International Journal of Advance Research in Engineering, Science & Technology(IJAREST),
ISSN(O):2393-9877, ISSN(P): 2394-2444,
Volume 2, Issue 6, June- 2015, Impact Factor: 2.125

Effect of Bond Strength on Performance of Pavement

Suhas.S.Biradar, Avinash.Angadi, Pradeep.Yaragatti,Hanmantarao.G.Joteppanavar
[1]Civil Engineering, KLE Dr.M.S.Sheshgiri college of Engineering & Technology,Belgaum,suhas.biradar@gmail.com
[2]Civil Engineering, KLE Dr.M.S.Sheshgiri college of Engineering & Technology,Belgaum,Avinahangadi93@gmail.com
[3]Civil Engineering, KLE Dr.M.S.Sheshgiri college of Engineering & Technology,Belgaum,pbypradeep@gmail.com
[4]Civil Engineering, KLE Dr.M.S.Sheshgiri college of Engineering & Technology,Belgaum,hanamantaraojoteppanavar@gmail.com

Abstract
A mechanistic model of flexible pavement was considered and constructed to study the effects of shear modulus on performance of various pavement layers. The objective of this study was to quantify the effects of interlayer bonding by varying of the shear modulus of the contact layer. Flexible pavements are complex structures consisting of several layers of asphalt and granular materials . Bonding is assessed due to increase or decrease of critical stress values when static loading condition is maintained with constant elastic properties such as Modulus of elasticity, shear modulus etc. The flexible pavement works as a single structure due to good bonding between the different layers. Lack of bonding destroy continuity, decrease structural strength, and allow water to enter sub layers. Bituminous tack coat is laid to ensure contact of the aggregates of the various layers of the flexible pavement. The viscous nature of the flexible pavement, allows its different layers to sustain significant plastic deformation, although distresses due to repeated heavy loading over time which is the most common failure mechanism. It is generally believed that, the pavement stress distribution is extremely influenced by the adhesion conditions at the layer interface .This characteristic is used to access pavement performance.

Keywords*—Adhesion, Bonding, Critical stress, Elastic Modulus, Shear Modulus, Flexible pavement ,Finite Element analysis, Static wheel load, Multiple regression analysis*

I. INTRODUCTION

The viscous nature of the flexible pavement, allows its different layers to sustain significant plastic deformation, although distresses due to repeated heavy loading over time which is the most common failure mechanism.

Poor adhesion at layer interface may cause adverse effects on the structural strength of the pavement system and form numbers of premature failures. To increase bonding between layers, bituminous tack coats are applied prior to overlay. Bituminous emulsions are normally used as tack coats. In spite of their extensive application, the thoughts among pavement engineers differ regarding the effectiveness of tack coat in enhancing the adhesion between the two layers. This tack coat also made of a thin layer of bitumen residue and its objective is to provide adequate adherence between the layers. If the quantity of bituminous emulsions used is in excess or less than the required one, the interface bonding will not be satisfactory. One assumption in the structural response model for flexible pavements is that the asphalt layers and completely bonded to each other, when in all reality they may not be[1]. There is no widely accepted test method for measuring the degree of bonding between pavement layers. Without proper modeling of the bond between layers, the calculation of pavement response, and thus the design of the flexible pavement structures, cannot be accurate. Poor bonding between pavement layers can cause many distresses[2]. One of the most common distress is slippage failure, which usually occurs where heavy vehicles are often accelerating, decelerating, or turning. Being a layered structure, the life of an asphalt pavement not only depends on the strength and stiffness of its individual layers, but also on the bond between them. The bonding characteristics are mainly influenced by the bond strength analysis of the flexible pavement layers which is mainly influenced on elastic properties such as modulus of elasticity, shear modulus, poison's ratio etc. The present work involves accessing of the effect of the bond strength on the performance of pavement as a elastic body by the stresses induced in the individual layers due to the effect of loading and elastic properties[3]. The increasing values of the critical stresses indicate poor bonding while the decreasing value of stresses indicate better bonding causing elastic deformation at the surface of the pavement as the effect of load transfer from layer to layer.

Present work involves determination of the relation between the elastic properties on the bond strength of the pavement by performing multilayer regression analysis. The bonding is accessed by introduction of a contact layer in the five layered flexible pavement whose shear modulus has the influence on the bonding characteristics. The analysis is done with a finite element approach of the flexible pavement by developing a mechanistic model in Ansys APDL with assigning of material properties i.e. modulus of elasticity ,shear/rigidity modulus and poison's ratio. Various models analyzed varying modulus of elasticity for sub grade sub base,base,binder course and wearing course with poison's ratio as well as thickness of the layers being unaltered and static loading condition. This gives the maximum and minimum stresses induced in the individual layer as the effect of modulus of elasticity. The minimum stresses are analyzed in order to determine the bonding condition of the performance of pavement. Similarly shear modulus is varied for contact layer and its effect on performance of pavement is analyzed. Thus an economic method of evaluating is done by Finite element analysis.

Where in the materials of pavement are assumed to be made up of finite number of elements. The basis of the finite element method is the representation of a body or a structure by an assemblage of sub-divisions called Finite Elements. These finite elements are obtained by means of fictitious cuts through the original structure. Adjoining elements may be thought of as being connected at common points, termed as nodes or nodal points.

II. LITERATURE REVIEW

2.1 Numerical Methods :

The formulation for structural analysis is generally based on the three fundamental relations: equilibrium, constitutive and compatibility. There are two major approaches to the analysis: Analytical and Numerical. Analytical approach which leads to closed-form solutions is effective in case of simple geometry, boundary conditions, loadings and material properties. However, in reality, such simple cases may not arise. As a result, various numerical methods are evolved for solving such problems which are complex in nature. For numerical approach, the solutions will be approximate when any of these relations are only approximately satisfied. The numerical method depends heavily on the processing power of computers and is more applicable to structures of arbitrary size and complexity. It is common practice to use approximate solutions of differential equations as the basis for structural analysis. This is usually done using numerical approximation techniques. Few numerical methods which are commonly used to solve solid and fluid mechanics problems are given below:

1. Finite Element Method
2. Finite Volume Method
3. Finite Difference Method

2.1.1 Finite Element Method

Thousands of engineers use finite-element codes, such as ANSYS, for thermo mechanical and nonlinear applications. Most academic departments offering advanced degrees in mechanical engineering, civil engineering, and aerospace engineering. The transformation of a physical phenomenon into differential equation involves differential calculus and mathematical concepts. So, once we have this physical phenomena expressed in terms of a differential equations and set of boundary conditions. Then they can adopt the finite element method to solve the problem for the solution at any point over the domain that we are interested. Finite element method is a numerical approach by which a general differential equation can be solved in an approximate manner. The solution that we are going to get from finite element method is an approximate and in some cases, this approximate solution may match with exact solution. Any physical phenomena can be any one of those listed above, that is, it can be any problem in structural mechanics, beams, plates, torsion of shafts and heat conduction problems, diffusion and groundwater flow problems. All these problems or any of these problems needs to be converted into a mathematical form. In the form of differential equation that needs to be solved over a domain subjected to some boundary conditions.

The basis of the finite element method is the representation of a body or a structure by an assemblage of sub-divisions called Finite Elements. These finite elements are obtained by means of fictitious cuts through the original structure. Adjoining elements may be thought of as being connected at common points, termed as nodes or nodal points. Then simple functions are chosen to approximate the variation of the actual displacements over each finite element. Such assumed functions are called Displacement Functions of Displacement Models. The unknown magnitudes of the displacement functions are the displacements at the nodal points. Hence, the final solution will yield the approximate displacements at the nodal points. The displacement model can be expressed in various simple forms, such as polynomials and trigonometric functions. Instead of displacement model, other type of models like the Stress model or Hybrid model may also be used.

2.1.2 Concepts of Elements and Nodes :

Any continuum/domain can be divided into a number of pieces with very small dimensions. These small pieces of finite dimension are called 'Finite Elements'. A field quantity in each element is allowed to have a simple spatial variation which can be described by polynomial terms. Thus the original domain is considered as an assemblage of number of such small elements. These elements are connected through number of joints which are called 'Nodes'. While discretizing the structural system, it is assumed that the elements are attached to the adjacent elements only at the nodal points. Each element contains the material and geometrical properties. The material properties inside an element are assumed to be constant. The elements may be 1D elements, 2D elements or 3D elements. The physical object can be modelled by choosing appropriate element such as frame element, plate element, shell element, solid element, etc. All elements are then assembled to obtain the solution of the entire domain/structure under certain loading conditions. Nodes are assigned at a certain density throughout the continuum depending on the anticipated stress levels of a particular domain. Regions which will receive large amounts of stress variation usually have a higher node density than those which experience little or no stress.

