DESIRED (NPT)-- = 12
WHEEL SPACING ALONG X-AXIS (XW)---------- = 121.92
WHEEL SPACING ALONG Y-AXIS (YW)---------- = 34.29

RESPONSE PT. NO. AND (XPT, YPT) ARE: 1 0.000 0.000 2 0.000 8.573
3 0.000 17.145 4 20.320 0.000 5 20.320 8.573 6 20.320 17.145
7 40.640 0.000 8 40.640 8.573 9 40.640 17.145 10 60.960 0.000
11 60.960 8.573 12 60.960 17.145
PERIOD NO. 1 LOAD GROUP NO. 1

POINT NO.	VERTICAL COORDINATE	VERTICAL DISPLACEMENT
1	0.00000	0.05362
1	15.24000	0.05021
1	35.56000	0.04455
2	0.00000	0.05429
2	15.24000	0.05133
2	35.56000	0.04549
3	0.00000	0.05466
3	15.24000	0.05157
3	35.56000	0.04581
4	0.00000	0.05218
4	15.24000	0.04903
4	35.56000	0.04504
5	0.00000	0.05307
5	15.24000	0.05015
5	35.56000	0.04593
6	0.00000	0.05340
6	15.24000	0.05050
6	35.56000	0.04623
7	0.00000	0.04962
7	15.24000	0.04637
7	35.56000	0.04394
8	0.00000	0.05043
8	15.24000	0.04727
8	35.56000	0.04469
9	0.00000	0.05070
9	15.24000	0.04758
9	35.56000	0.04495
10	0.00000	0.04859
10	15.24000	0.04529
10	35.56000	0.04334
11	0.00000	0.04930
11	15.24000	0.04609
11	35.56000	0.04404
12	0.00000	0.04954
12	15.24000	0.04637
12	35.56000	0.04428

The results obtained by KENLAYER shows that the maximum surface deflection is 0.02044 in. (0.52 mm) and occurs at point 3, the maximum deflection at the first interface is 0.02030 in. (0.52 mm) and also occurs at point 3, while the maximum deflection at the second interface is 0.01820 in. (0.46 mm) and occurs at point 6. The movement of maximum deflections from point 3 to point 6 as the depth increases is reasonable because the effect of tandem loads is more pronounced at greater depths.