Analysis and Design of Bituminous Pavements

Prof. J. Murali Krishnan

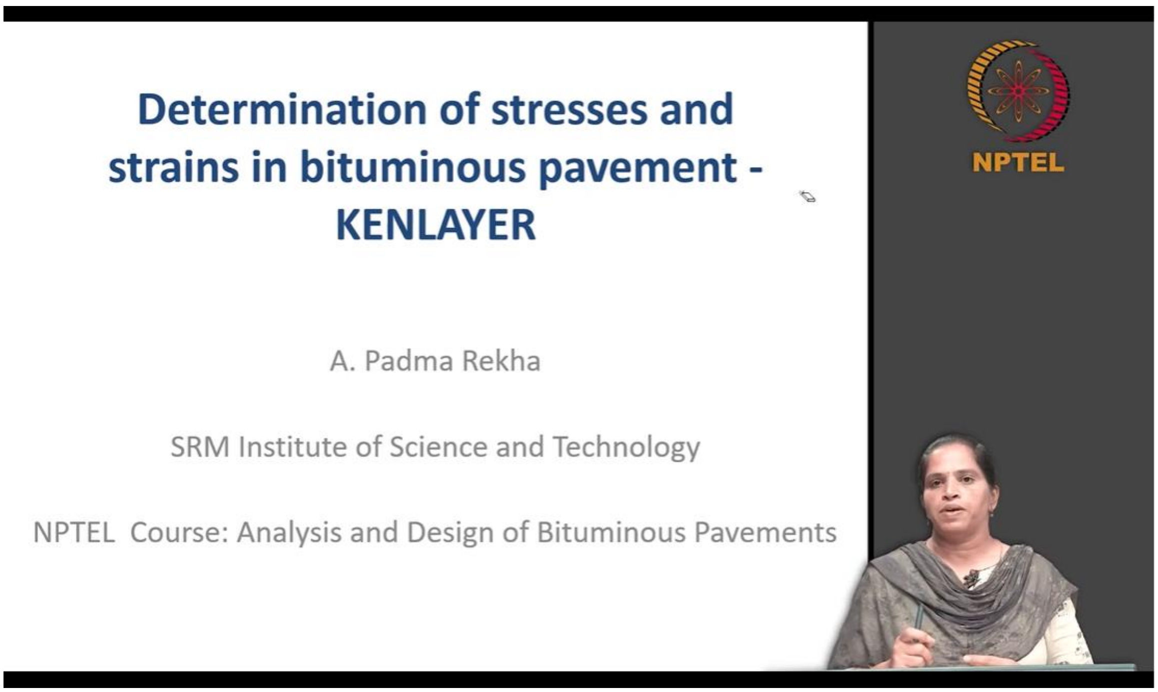
Department of Civil Engineering

Indian Institute of Technology, Madras

Lecture - 09

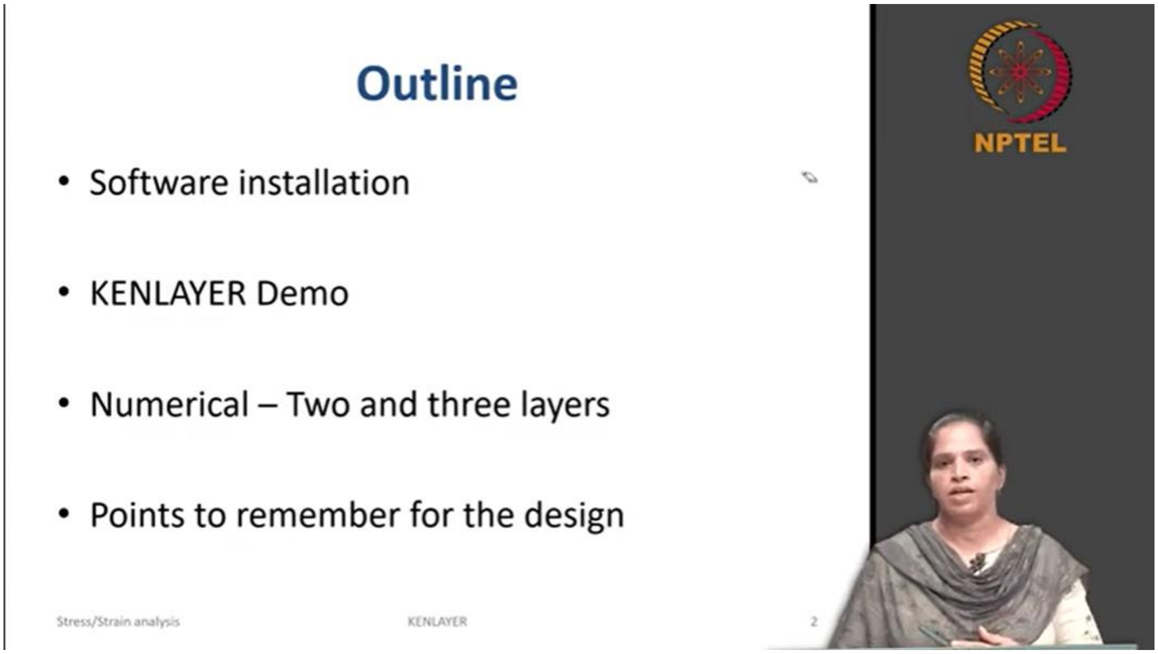
KENLAYER - 1

(Refer Slide Time: 00:23)



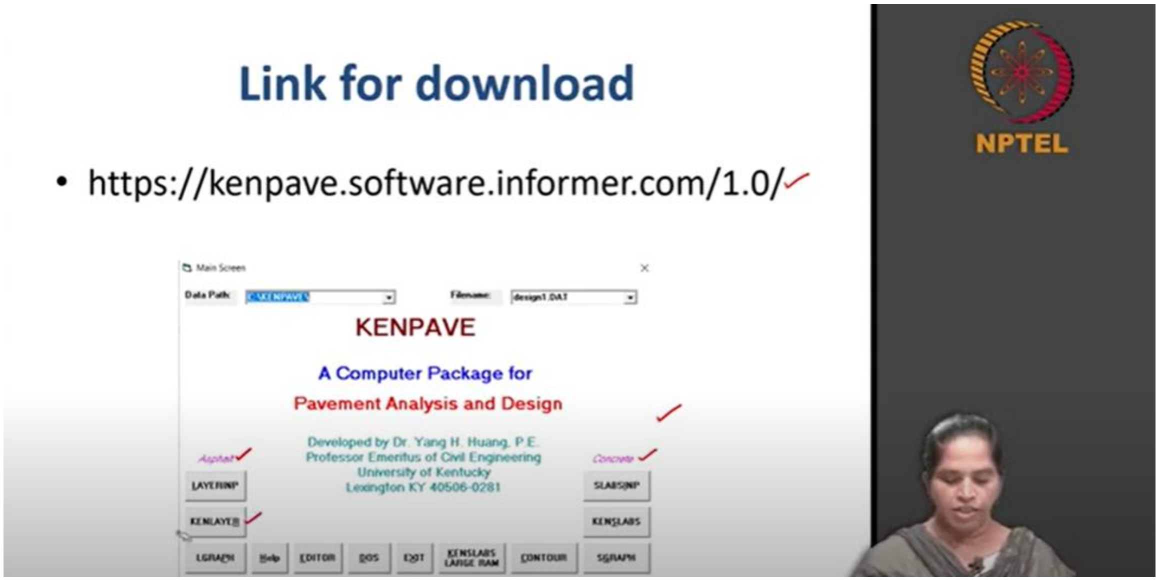
Hello everybody, in this course so far, you have learned how to determine stresses and strain in the bituminous pavement using a design charts. Now, we will do the same thing using a software called KENPAVE. So, in a KENPAVE software, there is a module called KENLAYER. So, this KENLAYER is used for determining stresses and strain in the bituminous pavement. So, we will see how to use a KENLAYER for the analysis purpose. So, this is going to be the outline of the presentations.