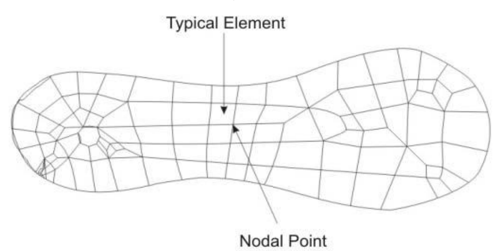

Figure 1:Idealization of elements and Nodes

2.1.3 Degrees of Freedom

A structure can have infinite number of displacements. Approximation with a reasonable level of accuracy can be achieved by assuming a limited number of displacements. This finite number of displacements is the number of degrees of freedom of the structure. For example, the truss member will undergo only axial deformation. Therefore, the degrees of freedom of a truss member with respect to its own coordinate system will be one at each node. If a two dimension structure is modeled by truss elements, then the deformation with respect to structural coordinate system will be two and therefore degrees of freedom will also become two. The degrees of freedom for various types of element are shown in Fig.

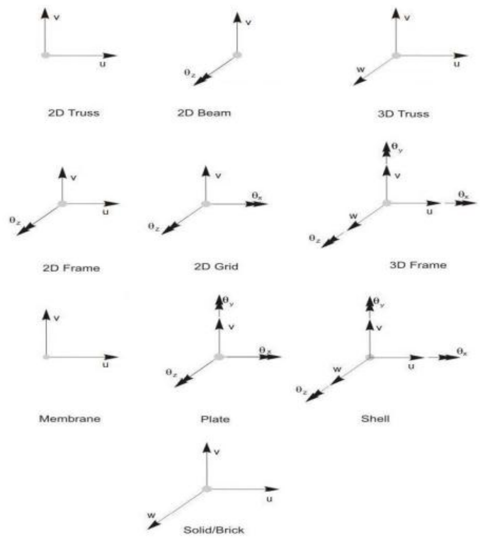

Figure 2:Degrees of freedom for various structural members

Advantages of FEA :

1. The physical properties, which are intractable and complex for any closed bound solution, can be analyzed by this method.
2. It can take care of any geometry (may be regular or irregular).
3. It can take care of any boundary conditions.
4. Material anisotropy and non-homogeneity can be catered without much difficulty.
5. It can take care of any type of loading conditions.
6. This method is superior to other approximate methods like Galerkin and Rayleigh-Ritz methods.
7. In this method approximations are confined to small sub domains.
8. In this method, the admissible functions are valid over the simple domain and have nothing to do with boundary, however simple or complex it may be.
9. Enable to computer programming.

Disadvantages of FEA :

1. Computational time involved in the solution of the problem is high.
2. For fluid dynamics problems some other methods of analysis may prove efficient than the FEM.

Loading Conditions

There are multiple loading conditions which may be applied to a system. The load may be internal and/or external in nature. Internal stresses/forces and strains/deformations are developed due to the action of loads. Most loads are basically "Volume Loads" generated due to mass contained in a volume. Loads may arise from fluid-structure interaction effects such as hydrodynamic pressure of reservoir on dam, waves on offshore structures, wind load on buildings, pressure distribution on aircraft etc. Again, loads may be static, dynamic or quasi-static in nature. All types of static loads can be represented as:

• Point loads, Line loads, Area loads and Volume loads

The loads which are not acting on the nodal points need to be transferred to the nodes properly using finite element techniques.

III. ANSYS APDL

ANSYS is general-purpose finite-element modeling software for numerically solving a wide variety of mechanical problems. These problems include static/ dynamic, structural analysis (both linear and nonlinear), heat transfer, and fluid problems, as well as acoustic and electromagnetic problems.

In general, a finite-element solution may be broken into the following three stages.

(1) Preprocessing: defining the problem the major steps in preprocessing are:

(i) Define key points/lines/areas/volumes.

(ii) Define element type and material/geometric properties.

(iii) Mesh lines/areas/ volumes as required. The amount of detail required will depend on the dimensionality of the analysis, i.e., 1D, 2D, axisymmetric, and 3D.

(2) Solution: assigning loads, constraints, and solving, this step involves assigning of loads to the model. Here it is necessary to specify the loads (point or pressure), (constraints translational and rotational), and finally solve the resulting set of equations.

(3) Post processing: further processing and viewing of the results in this step following are obtained

- (i) Lists of nodal displacements.
- (ii) Element solution
- (iii) Deflection plots.
- (iv) Nodal solutions.

IV. MODELING

FINITE ELEMENT ANALYSIS

Finite element method was used to analyze the pavement section resting on subgrade soils. The software ANSYS was used for finite element modeling. The pavement section was modeled as a 3-D axisymmetric problem and 20-noded structural solid element was used for the analysis. The stresses and deformations within the pavement section and vertical strain at top of the subgrade were captured. A six-layer flexible pavement system was modeled and analyzed. Figure shows the typical model for six layered flexible pavement resting on subgrade soils A Similar models were developed for pavement resting on subgrade soils. The thickness of each layer in the pavement was modeled as per Indian practice code IRC:37-2001. For the c/s area is assumed as 18m x 3.5m &

thicknesses are of each layer in the pavement section resting on subgrade soils are presented as below.

- Sub grade = 2m
- Sub base = 150-250mm
- Base = 100- 250mm
- Binder course = 90mm
- Contact layer= 10mm
- Wearing course = 50mm

A point load was applied on a node in downward direction. For the application of FEM Analysis, the layered system of infinite extent is reduced to an approximate size with finite dimension. The elasto-plastic analysis was carried out to evaluate the primary response of the pavement resting on subgrade soils. The multilinear isotropic hardening model (MISO) available in ANSYS was used to evaluate the stresses, strains and deformations within the pavement sections. The mixed incremental method is used in present study for elasto-plastic analysis of 3-D axisymetric finite element model.
This method combines the advantages of both the incremental and the iterative schemes. The external load, here, is applied incrementally, but after each increment, successive iterations are performed to achieve equilibrium.

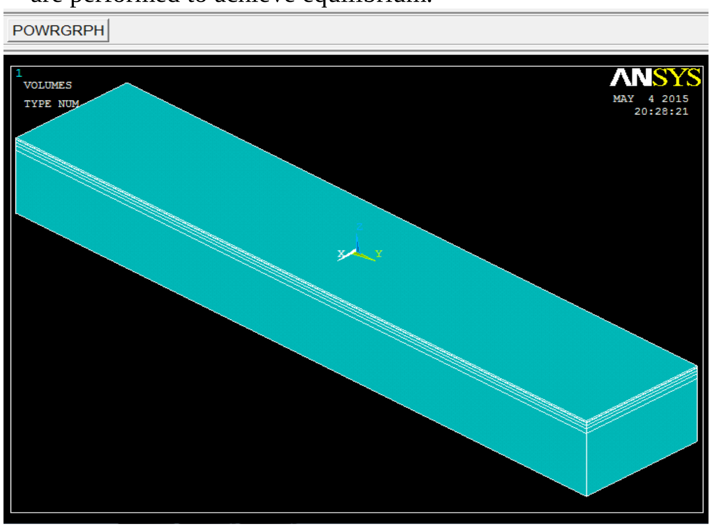

Figure 3: 3D view of the Flexible pavement Model

V. WORKING WITH ANSYS

Structural Analysis

ANSYS Structural software addresses the unique requirements of pure structural analysis without the need for extraneous tools. The software delivers all the power of non linear and linear capabilities. To deliver the high quality , reliable structural results.

Thermal Analysis

ANSYS thermal software addresses the unique requirements of pure thermal analysis without the need for extraneous tools. To keep temperature from affect a components life, ANSYS software can predict temperature ranges in which a component will work and when it will be break down ,as well as how hot cold components will get in various altitudes , climates and weather conditions.

Ansys Fluid

ANSYS computational fluid dynamics (CFD) simulation allows to predict , with confidence the impact fluid flows on surfaces-throughout design manufacturing as well as during end use.

Flotran CFD

The ANSYS FLOTRAN elements, fluids, solve for two- and three-dimensional flow, pressure and temperature distributions in a single phase viscous fluid. For these elements, the ANSYS program calculates velocity components, pressure, and temperature from the conservation of three properties: mass, momentum, and energy.

Electro Magnetic

Software can do the following types of static, harmonic, and transient magnetic analysis. 3-D static magnetic analysis, 3-D harmonic magnetic analysis, 3-D transient magnetic analysis.
As well as it gives the nodal, edge high frequency electrical results and their effect. Initially the APDL ANSYS software run from the start menu, global user interface is popped up. Preferences set to structural as shown in fig.

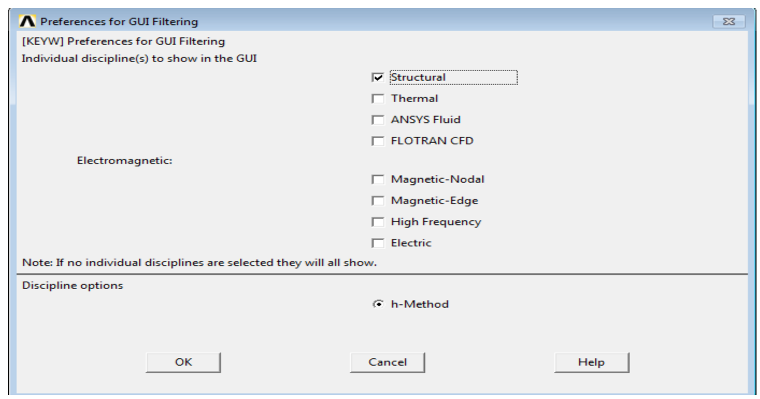

Figure 4:Type of analysis is specified in preferences

VI. DEFINING ELEMENT TYPES AND REAL CONSTANTS

Different Element Types In Ansys Library

Various types of elements are available for the purpose of modeling in the ansys parametric digital library. Type of element is dependent mainly on the behavior of the material due to its properties. Suitability of type of element is dependent on type of analysis being carried out Following are the various types :

Structural Mass

A Point mass can be added to an ANSYS FEA model to include the load due to mass that does not stiffen a structure, but that is important to capturing inertial loads such as those due to gravity, rotation about an axis, or dynamics.

