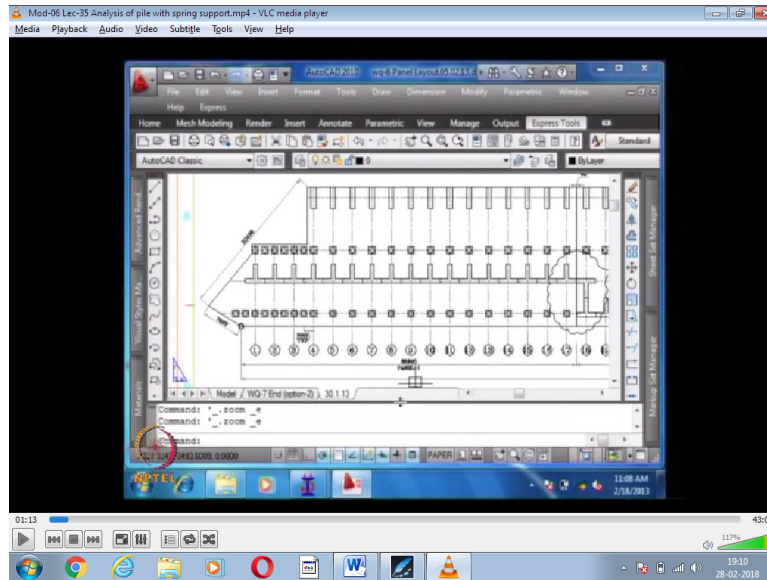


Port and Harbour Structures
Professor R. Sundaravadivelu
Department Of Ocean Engineering
Indian Institute of Technology Madras
Mod-06
Lec-35

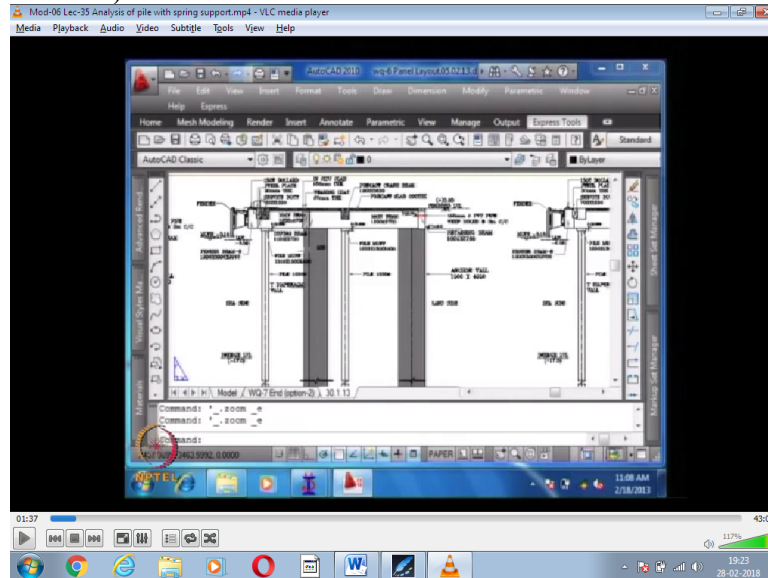
Analysis of pile with spring support

(Refer Slide Time: 00:32)