(Refer Slide Time: 00:45)



So, first we will install the software. Once we install it, we will see a general demo on the software so that we will get familiarized with the terminologies used in determining the stresses and strains. Once we familiarize with all the terminologies, we will solve few numerical on two layered system and a three-layered system using different axle configurations. So, further after understanding all the features of the KENLAYER and understanding the results we get it, we will consolidate all our learnings here so that we can use these learnings further when we do a pavement design.

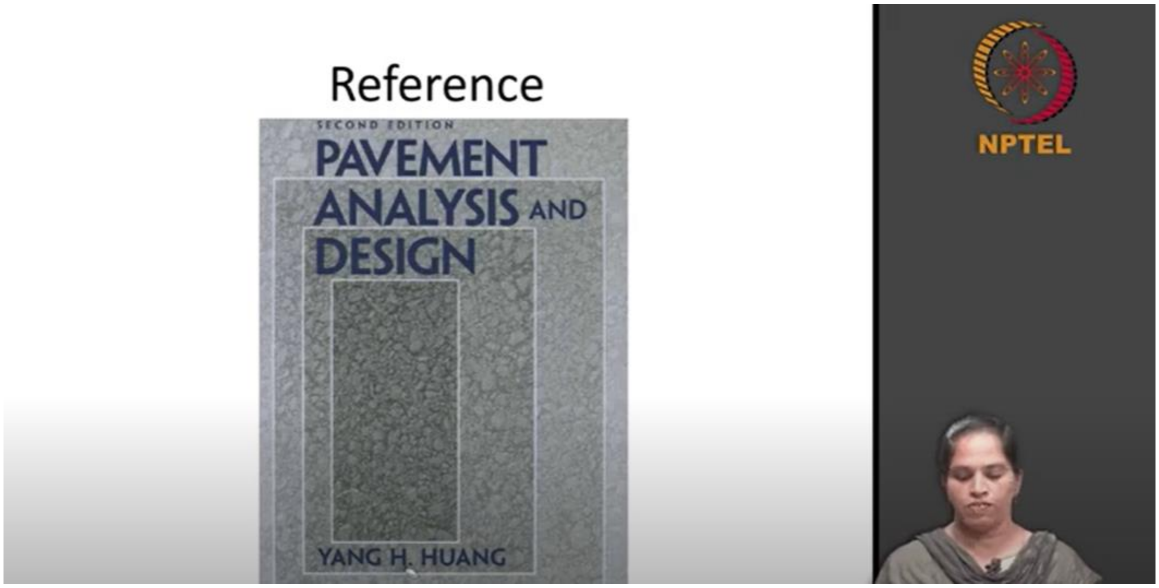
(Refer Slide Time: 02:17)



So, this is the link for downloading the software. This is an open-source software. You can download it and you can install the software using a setup file in the folder. After the installation, you open the KENPAVE software. The main screen of the KENPAVE software looks something like this. So, in this main screen, you can see two modules here.

One is corresponding to asphalt pavement and second is corresponding to the concrete pavement. So, in asphalt pavement, we are interested in the asphalt pavement module. So, if you can see this asphalt pavement module, the program main button is named as a KENLAYER, that is why we call it as a KENLAYER software here. So, we are going to now focus only on the half part of a software which corresponds to the analysis of stresses and strain in the bituminous concrete pavement.

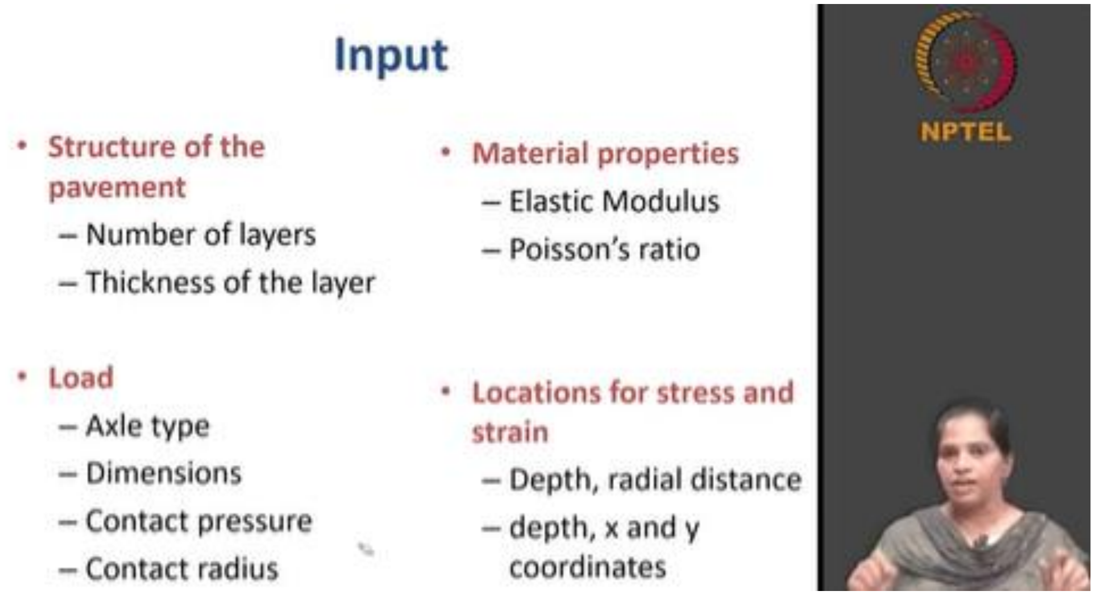
(Refer Slide Time: 02:34)



All the numerical solved are from this reference Pavement Analysis and Design by Huang.