Link

Depending upon the application, the element may be thought of as a **truss** element, a cable element, a link element, a spring element, etc. The three-dimensional spar element is a uniaxial tension-**compression** element with three degrees of freedom at each node: translations in the nodal x, y, and z directions.

Beam

The BEAM element is suitable for analyzing slender to moderately stubby/thick beam structures. This element is based on Timoshenko beam theory. Shear deformation effects are included.

Pipe

PIPE is a uniaxial element with tension-compression, torsion, and bending capabilities. The element has six degrees of freedom at two nodes: translations in the nodal x, y, and z directions and rotations about the nodal x, y, and z axes

Solid

A solid element that exhibits quadratic displacement behavior. The element is defined by 20 nodes having three degrees of freedom per node: translations in the nodal x, y, and z directions. The element supports plasticity, creep, hyperelasticity, stress stiffening, large deflection and large strain capabilities.

Shell

Thin walled structures present unique challenges for numerical simulation. It is conceptually simple to represent a thin walled structure using traditional 3-D shell element. Shell element has smaller thickness between the walls.

Solid Shell

Solid shell elements are indispensable for the study of the mechanics of complex structures. Two classes of shell elements are commonly used in finite element analyses of thin structures, classical two-dimensional elements and three-dimensional continuum elements.

Constraint

Constraints include not only roller, pin, and fixed supports in mechanical systems, but also things like voltage, frequency and temperature for various other types of analyses.

Contact

Contact may be used to represent contact and sliding between two surfaces (or between a node and a surface) in 3-D. The element has two degrees of freedom at each node: translations in the nodal x and y directions. Contact occurs when the contact node penetrates the target line. Preferably used for problems involving contact pressure.

Gasket

The modeling of diesel engine cylinder head gasket joints is complicated by the non linear response of the head gasket's materials. Linearization of these materials responses can lead to significant errors in the solution's results.

Cohesive

In cohesive zone modeling of interfacial fracture, despite the fact that it has been well established theoretically and has been demonstrated many times in various forms. Common cohesive elements are used in software for analysis.

Combinations

Model with combination of shell and beam elements - enough to answer about its deformation, but if you need predominantly 3D stress especially at the connection between plates and beam then you are modeling with solid elements at least at this part of the connection.

Ansys Fluid-

ANSYS Finite Element Model to include a contained fluid inside a shell or solid model of a container, in order to capture the effect of fluid pressure and fluid mass on linear, modal dynamic, plus nonlinear static and transient dynamic models.

Pore Pressure

Software gives more comprehensive approach was proposed, coupling hydrodynamic and geotechnical models, in order to assess the stability of the ensemble structure-foundation soil. Computational fluid dynamics (CFD) or physical models can be used to provide loads on structures due to wave action. Used for analysis of pore water pressure in case of the earthen dams.

User Matrix

Top-down sub structuring is also possible in ANSYS (the entire model is built, then super-element are created by selecting the appropriate elements). This method is suitable for smaller models and has the advantage that the results for multiple super-elements can be assembled in post processing.

Superelement

Sub structuring is a procedure that condenses a group of finite elements into one element represented as a matrix. This single matrix element is called a super element.

Surface Effect

Surface effect may be used for various load and surface effect applications. It may be overlaid onto an area face of any 3-D element. The element is applicable to three-dimensional structural analyses. Various loads and surface effects may exist simultaneously.

Not Solved

Heat of fusion or changes in thermal properties over temperature ranges, rather it is concerned with the element death procedure. More accurate models using element death can then be created as required models for thermal analysis.

The ANSYS element library contains more than 100 different element types. Each element type has a unique number and a prefix that identifies the element category. From ANSYS Main Menu select Preprocessor or Element Type→ Add/Edit/Delete. In response, the frame, shown in Figure appears. Click on [A] Add button and a new frame, shown in Figure , appears. Select an appropriate element type for the analysis performed, e.g., [A] Solid and [B] 20node 186 as shown in Figure . Element properties that depend on the element type, such as cross-sectional properties of a solid element. As with element types, each set of real constant has a reference number and the table of reference number versus real constant set is called the real constant table. Not all element types require real constant, and different elements of the same type may have different real constant values. ANSYS Main Menu command **Preprocessor → Modeling → Create →Element types**

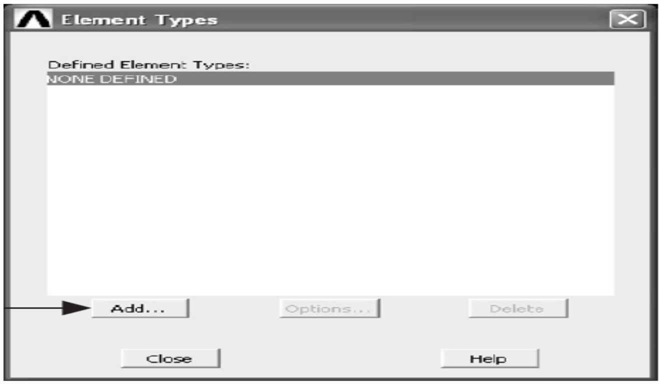

Figure 5:Tab for defining of the type of element used for analysis

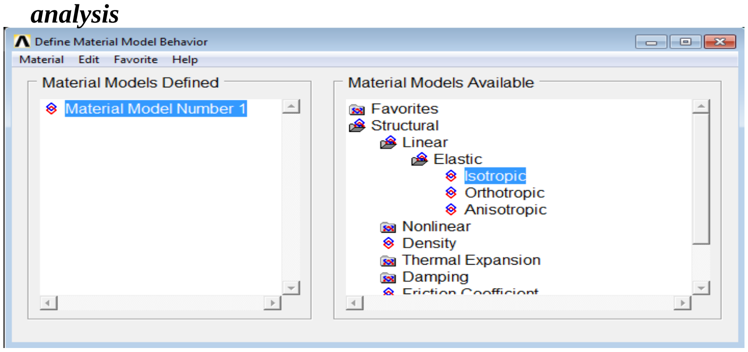

Figure7:Defining of the material properties for the layers

Defining Material Properties

Material properties are required for most element types. Depending on the application, material properties may be linear or nonlinear, isotropic, orthotropic or anisotropic, dependent. As with element types and real constants, each set of material properties has a material reference number. The table of material reference numbers versus material property sets is called the material table. In one analysis there may be multiple material property sets corresponding with multiple materials used in the model. Each set is identified with a unique reference number. Although material properties can be defined separately for each finite-element analysis, the ANSYS program enables storing a material property set in an archival material library file, then retrieving the set and reusing it in multiple analyses. The material library files also make it possible for several users to share commonly used material property data In order to create an archival material library file, the following steps should be followed:

(i) Tell the ANSYS program what system of units is going to be used from saved different systems of units.

(ii) Define properties of, for example, isotropic material. Use **ANSYS Main Menu** and select **Preprocessor→Material Props→Material Models**. A frame shown in Figure appears.

Define Material Model Behavior

As shown in Figure, [A] **Isotropic** was chosen. Clicking twice on [A] **Isotropic** calls up another frame shown in Figure 2.13. Enter data characterizing the material to be used in the analysis into appropriate field. For example, [A] **EX** =300 and [B] **PRXY** =0.3 as shown in Figure. If the problem requires a number of different materials to be used, then the above procedure should be repeated and another material model created with appropriate material number allocated by the program.

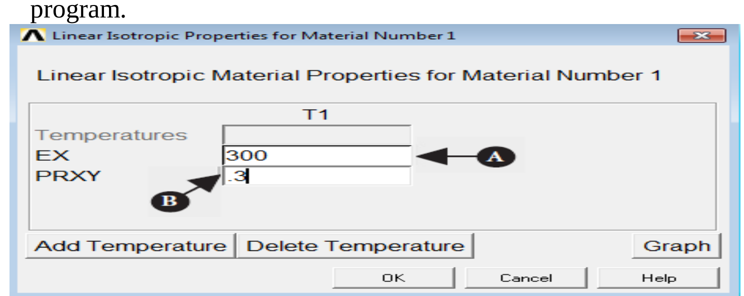

Figure 8:Input of the elastic properties i.e. elastic modulus and poison's ratio for each layer

And hence as above given procedure the required numbers of material properties are added and the required material properties were created as per need of the analysis.