So, I hope all of you are now aware of this software. And Professor Murali already introduced the software to you and he already explained about the principle behind the determination of stresses and strain. He explained about the superposition concept in which you have a dual wheel load or multiple wheels. And, you now know what is the principle behind determining stresses and strain. So, I am directly going to start a demo on this software.

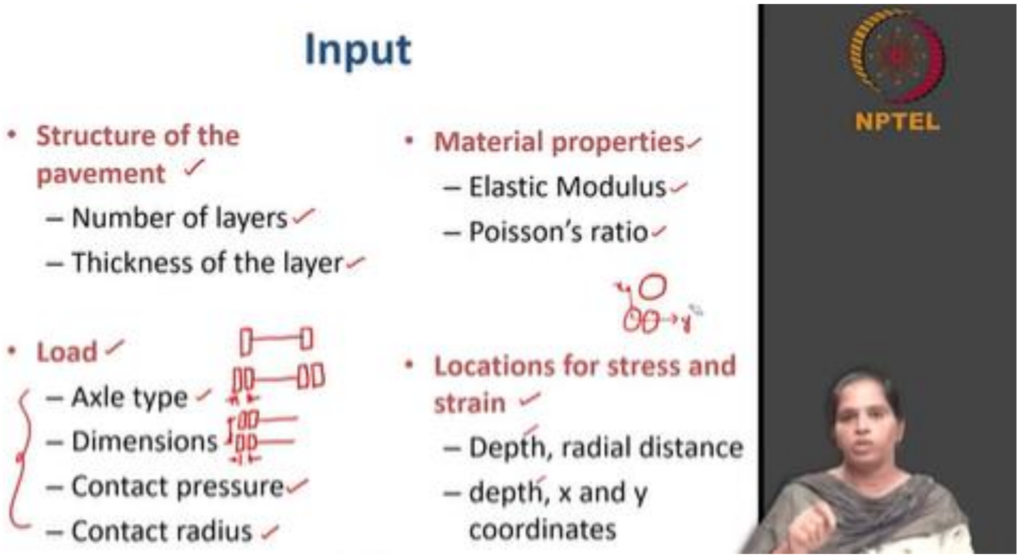
(Refer Slide Time: 03:10)



So, before we see the screen of the software, we will just consolidate what are the inputs and outputs you are going to get from the software. So, when we start with a demo, you can group the type of input based on this category. So, first input is the structure of pavement. So, you define the structure of pavement using number of layers and thickness of each layer. So, you need a thickness of each layer and number of layers. So, giving this, you can define what the structure of pavement exactly.

Now, you have a material property, you know it is a linear elastic theory. So, to determine stresses and strain in the linear elastic layers, you need the elastic modulus of different layers and you need the Poisson's ratio of each layer.

(Refer Slide Time: 07:47)

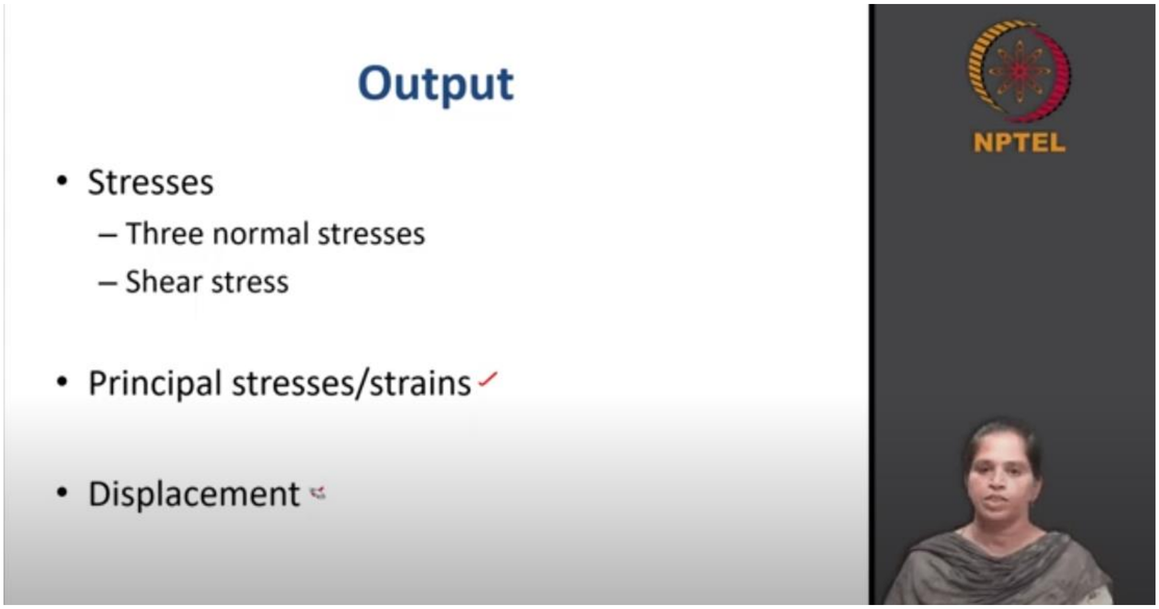


Now, you need to tell what exactly the load that is acting on the structure. You know it is going to be a wheel load. So, to define what exact wheel load is acting on the structure, these are the data you need. So, first we will see what is axle type, you know there are 4 types of axle configuration, single axle single wheel, which looks something like this, only one wheel on one side and axle is also only one axle. Or, it can be a single axle dual wheel. So, 2 wheels on each side, which are spaced at some distance from each other, but you have only one axle. In a tandem axle, you have 2 axles, so 4 wheels on each side. So, now, you have to define this wheel position exactly, you may be needing centre to centre distance between 2 wheels and centre to centre distance between 2 axles. Similarly, if you have a tridem axle, you will have another 2 wheels added to this system of tandem axle. So, you have 4 different types of axle configurations. So, for determining stresses and strain, you can use any of these load configurations because all these loads are common in a pavement. So, you can use any of these loads as required and determine stresses and strains in the critical locations. So, you need what type of axle we are going to use and you need to define the dimensions here.

So, in addition to distance between 2 wheels and distance between 2 axles, you also need what is the magnitude of load that is going to act on the pavement. So, the magnitude of load is defined here using a contact radius and contact pressure. So, we assume that the wheel load is distributed to a pavement uniformly in a circular pattern. So, what is the contact pressure followed by what is the contact radius? So, by defining these 2 parameters, you can exactly define what is the load that is going to act on the pavement.