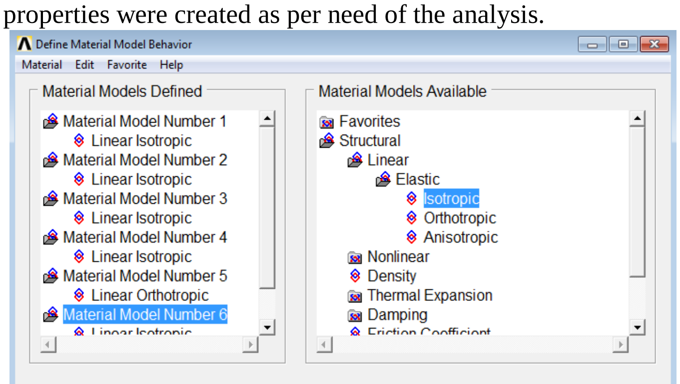

Figure 9:Defining the elastic material property for each model number

I. CONSTRUCTION OF THE MODEL

Once material properties are defined, the next step in an analysis is generating a finite element Model – nodes and element adequately describing the model geometry. There are two methods to create the finite-element model: solid modeling and direct generation. With solid modeling, the geometry of shape of the model is described, and then the ANSYS program automatically meshes the geometry with nodes and elements. The size and shape of the elements that the program creates can be controlled. With direct generation, the location of each node and the connectivity of each element are manually defined. Several convenience operations, such as copying patterns of existing nodes and elements, symmetry reflection, etc., are available. **Creating The Pavement Model:** The model can be created using the ANSYS. Use

ANSYSMainMenu→PreprocessorsModeling→Create→Volumes→Block→By dimensions as shown in below fig,

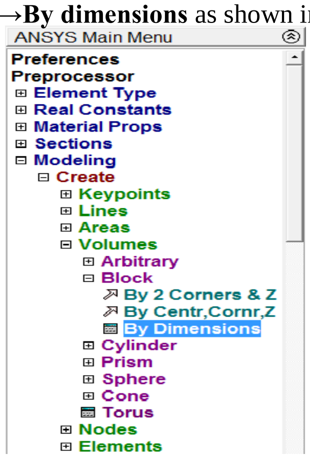

Figure 10:Modeling of the geometry of the flexible pavement layers

And by clicking on by dimension a dialogue box create block by dimension popped up and using the dialogue box the required model created layer by layer as the analyses required and required inputs are entered to get the desired model for the analyses.

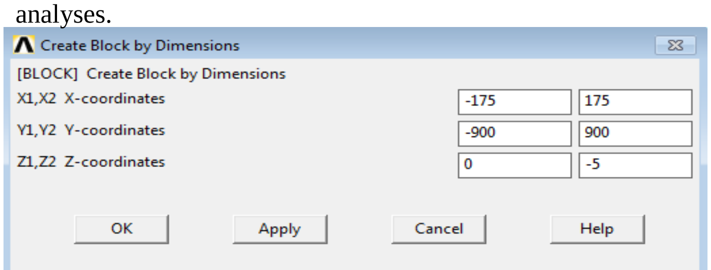

Figure 11:Input of dimensions for creating volumes by block defining length, breadth and thickness

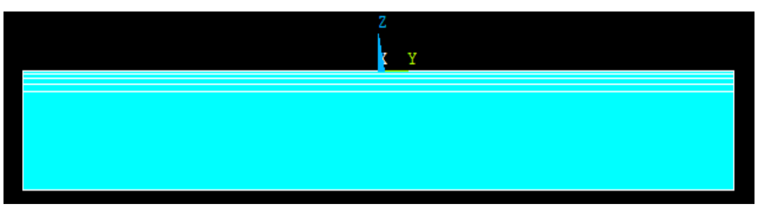

Figure 12:Longitudinal Section of the flexible pavement

VIII. ATTRIBUTES

Attributes are nothing but the material properties defining the behavior of the model. This step involves assigning of the predefined material properties to the individual layer volume of the flexible pavement. Also this step involves picking of the volume to which an individual material property is to applied. There are various methods of picking such as single, box, circle, polygon and loop. Single picking involves picking of a volume one at a time. Box picking involves dragging of the cursor in a window pattern or box pattern coinciding volume to be picked. Polygon method is used for picking the polygons or any geometry of irregular shape. The method of circle picking adopt picking by creation of circle around the geometry whose elements and nodes are to be circumscribed within the circle. Unpicking of specified volume can be done by selecting the unpick volume and specifying the method with which the volume was picked. While unpicking the cursor will be in reverse direction as that of while picking. **ANSYS Main Menu→ Preprocessor →Meshing Attributes→Picked volumes.** The Volume Attributes window opens as shown in below figure, Where in the picking method adopted is single and selected volume when picked turns pink

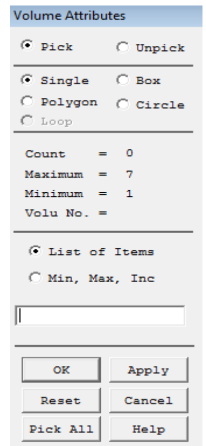

Figure 13:Volume attributes tab with single volume method of picking

(i) An upward arrow (↑) appears in the **ANSYS Graphics (Black)** window. Move this arrow to the beam area and click this area to mesh.
(ii) The colour of the volume turns from light blue into pink. Click **OK** button to see the volume meshed by 20-node rectangular finite elements as shown in fig.

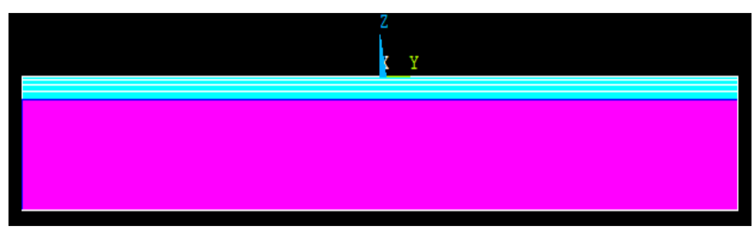

Figure 14:Longitudinal section of flexible pavement with volume of sub grade layer being picked to assign the material property defined in the material model properties

As above procedure mentioned all the elemental volumes created assigned with the specific properties which are given prior to the creating the module. In the material properties window.

Gluing all the layers

ANSYS MainMenu →Preprocessor →Modeling →Operate →Boolean→Glue→Volumes

A glue volumes window and upward arrow (↑) appears to select the layers to be glued in order to contact the layer and enhance the behavior of model as elastic. Picking of the individual layer is done by method of single picking. Two layers are picked at a time and are glued with each other. The process is repeated similarly for all the layers of the flexible pavement geometry model. Gluing is done before meshing. The color of the volume turns from light blue into pink. Click **OK** button to see the volumes glued by 20-node186 rectangular finite elements as shown in fig.

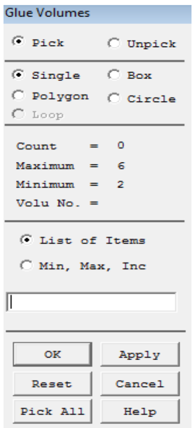

Figure 15:Glueing of the layers to ensure contact and for load transfer

IX. MESHING

Meshing involves the discretization of volumes into finite number of smaller elements by the concept of nodes and elements. Meshing the display of the of smaller blocks of the volume by creation of finite number of elements with finite number of nodes. Individual element is made up of 8 number of nodes. There are various methods of meshing such as mapped volume, volume sweep, line mesh, area mesh and tet mesh. Volume sweep involves sweep through the entire volume but if the element is poor this may not work. Mapped mesh involves creation of element with various face sided by distributing the entire volume into finite number of elements. Line mesh involves distribution of line into smaller element lines. Mapped volume with 4 to 6 sided is adopted.

ANSYSMainMenu→Preprocessor→Meshing→Mesh→ Volume→Mapped→4 to 6 sided

(i) The **Mesh Volumes** window opens.
(ii) The upward arrow appears in the **ANSYS Graphics** window. Move this arrow to the quarter plate area and click the volumes which are to be meshed.
(iii) The colour of the volumes turns from light blue into pink. Click the **OK** button to see the area meshed by 20-node186 isometric finite elements as shown in fig .

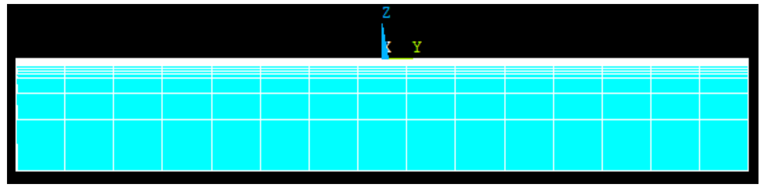

Figure 16:Creation of finite elements by concept of nodes and elements.