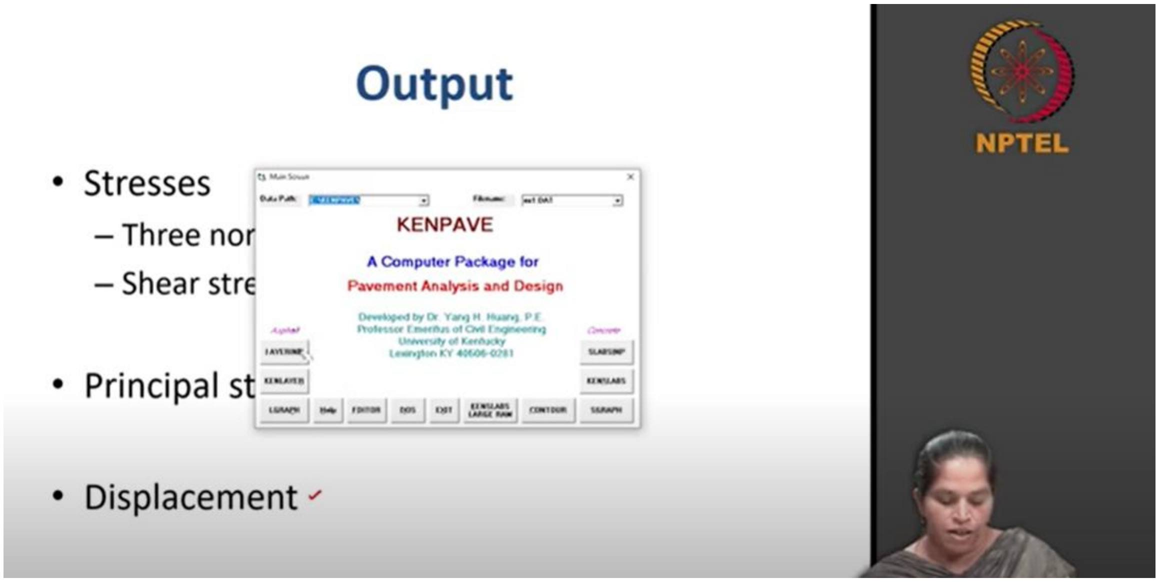
And finally, locations of stresses and strain, where exactly you need to determine stresses and strain. So, you need to give the depth at which you need stresses and strain. Do you need it at the surface of the pavement or at the interface of 2 layers or middle of some layer? So, you need to define the depth, depth is referenced from the top surface, 0 at the surface and when you go down, it increases. And to give the radial distance, what is the radial distance from the center of loading? Generally, if it is single axle single wheel, the plan looks like this, we assume it to be circular, the center point of the loading is considered as a 0 point. So, if it is any axle other than single axle, say if you have a single axle dual wheel, so you have 2 wheels, so center is defined from the center of one loading. So, in such case, you need x coordinate and y coordinate. So, this direction (vertical, bottom to top) is y and this direction (horizonal – left to right) is defined as x. So, you can define in addition to depth, x and y coordinates to tell what locations you need to determine stresses and strains. So, these are the 4 groups of input we need to give to the software to determine stresses and strains.

(Refer Slide Time: 08:53)



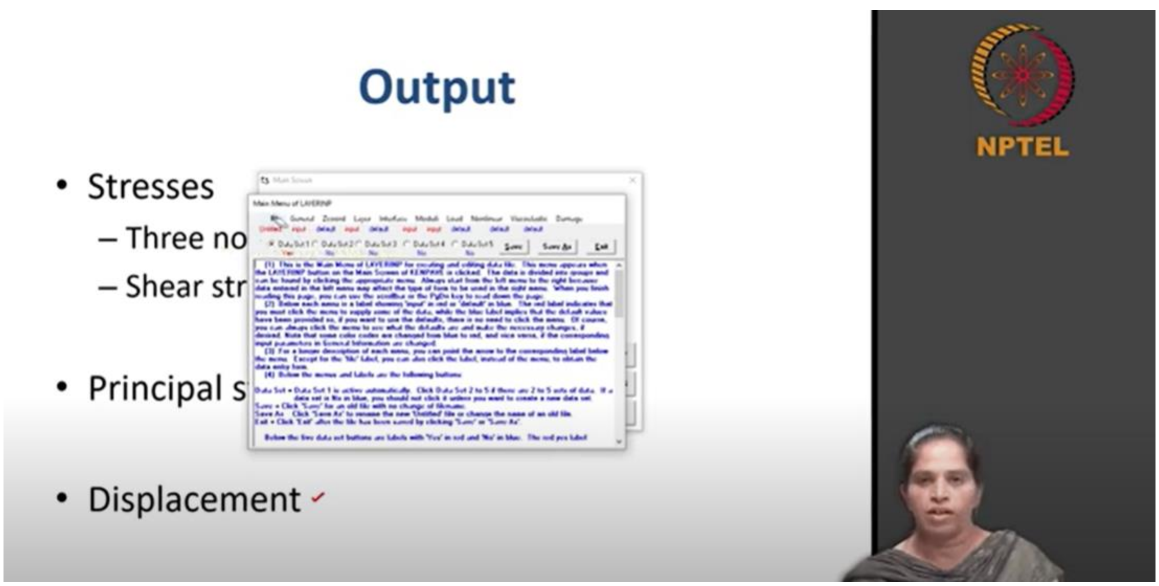
So, now what is the output you get. So, you all know that we need stresses and strain. So, output will be 3 normal stresses and shear stress in case if it is a single axle single wheel. So, in case of a single axle single wheel, the critical location is exactly at the center where the shear stress is 0. So, the normal stress itself will be the principal stresses and principal strain. So, if the load is different from a single axle, so if you have 2 tires, 4 tires or 6 tires, then the principal stress is going to be the critical stress and you need to determine principal stress from the 3 normal stresses and a shear stress value. So, for the design purpose, we will be using principal stresses and strain. So, you will also get displacement along with the stresses and strain. So, this is the output what we will get from the KENLAYER software.

(Refer Slide Time: 09:10)



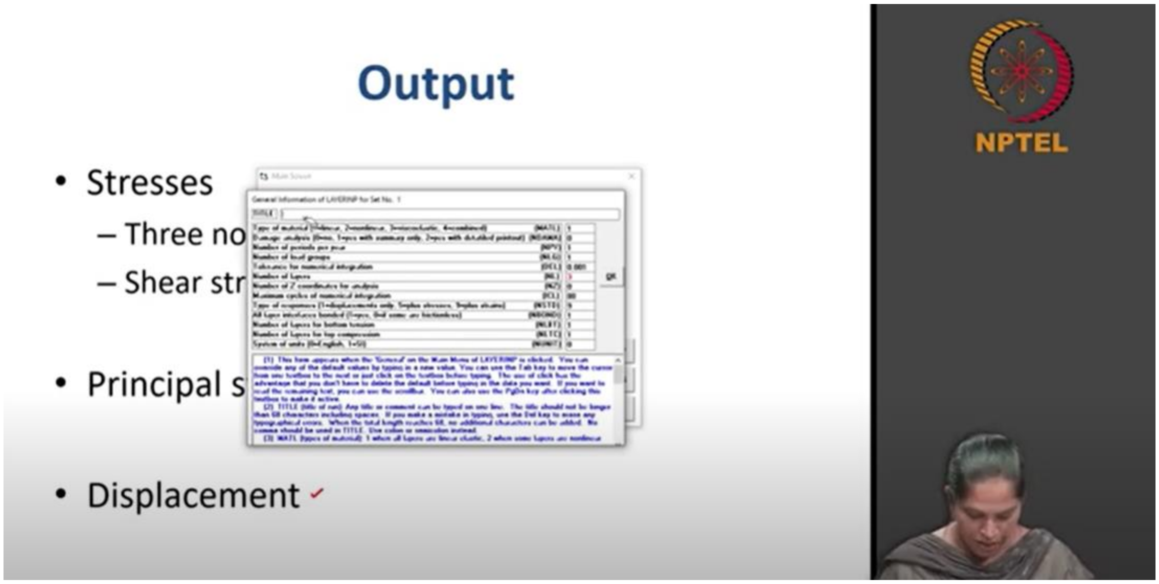
Now, let us go to the software and we will see the demo of the software. So, this is the main screen of the software. We are going to use the KENLAYER portion of the KENPAVE software. So, let us open the KENLAYER input, we call it as layer input.

(Refer Slide Time: 09:24)



So, if you open the layer input, you can see a file here, you can use a new file. So, before the new file gets saved, it will be named as 'untitled'. We will save it after we give all the input here. So, let us go to the general input. So, in a general input, the first is going to be the type of material.

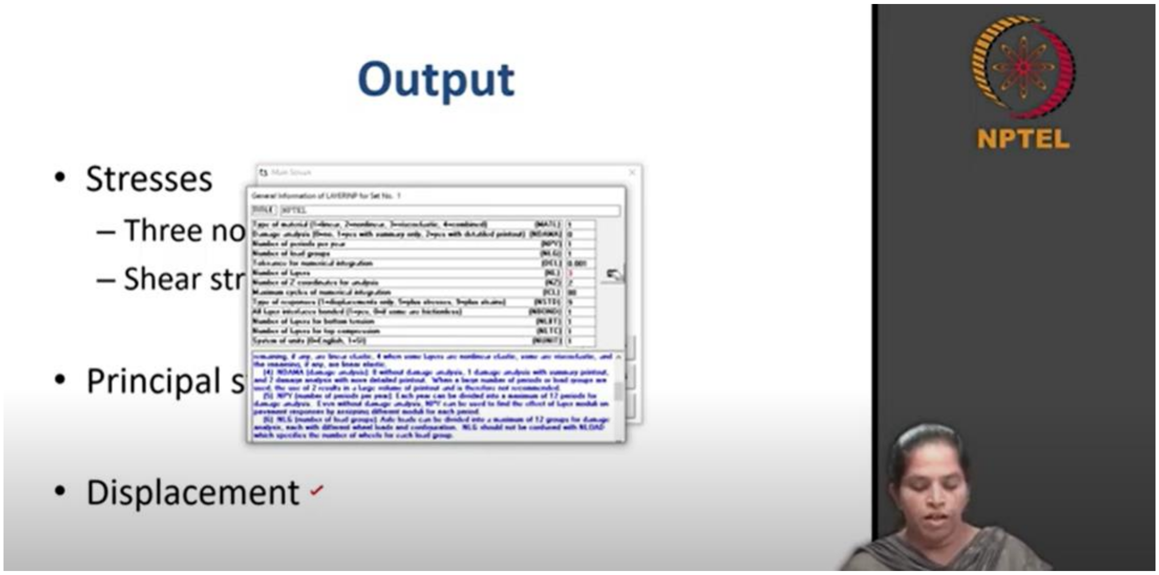
(Refer Slide Time: 09:38)



So, before we see the input related to the pavement structure, let us title it, I will title it as NPTEL. So, now, we need to give what type of material it is. Here, if you focus on what type of material we are going to use, you have 1 for linear, 2 for nonlinear, you need to assign 3 for viscoelastic material and 4 for a combined behavior. So, you know that whatever stresses and strain we are determining here, we are going to use a linear elastic theory for this. So, you can assign 1 for the linear elastic theory. Let us not worry about a nonlinear behavior, viscoelastic behavior or a combination of any of this in this case now. So, once we give the type of material, we need to define whether we need to perform a damage analysis or not. As of now, we have not learned about damage analysis. So, once we learn the theory behind the damage analysis, we will come back to this. So, as of now, we will keep damage analysis to be 0. Followed by this, you have an input for number of periods per year. This is something that facilitates us to define different modulus for different month. For example, if you take a subgrade layer, you can have different modulus for every month or in a winter you have different modulus value, in a summer you have a different modulus value. So, this allows us to define different modulus for different period.