X. APPLICATION OF LOAD AND FIXING THE NODES

ANSYSMainMenu→Solution→DefineLoads→Apply→Structural→Displacement→On Nodes

(i) Applying restrains below the bottom layered nodes and both sides of the model .
(ii) The bottom layer is restrained in all DOF and at the sides the model restrained only in X-direction and allowing deformations in Z –direction. As shown in the fig,

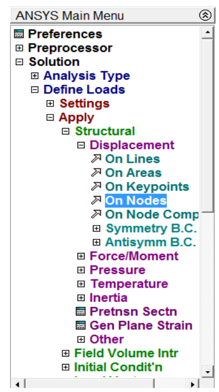

Figure 17:Restricting the degree of freedom

XI. APPLICATION OF LOAD ON NODE

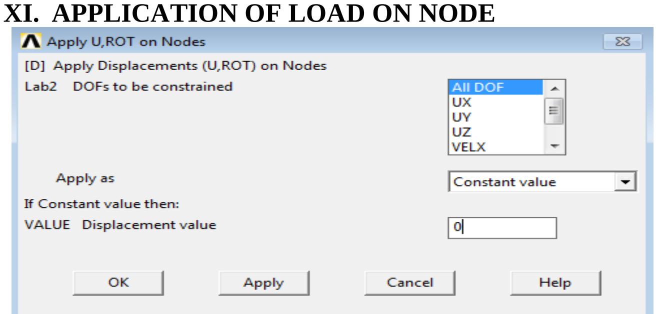

Figure 18:Restriction of DOF by applying displacement on nodes

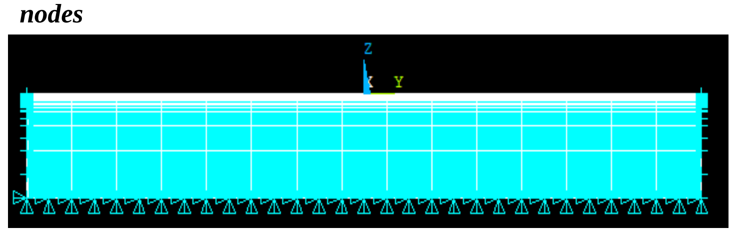

Figure 19:Ristricting all directions of rotations at the nodes

ANSYS UtilityMenu→PlotCtrls→Numbering in order to open the **Plot Numbering Controls** window as shown in Fig.

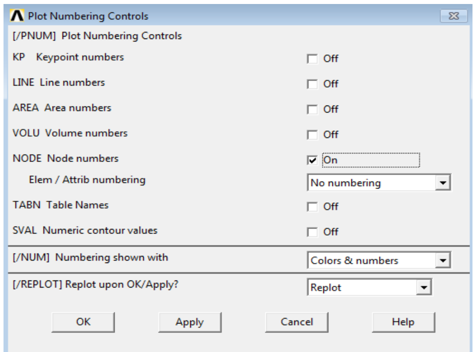

Figure 20:Switching on the numbering to determine the node numbers in the volume .

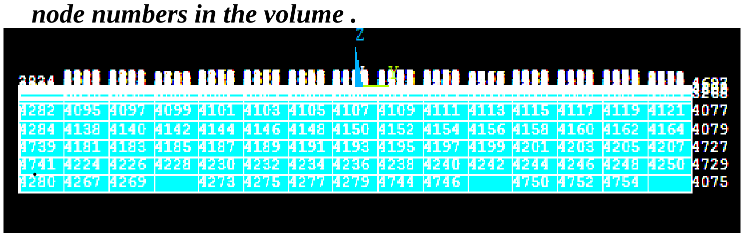

Figure 21:Display of the number for individual nodes

ANSYSMainMenu→Solution→DefineLoads→Apply→Structural→Force/Moment→On Nodes

Window popes up showing Apply F/M on Nodes and as per procedure the load should be applied at the center of the model and node having number 126 exactly at the center.

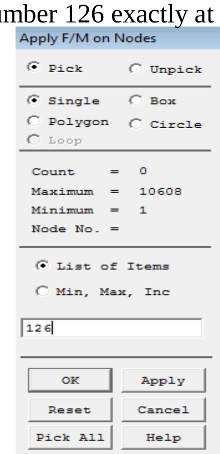

Figure 22:Applying point load on a specific node number

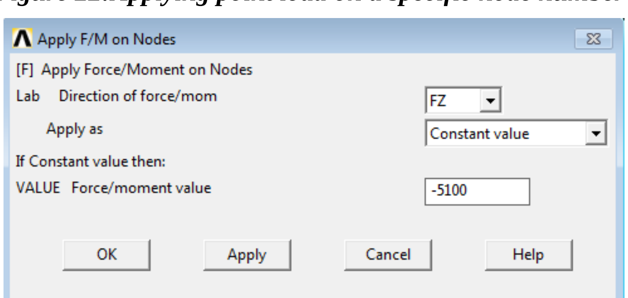

Figure 23:Magnitude and direction of point load is specified

X. ANALYSIS

Various types of analysis can be performed such a static, dynamic, transient, harmonic, spectrum, modal, buckling and substructuring. Static linear analysis is carried out after applying the loads and restraining the nodes then we have to analyze the model which is created as below steps. **ANSYS MainMenu→ Solution→ Analysis→ Analysis type**

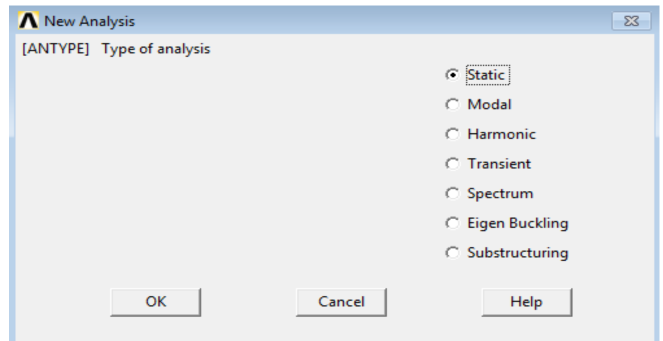

Figure24:Type of analysis is specified as Static

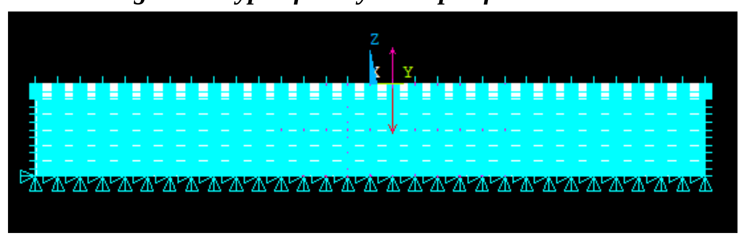

Figure25:Model with all assigned load conditions and material properties ready to be analyzed

Sparse Matrix-

In numerical analysis, a **sparse matrix** is a matrix in which most of the elements are zero. By contrast, if most of the elements are nonzero, then the matrix is considered dense. The fraction of zero elements over the total number of elements in a matrix is called the sparsity (density).

XII. SOLUTION

Solution is the end step of the analysis. Two types of solution can be obtained one is element solution and the other is nodal solution. Ansys involves obtaining various solutions as stress solutions,DOFsolution,Mechanical strain solution, creep strain solutions, plastic strain solutions etc.Once all the parameters such as load, material properties and degrees of freedom to be restrained are specified. After the static analysis being carried out the solution is obtained from the general post processing step as **General postprocess → list results → Nodal solution → stress → Z - component stresses.** As shown in **fig (a):**

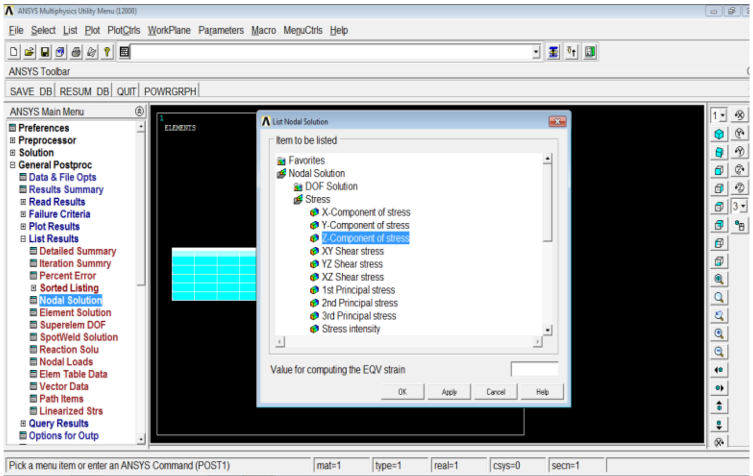

Figure 26:Obtaining the solution list as the Nodal solutions

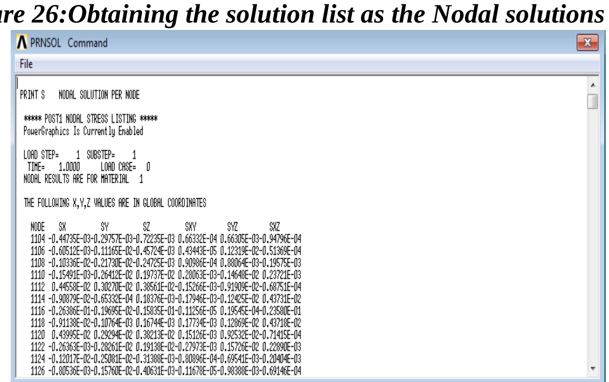

Figure 28: Results obtained is displayed with the node numbers and corresponding stresses on the nodes

XIII. RESULTS AND DISCUSSIONS

The interlayer bonding of modern multi-layered pavement system plays an important role to achieve long term performance of a flexible pavement. Better bonding condition at the interface between layers will cause the decreasing of critical strain. While increasing values of critical strain indicates poor bonding between the various layers namely sub-grade layer, sub-base layer, base layer, binder course and wearing course. This study is aimed to evaluate the bond strength at the interface between pavement layers by varying the elastic properties which contribute to the performance of pavement due to the effect of bond strength of the pavement[4].