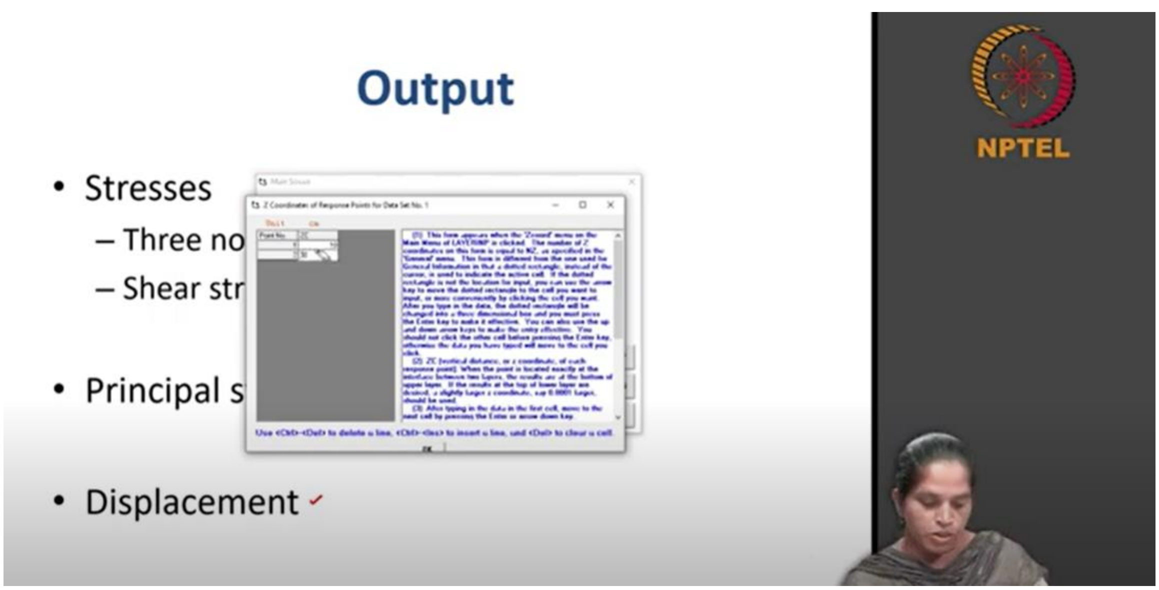
So, you can go up to maximum of 12 that is 1 modulus per month in a year. So, as of now, we will give only 1 modulus and we assume that the modulus value is going to be constant throughout the year. So, here the number is going to be 1. Number of load groups - this is the place where you define how many numbers of axle the pavement is going to get subjected to. It is not based on the load intensity, it is based on the axle configurations. And in case if the contact area is different, you can use different load groups, but you have a constraint here, you have the number of load groups to be defined as limited. You can just read through the help menu available here and find out what is the number of load group we can define. It is generally 12 numbers, but you can always use the help menu here to understand what exactly these terms are. So, followed by number of load groups, you have tolerance for numerical integrations. So, as of now, the default value of tolerance which is 0.001 is given. So, let us stick to the default value. So, this tolerance is to terminate the program. So, when you run a program, we get the results based on a numerical integration. So, when the integration converges, this is the tolerance value it uses. Number of layers, here, let us use a two-layer system or three-layer system, for the demo purpose, I will give this number to be 3. Number of z coordinate for analysis - this z coordinate is the depth. So, at what depth you need the stresses and strain value. So, you need to define a number here, which is the exact depth. So, let me give two numbers, it indicates that at two different depths, I need stresses and strain. Maximum cycle for numerical integration, tells you what number of integrations that you have to go for. So, here let us stick to the default value and we will see how many integrations the software has used for converging the result later when we see the result from the software. So, type of response, you can use 1 for displacement, 5 for displacement and stress, 9 for displacement stresses and strain. Let us give 9 here so that we see all three values. All layer interfaces are bonded. So, we give this bonded as 1, unbonded as 0. So, you know we are going to look into the bonded layer. So, it is 1. Number of layers for bottom tension and top compression-we will revisit this when we do the damage analysis. So, as of now we will keep it as a default number. System of units, let us follow SI system. So, I will give this 1. So, this is the general input. Once you complete the general input, you will have a status updated here as done.

(Refer Slide Time: 14:38)



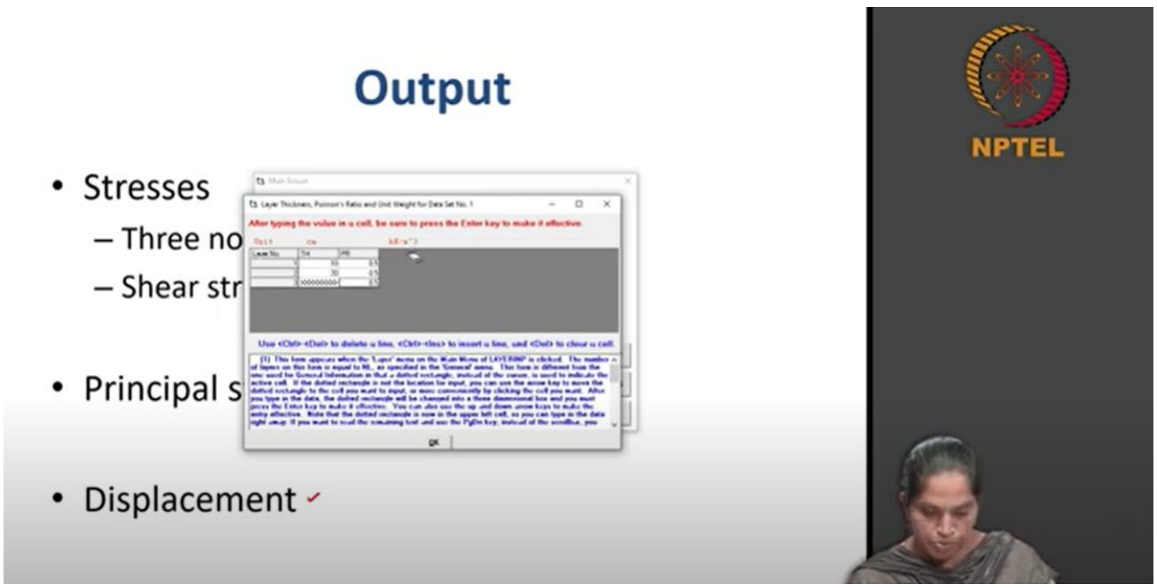
Now, go to z coordinate. So, you see, we have given two depths, two points where we need to determine stresses and strain. So, you have two points here. You can give at what exact depth you need the stress and strain. Let me assume, let me give some number, but it has to be given in centimeters. So, I will assume the first layer thickness to be 100 mm. So, I give 10 centimeters and I will assume the second layer thickness to be 20 centimeters. So, I will give 10 plus 20 that is 30 centimeters.

(Refer Slide Time: 15:19)



So, once done, next is layer input. So, in the layer input, you have Poisson's ratio and thickness of different layers. We assumed a 3-layer structure, we need to give the layer input for 2 layers and third layer that is a subgrade layer is considered as an infinite layer. So, first, thickness of first layer is 10 centimeters, thickness of second layer, we will take it as 20 centimeters. So, you can define the Poisson's ratio here. So, let me give a Poisson's ratio value of 0.5 for each layer. So, asphalt layer you can take it as 0.35 and soil layer somewhere from 0.35 to 0.5. You see there is also another column hidden here kilo Newton per meter cube, which defines the density. We do not need this number here that is why it is hidden. We are doing only the linear elastic analysis.

(Refer Slide Time: 16:11)