Sub-grade Layer

The critical stresses on the sub grade layer mainly depends on the elastic property i.e. modulus of elasticity. The modulus of elasticity is varied for sub grade in the range of 300 to 700kg/cm^2 as specified by IRC 37:2001 and the various decreasing critical stress values are obtained as tabulated in table 1. It is observed that as the modulus of elasticity is increased the value of critical stress decreases indicating good bonding between the sub grade and top layers . The stresses are obtained by substitution of modulus of elasticity in the following equation:

$$y=0.002x-10.80 \qquad (1)$$

y- dependent variable, stress
x- independent variable, modulus of elasticity

Young's Modulus kg/cm^2	Critical Stress kg/cm^2
300	10.108
340	9.935
380	9.787
420	9.659
460	9.547
500	9.448
540	9.360
580	9.281
620	9.209
660	9.144

Table 1: Stress values for layer 1- Sub grade layer

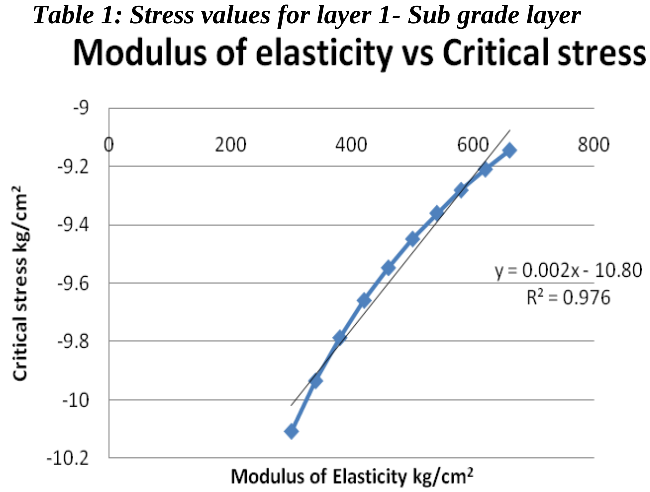

Figure 29: Plot of modulus of elasticity vs. critical stress as in table 1.

Sub-base layer

In order to access the bond strength of the top and bottom layers with respect to the sub base layer of 150mm thickness the Elastic modulus is varied in order to obtain critical stresses induced in the sub base layer. The modulus of elasticity is varied for sub base layer in the range of 700 to 1000kg/cm^2 as specified by IRC 37:2001 and the various stress values obtained as tabulated in table 2 . A graph of Stress vs Modulus of elasticity is plotted and stress is calculated as per equations . The stresses in sub base layer are obtained by substitution of various values of modulus of elasticity in the following equation:

$$y = 2.915\ln(x) - 29.14 \qquad (2)$$

y- dependent variable, stress
x- independent variable, modulus of elasticity
The modulus of elasticity are substituted in the above equation starting from 701 kg/cm^2 and corresponding stress values are tabulated. The modulus of elasticity is varied until the change critical stress values is negligible.

Young's Modulus kg/cm^2	Critical Stress kg/cm^2
701	10.108
730	9.9348
760	9.7869
790	9.659
820	9.5471
850	9.4484
880	9.3602
910	9.2809
940	9.2095
970	9.1446

Table 2:Stress values for layer2 sub-base layer

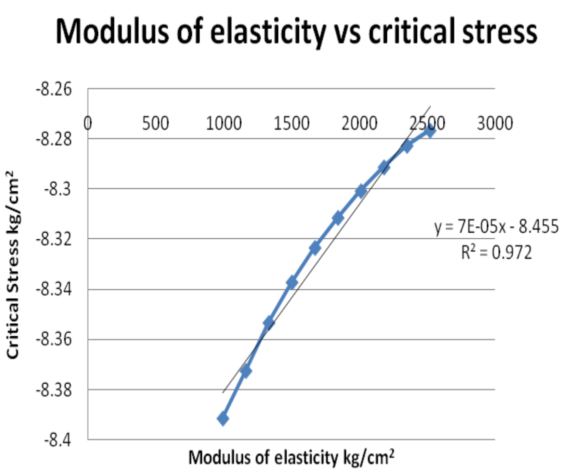

Figure 30: Plot of modulus of elasticity vs. critical stress given in table 2.

Base Layer

In order to access the bond strength of the top and bottom layers under the influence of wheel load of 5100kg with respect to the base layer of thickness 90mm whose Elastic modulus is varied in order to obtain stresses induced in the sub base layer. The modulus of elasticity is varied for sub base layer in the range of 900 to 2600kg/cm^2 as specified by IRC 37:2001 and the various stress values obtained as tabulated in table 3. A graph of Stress vs Modulus of elasticity is plotted and stress is calculated as per equations. Decrease in values of critical stress indicate good bonding. The stresses in base layer are obtained by substitution of various values of modulus of elasticity in the following equation:

$$y = 7E\text{-}05x - 8.455 \qquad (3)$$

y- dependent variable, stress
x- independent variable, modulus of elasticity

Young's modulus kg/cm^2	Critical Stress kg/cm^2
992	8.392
1162	8.373
1332	8.353
1502	8.337
1672	8.323
1842	8.311
2012	8.301
2182	8.291
2352	8.283
2522	8.276

Table 3: Stress values for Base layer

The modulus of elasticity are substituted in the above equation starting from 992kg/cm^2 and corresponding stress values are tabulated. The process is repeated until no further change in values stress is observed when exceeding the value of modulus of elasticity exceeds specified range .

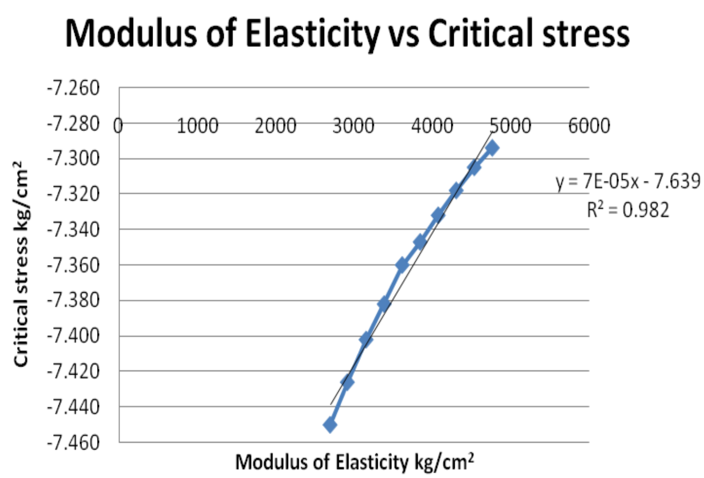

Figure 31: Plot of modulus of elasticity vs. critical stress given in table 3.

Binder Course

This layer has greater influence of bonding as it is in contact with the bonding layer. Stresses due to wheel load has greater impact on the performance of the pavement as load is transferred from wearing coarse to binder course through the bonding layer. This layer of thickness 100mm whose Elastic modulus is varied in order to obtain stresses induced in the sub base layer. The modulus of elasticity is varied for sub base layer in the range of 2600 to 4800 kg/cm^2 as specified by IRC 37:2001 and the various stress values obtained as tabulated in table 4. A graph of Stress vs Modulus of elasticity is plotted and stress is calculated as per following equation:

$$y = 7E\text{-}05x - 7.637 \quad (4)$$

y- dependent variable, stress
x- independent variable, modulus of elasticity

Young's modulus kg/cm^2	Critical stress kg/cm^2
2696	7.45
2916	7.426
3156	7.402
3386	7.382
3616	7.350
3846	7.347
4076	7.332
4306	7.318
4536	7.3056
4766	7.294

Table 4:Stress values for Binder course

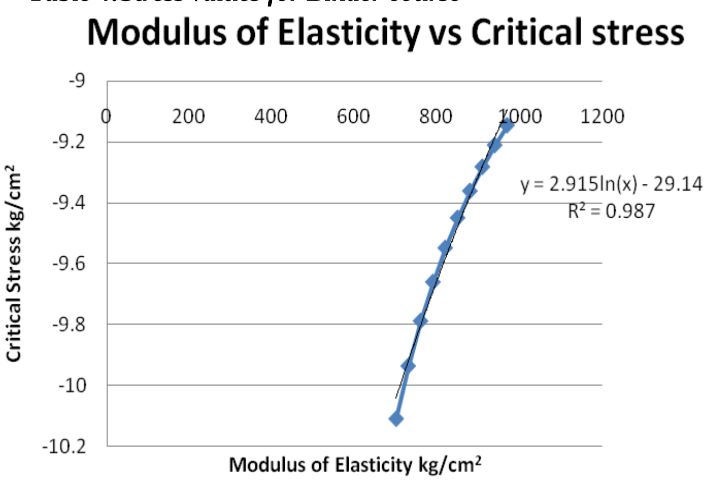

Figure 32: Plot of modulus of elasticity vs. critical stress given in table 4.

The modulus of elasticity are substituted in the above equation starting from 2696kg/cm^2 and corresponding stress values are tabulated. The process is repeated until no further change in values stress is observed when exceeding the value of modulus of elasticity exceeds specified range.

Bonding Layer

In order to ensure the load transfer from the surface of the pavement in important to ensure contact of the layers of the pavement with each other[6]. Bonding layer acts as contact for ensuring of proper bond between wearing course and binder course in order to determine bond stress under the influence of wheel load of 5100kg.The thickness of the bond layer is assumed to 10mm for determining the bond stresses. The bond strength is greatly influenced by shear modulus of the bond layer. The Shear modulus is varied in the range of 9000 to 12000 kg/cm^2.However the modulus of elasticity is kept constant. A graph of Stress vs Shear modulus is plotted and bond stress is calculated as per equations. The multilayer stress values for various layers are plotted and are used to evaluate bond strength. Various stress values for respective shear modulus are given in table 5. The stresses in bonding layer are obtained by substitution of various values of shear modulus for constant modulus of elasticity in the following equation:

$$y = 1E\text{-}05x - 8.502 \quad (5)$$

y- dependent variable, stress
x- independent variable, modulus of elasticity

The modulus of elasticity are kept constant and shear modulus are substituted in the above equation starting from 9000kg/cm^2 and corresponding stress values are tabulated. The process is repeated until no further change in values stress is observed when exceeding the value of modulus of elasticity exceeds specified range.

Shear modulus kg/cm^2	Critical Stress kg/cm^2
9000	8.418
9300	8.414
9600	8.411
9900	8.408
10200	8.405
10500	8.402
10800	8.400
11100	8.397
11400	8.394
11700	8.391

Table 5:Stress values for Bonding layer

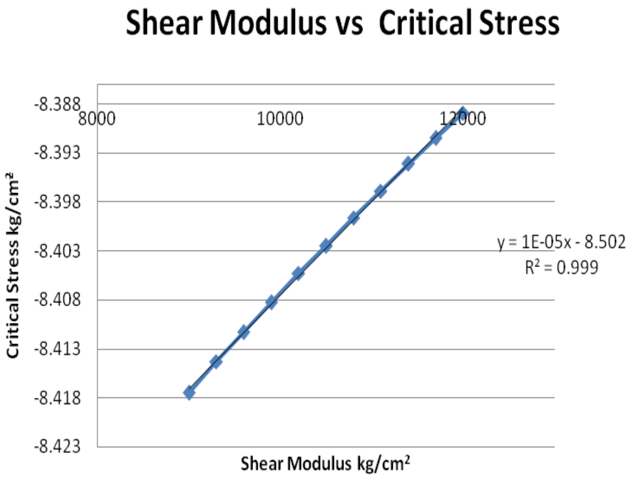

Figure 33: Plot of Shear modulus vs. critical stress given in table 5.

Wearing Course

This layer of the pavement is in direct contact with the wheel load hence the stress induced in this layer is maximum under the influence of load 5100kg.the stresses here are responsible for the pavement deformation as well as transferring these stresses to minimum for the lower layer. The thickness of this layer 50mm in which the stresses are influenced mainly due modulus of elasticity of the layer. The modulus of elasticity is varied for sub base layer in the range of 4033 to 7660 kg/cm^2 as specified by IRC 37:2001.Higher the value of stresses in this surface leads to poor bonding with the bottom layers leading to early failure of the pavement under the traffic. Various stress values for respective shear modulus are given in table 6. The stresses in wearing course are obtained by substitution of various values of modulus of elasticity in the following equation:

$$y = -1.25\ln(x) + 0.194 \qquad (6)$$

y- dependent variable, stress

x- independent variable, modulus of elasticity

Young's Modulus kg/cm^2	Critical Stress kg/cm^2
4033	10.237
4436	10.340
4839	10.439
5242	10.534
5645	10.626
6048	10.714
6451	10.799
6854	10.882
7257	10.962
7660	11.039

Table 6:Stress values for Wearing Course

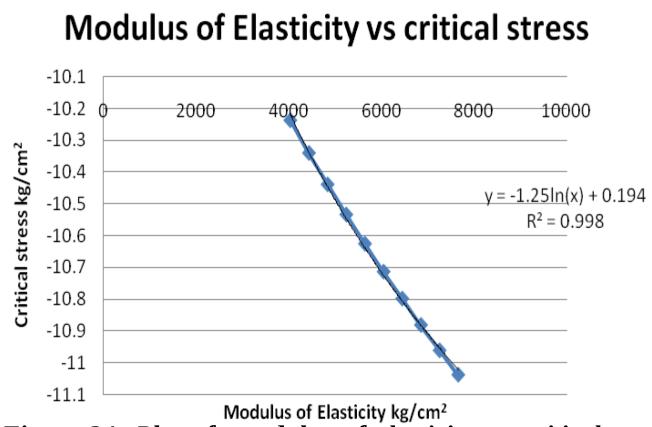

Figure 34: Plot of modulus of elasticity vs. critical stress given in table 6.

The modulus of elasticity are substituted in the above equation starting from 2696kg/cm^2 and corresponding stress values are tabulated

Multilayered Result

Multilayered results of the critical stresses are tabulated with respect to bond strength and the multilayered analysis is done using SPSS statistics software with the assigning of the dependent variables as critical stress values and modulus elasticity of the different layers as independent variable including the shear modulus of bonding layer in order to determine the bond strength. Multilayered analysis involves assigning of various values of elastic modulus for various layers keeping other parameters and the data is as tabulated below:

Table 7:Multilayered analysis of the bond strength

Critical stress kg/cm2	**bond strength**	**Modulus of elasticity**				
		layer 1	**layer 2**	**layer3**	**layer4**	**layer5**
8.418	9000	4033	2696	992	701	300
8.414	9300	4033	2696	992	701	300
8.411	9600	4033	2696	992	701	300
8.408	9900	4033	2696	992	701	300

8.405	10200	4033	2696	992	701	300
8.403	10500	4033	2696	992	701	300
8.400	10800	4033	2696	992	701	300
8.397	11100	4033	2696	992	701	300
8.394	11400	4033	2696	992	701	300
8.392	11700	4033	2696	992	701	300
10.237	11700	4033	2696	992	701	300
10.340	11700	4436	2696	992	701	300
10.439	11700	4839	2696	992	701	300
10.534	11700	5242	2696	992	701	300
10.626	11700	5645	2696	992	701	300
10.714	11700	6048	2696	992	701	300
10.799	11700	6451	2696	992	701	300
10.882	11700	6854	2696	992	701	300
10.962	11700	7257	2696	992	701	300
11.039	11700	7660	2696	992	701	300
7.450	11700	7660	2696	992	701	300
7.426	11700	7660	2916	992	701	300
7.402	11700	7660	3156	992	701	300
7.382	11700	7660	3386	992	701	300
7.350	11700	7660	3616	992	701	300
7.347	11700	7660	3846	992	701	300
7.332	11700	7660	4076	992	701	300
7.318	11700	7660	4306	992	701	300
7.306	11700	7660	4536	992	701	300
7.294	11700	7660	4766	992	701	300
8.392	11700	7660	4766	992	701	300
8.373	11700	7660	4766	1162	701	300
8.353	11700	7660	4766	1332	701	300
8.337	11700	7660	4766	1502	701	300
8.323	11700	7660	4766	1672	701	300
8.311	11700	7660	4766	1842	701	300
8.301	11700	7660	4766	2012	701	300
8.291	11700	7660	4766	2182	701	300
8.283	11700	7660	4766	2352	701	300
8.276	11700	7660	4766	2522	701	300
10.108	11700	7660	4766	2522	701	300
9.935	11700	7660	4766	2522	730	300
9.787	11700	7660	4766	2522	760	300
9.659	11700	7660	4766	2522	790	300
9.547	11700	7660	4766	2522	820	300
9.448	11700	7660	4766	2522	850	300
9.360	11700	7660	4766	2522	880	300
9.281	11700	7660	4766	2522	910	300
9.210	11700	7660	4766	2522	940	300
9.145	11700	7660	4766	2522	970	300
10.108	11700	7660	4766	2522	970	300
9.935	11700	7660	4766	2522	970	340
9.787	11700	7660	4766	2522	970	380
9.659	11700	7660	4766	2522	970	420
9.547	11700	7660	4766	2522	970	460
9.448	11700	7660	4766	2522	970	500
9.360	11700	7660	4766	2522	970	540

9.281	11700	7660	4766	2522	970	580
9.209	11700	7660	4766	2522	970	620
9.144	11700	7660	4766	2522	970	660

SPSS Statistics

Linear regression is the next step up after correlation. It is used when we want to predict the value of a variable based on the value of another variable. The variable we want to predict is called the dependent variable (or sometimes, the outcome variable). The variable we are using to predict the other variable's value is called the independent variable (or sometimes, the predictor variable).If you have two or more independent variables, rather than just one, you need to use multiple regression. In the present case the values of layers are tabulated varying one particular property by keeping other values constant. **Procedure for multilayer Analysis:**

1. Run the SPSS Statistics 17.0,a window appears as:

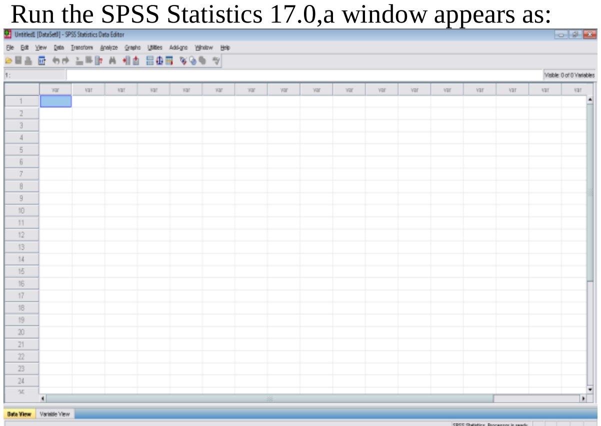

Figure 35: data sheet for entering the variables involved in analysis.