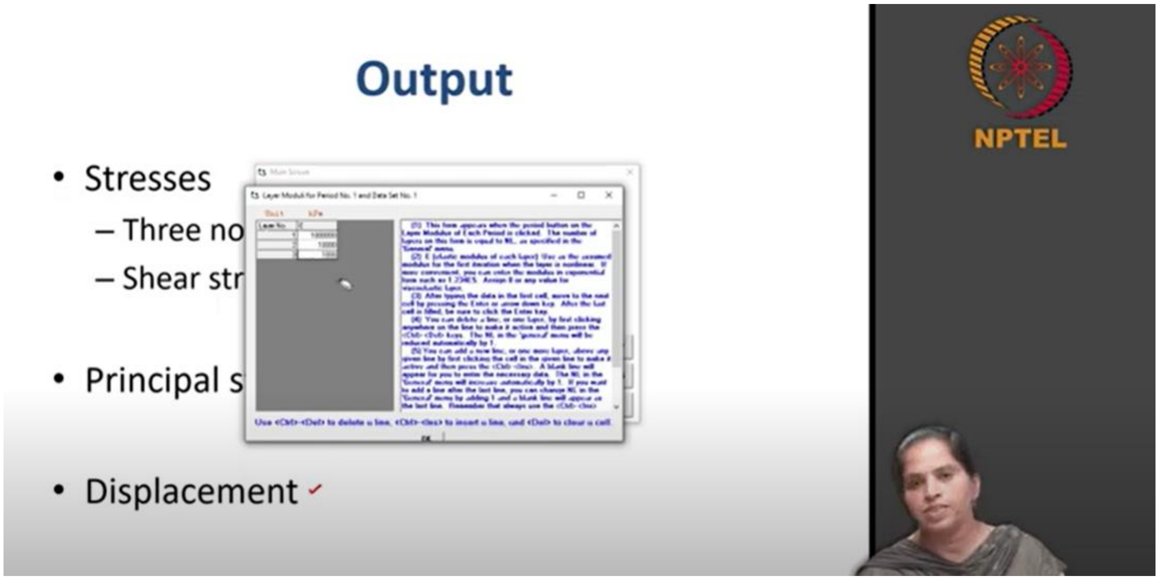


Once we define the layer input, the next one is the interface. So, we have given the interface to be bonded and hence there is nothing to change in this.

Next is the moduli input. So, when you go to moduli input as we have seen before, you can give an input for 12 different periods. So, you have 12 different provisions here. Now, we have given only 1 in that. So, we have only period 1 to be highlighted. So, now we have 3 layers, you can define modulus of 3 layers in terms of kilo Pascal.

So, let me give the modulus of first layer to be 10^6 kilo Pascal, modulus of second layer to be 10^4 kilo Pascal and third layer to be 1000 kilo Pascal. So, I am just taking arbitrary numbers here. So, we will use a specific number when we solve a numerical problem.

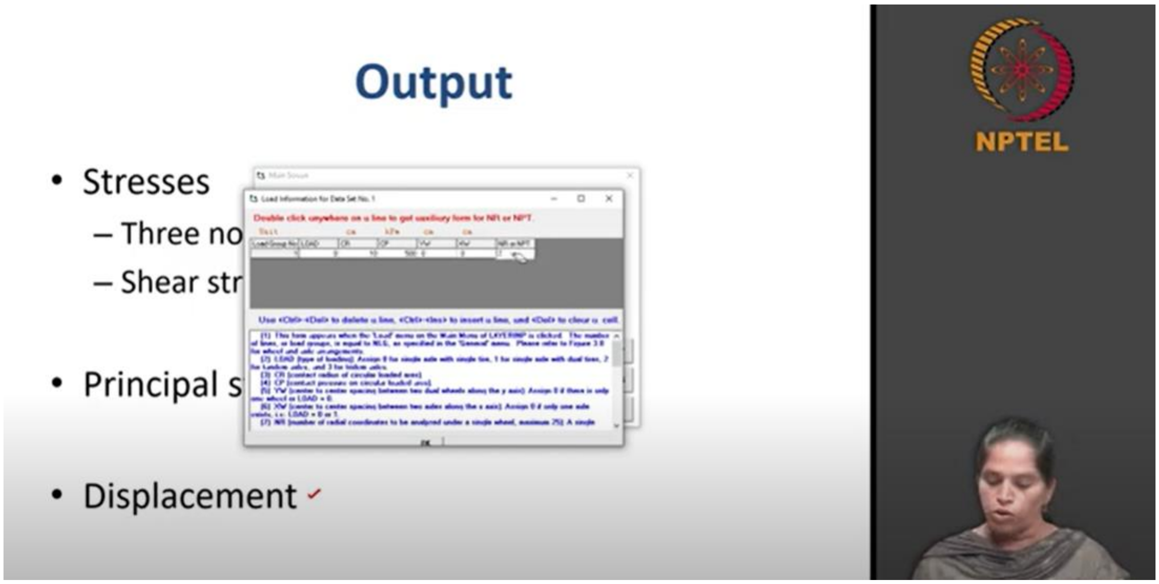
(Refer Slide Time: 17:15)



So, once the modulus input is done, next you can move to the load input. Now, when you open the load input, the first input which you have to look into is axle configurations. Axle configuration is defined in this load column. We assign 0 for defining single axle single wheel, 1 for defining single axle dual wheel, 2 for tandem axle and 3 for tridem axle. Here, let us use 0 for the demo purpose which says that it is a single axle single wheel. You can define the contact radius and let us use 100 millimeter or 10 centimeter here for the demo purpose. I am just picking some arbitrary number here. Contact pressure in kilo Pascal, maybe 500 kilo Pascal. So, YW and XW are distance that defines the distance between 2 wheels and distance between 2 axles. Since it is only 1 wheel here as of now, for 0 category of load, this YW and XW are 0. NR or NPT is the number of radial distance or lateral points where you need to determine stresses and strain. So, if I define this to be 2, when you click on this, you will have 2 points to define. So, 0 is exactly at the center of loading. You can give any number of points here. Now, let us give 2 to be on the periphery.

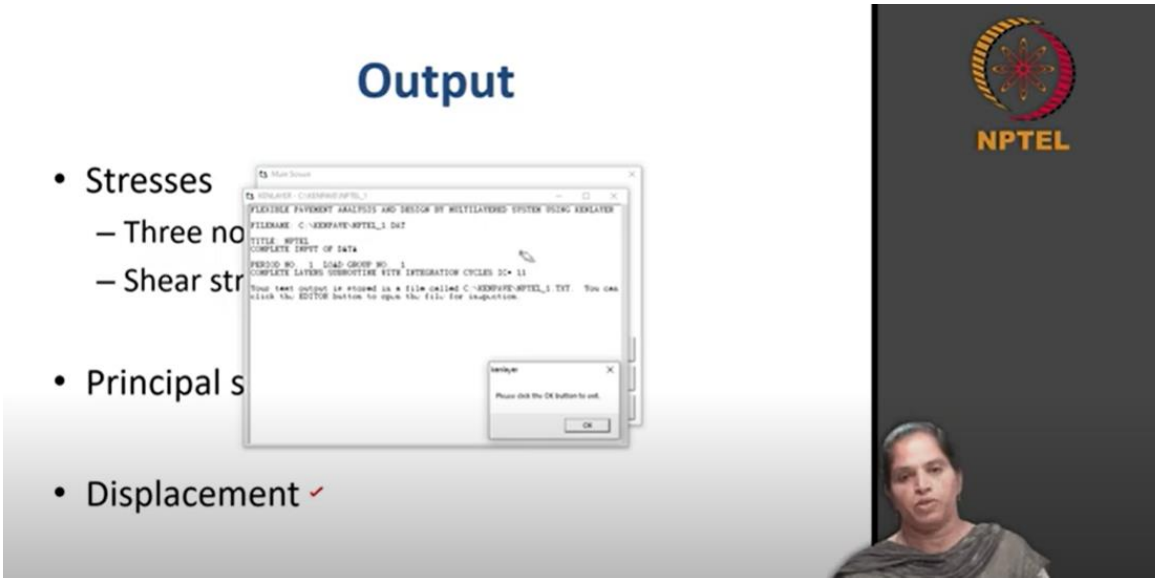
So, we have taken a radius value to be 10 centimeters. So, I will give the second point to be 10 centimeters and once we have done, so our load input is done.