2. Enter data of multilayer in various variable columns:

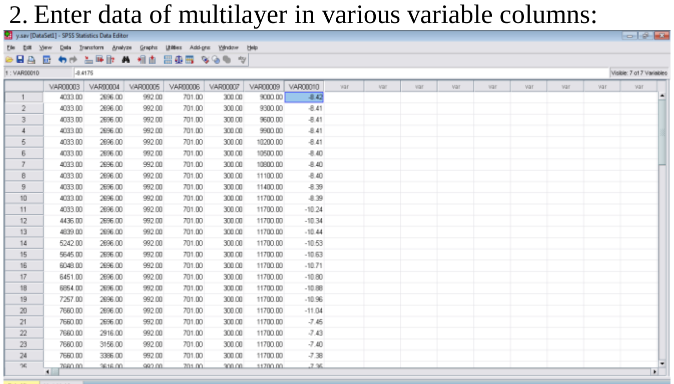

Figure 36:The data entered for multilayer regression analysis

3. Go to the Analyze tab select regression as linear a tab pops as shown below consisting of tabs for dependent and independent variables. The stresses are assigned as dependent variables and all other parameters as independent variables and select ok as shown in figure 3:

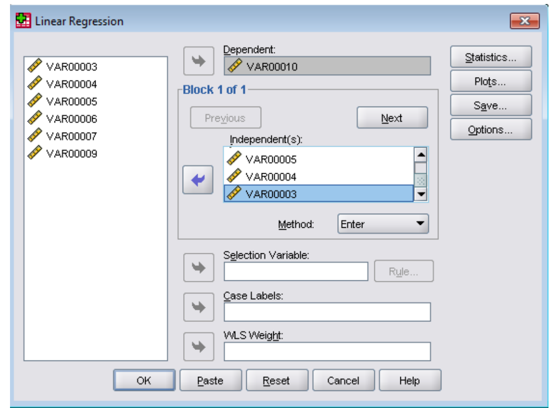

Figure 37:Variable column 10 is assigned as independent variable which is critical stress and other variable columns as dependent variables.

4. The analysis consists of various tests such as ANOVA Table, Model summary and coefficients table as output as shown in figure 4:

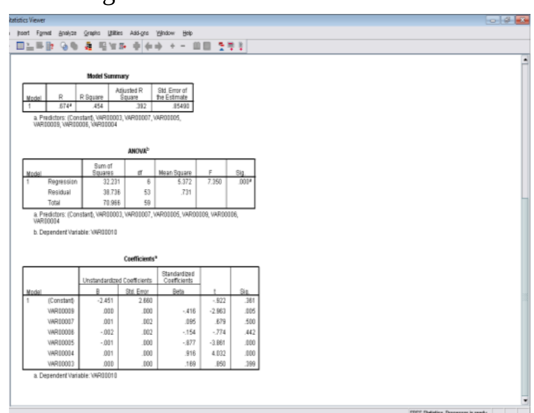

Figure 36:Values of coefficients obtained for various layers by method of F -test and T- test

Stress is computed as given by equation:

$y = a_1x_1 + a_2x_2 + a_3x_3 + a_4x_4 + a_5x_5$......

where y= Bond stress

a_1 - Coefficient for sub-grade = 0.000

a_2 - Coefficient for sub-base= 0.001

a_3 - Coefficient for base=-0.001

a_4 - Coefficient for binder course=-0.002

a_5 - Coefficient for surface layer=0.001

Bond stress for a particular set of values is calculated as:

Stress =(0.000 x Modulus of elasticity of sub grade +0.001 x Modulus of elasticity of sub base - 0.001 x Modulus of elasticity of base - 0.002 x Modulus of elasticity of binder course + 0.001 x Modulus of elasticity of surface)

T test

The t-test looks at the **t**-statistic, t-distribution and degrees of freedom to determine a p value (probability) that can be used to determine whether the population means differ. The **t-test** is one of a number of hypothesis tests. To compare three or more variables, statisticians use an analysis of variance (ANOVA).

F Test

An F-test i.e. test of best fit is any statistical test in which the **test** statistic has an **F**-distribution under the null hypothesis. It is most often used when comparing statistical models that have been fitted to a data set, in order to identify the model that best fits the population from which the data were sampled.

14. Conclusions

Following conclusions could be drawn from this study:

1. The equation for stress in sub grade layer is given by the equation: y=0.002x-10.80
2. The equation for stress in sub base layer is given by the equation: y = 2.915ln(x) - 29.14
3. The equation for stress in base layer is given by the equation: y = 7E-05x - 8.455
4. The equation for stress in binder course is given by the equation: y = 7E-05x - 7.637
5. The equation for stress in bonding layer is given by the equation: y = 1E-05x - 8.502
6. The equation for stress in surface layer is given by the equation: y = -1.25ln(x) + 0.194
7. Bond strength is obtained by the following equation:

Y =(0.000 x Modulus of elasticity of sub grade +0.001 x Modulus of elasticity of sub base -0.001 x Modulus of elasticity of base - 0.002 x Modulus of elasticity of binder course + 0.001 x Modulus of elasticity of surface)

8. Better bonding condition at the interface between various layers of the flexible pavement is ascertained by the decreasing values of critical strain for wearing course, contact layer, binder course, base layer and sub base. As the results, the better structural capacity can be achieved with better bonding between layers.
9. The performance of the pavement under given static wheel load has deformed elastically inducing maximum and minimum stresses. The minimum or critical stress values are obtained from the equations generated.
10. The variation of the shear modulus has direct impact on performance of the pavement.
11. The multilayer analysis of the pavement is analysed and the bond stress is determined based on the equations.

References:

[1] Nagrale Prashant P, More Deepak (2014),Benefits Of Mechanistic Approach In Flexible Pavement Design, *Indian Highways* Vol.42 No.3.

[2] Xiaoyang Jia Baoshan Huang and Lihan (2013), Evaluating Interlayer Shear Resistance In Asphalt Pavement, Li-Tongji University Shanghai

[3] Xingsong Sun,(2011), Modelling Analysis Of Pavement Layer Interface Bonding Condition Effects On Cracking Performance, *A Dissertation Presented To The Graduate School Of The University Of Florida In Partial Fulfillment Of The Requirements For The Degree Of Doctor Of Philosophy*, University Of Florida

[4] Bidyut Bikash Sutradhar(2010), Evaluation Of Bond Between Bituminous Pavement Layers, *A thesis submitted in Partial fulfilment of the Master Of Technology In Civil Engineering*.

[5] Muslich Hartadi Sutanto, ST,MT, (2009),Assessment Of Bond Between Asphalt Layers, PhD thesis, University of Nottingham.

[6] Horak, E., Emery. S., Maina, J. W., Walker, B. (2009) Mechanistic Modelling of Potential Interlayer Slip at Base Sub-Base Level. *8th International Conference on the BearingCapacity of Roads, Railways, and Airfields.*

[7] Maurice Wheat (2007), Evaluation Of Bond Strength At Asphalt Interfaces. *A Report submitted in partial fulfilment of the requirements for the degree Master Of Science Department of Civil Engineering College of Engineering KANSAS STATE UNIVERSITY*, Manhattan, Kansas

[8] Laith Tashman(2006**)**, Evaluation Of The Influence Of Tack Coat Construction Factors On The Bond Strength Between Pavement Layers, *Report Prepared for Washington State Department of Transportation.*

[9] Hakim, B. A. (2002) The Importance of Good Bond between Bituminous Layers. *Proceedings of the 9th International Conference on the Structural Design of Asphalt Pavements.*

[10] Hachiya, Y. and Sato, K. (1998) Effect of Tack Coat on Bonding Characteristics at Interface between Asphalt Concrete Layers. *Proceedings of 8th International Conference on Asphalt Pavements, Vol. 1. pp. 349-362.*

[11]West, R., Moore, J., and Zhang, J. (2005). *Evaluation of Bond Strength Between Pavement Layers*. NCAT Report 05-08, National Center for Asphalt Technology, Auburn, Alabama