(Refer Slide Time: 18:44)



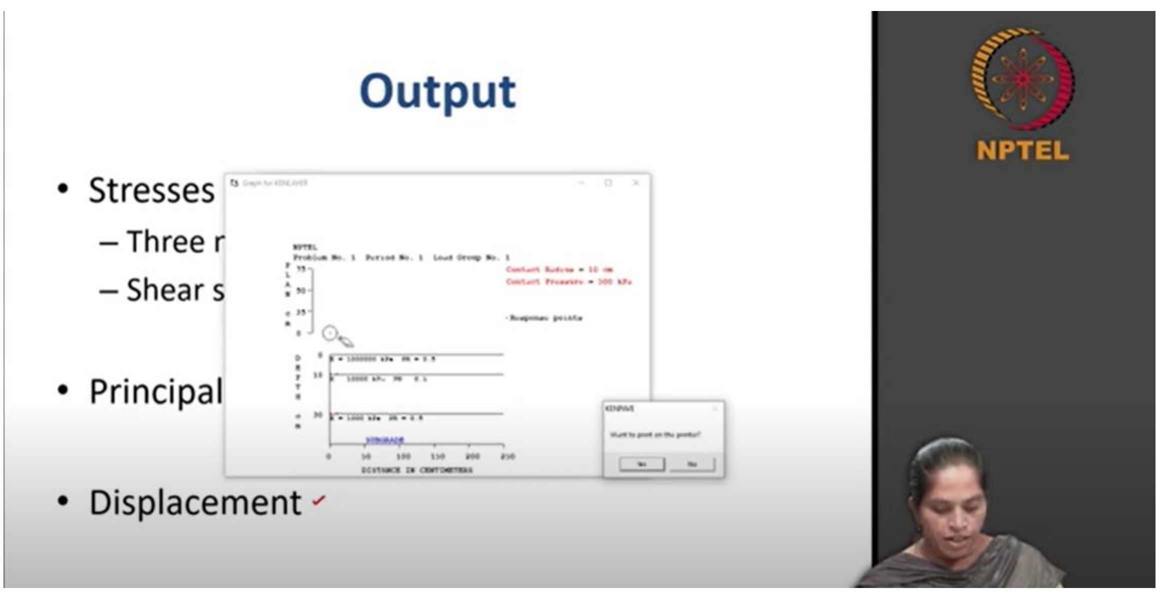
Nonlinear viscoelastic analysis and damage analysis are something we are not going to do. So, by default, it is hidden and we cannot give any input here. So, once this is done, you can just save it using ‘save as’. So, I will save it in the name ‘NPTEL’. Now, you can see that the file name is replaced to NPTEL_1.

(Refer Slide Time: 20:08)



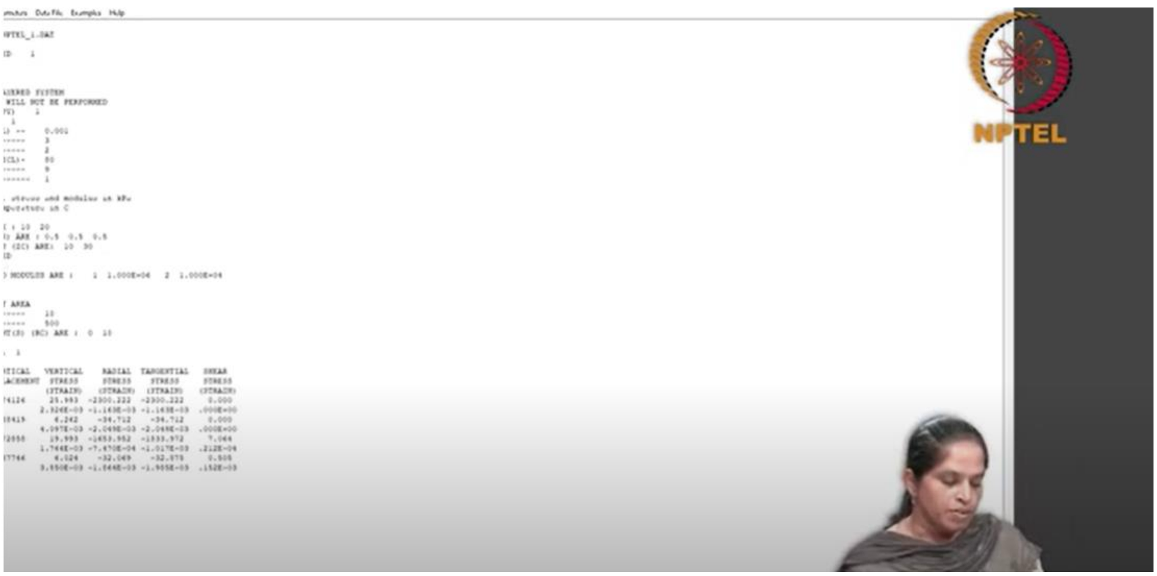
So, once we complete all the input, you can come out of it and you can compile it using this KENLAYER option. So, this is what I was saying. So, if you get this kind of an information, it means the program is completed and you can see the complete layer routines with integration cycles, it took 11 number of cycles. So, we gave 80, 80 is the default value and we kept it to be the same. It took 11 cycles for the results to get converged for a given number of tolerance value.

(Refer Slide Time: 20:40)



So, once you have done, you can just go to layer L graph and you can see the input in terms of a graphical representation. The top sketches for the plan and the bottom sketches for the section. So, you can see we have given single axle single wheel, the center point is exactly at 0. This has a contact radius of 10 centimeters and contact pressure of 500 kilo Pascal. We have defined 3-layer system. The thickness of different layer, modulus value and Poisson’s ratio value used for different layers are given in this. This is a diagrammatic representation of our input. So, once you compile it, you can see that result through editor option.

(Refer Slide Time: 21:30)



So, you can go open ‘file’, the file will be saved in .txt format. This is the file which we have given. When I open this editor file, first you will see all the inputs whatever you have given and the results vertical displacement, stress and strain will be given for any given radial distance and for any given vertical coordinates. So, in stresses and strain value, you have vertical stress, radial stress, tangential stress and shear stress. So, at the point where the radial distance is 0, the shear stress is 0, and on this periphery, you have a small amount of shear stress. So, you can identify which stress is critical at which locations from this table.