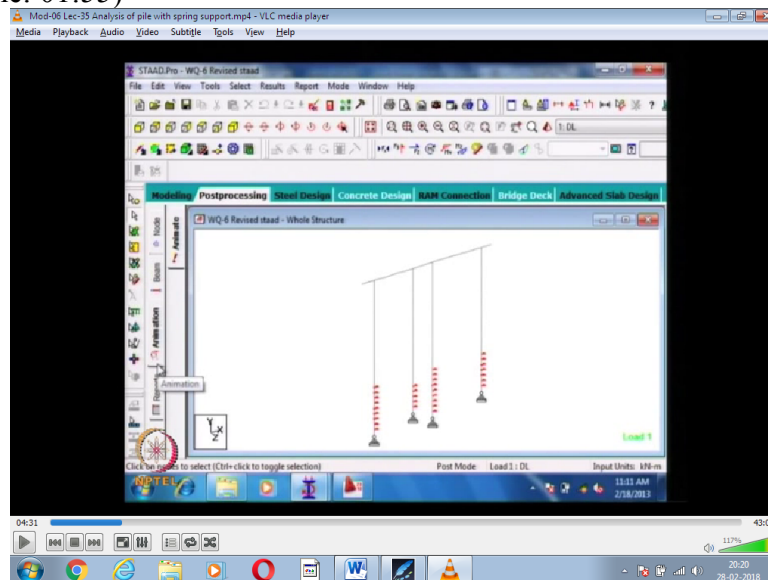
In this class we will see the start pro analysis a two D analysis of the structure which we discuss in the last class so the structure will consist of diaphragm wall, a T diaphragm wall which is shown here than there will be a pile here in the front, this is a side view then will have another pile then an anchor wall so these piles are spaced at four meters centre to centre anchor walls also at four meters the front row pile also at four meter centre to centre the T diaphragm wall width will be four meter and this is about three point six meter so will have front row pile, T diaphragm wall, middle row pile, and the anchor wall. The spacing between this pile and this pile twelve meter where the crane will be operating now we will see the cross section.

(Refer Slide Time: 01:19)



So in the cross section if you see this will be the front row pile, this will be the middle row pile, this will be the main diaphragm wall, this will be the anchor wall. You have the crane beam on top of this another crane beam on top of this, you have a retaining beam and then we have a fender panel here with a fender beam there will be a service stretch at this location. We will see the discretization by start pro final element analysis we will see the diaphragm wall, pile, anchor walls, module by beam element.

(Refer Slide Time: 01:55)



As you see here, we are using two types of element this is a beam element for the front row pile, this is a beam element for fine diaphragm wall, this will be the beam element for the middle row pile, this will be the beam element for the anchor wall and this is a beam elements for the beams connecting this you taking only two D analysis and in this we will be

seeing only the line diaphragm that means we take a typical span of four meter and then we are idealising this.

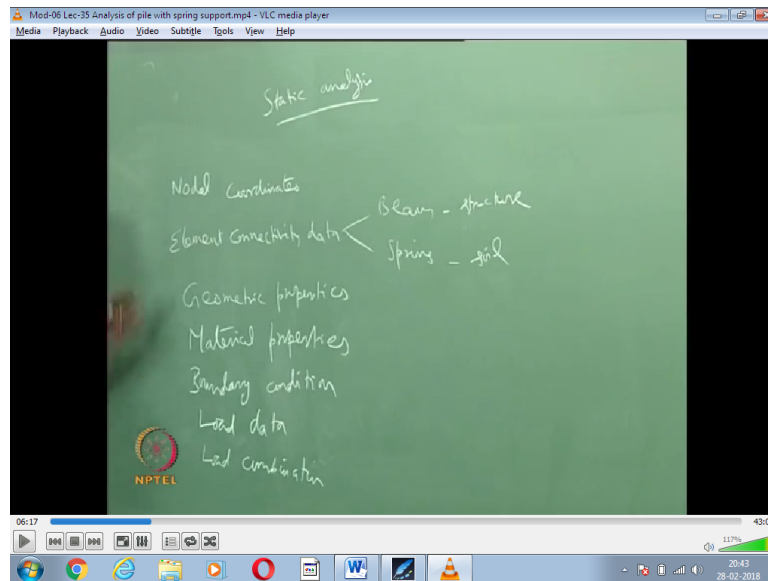
When you are go for three D rendering you will be seeing how looks like, before that we have to see the soil support up to minus seventeen meter here the soil is soft clay it will not offer any resistance latest highest cost says if the N value is less than one or two do not add do not take the soil support from the soft clay layer, so soft clay layer is up to minus seventeen so we have not modal the clay up to this point below that we have modal the clay by assuming a spring element.

This spring element is a discrete springs it is not a continued springs the spacing between the springs is normally taken as about one meter centre to centre. Typically we take one meter but generally the spacing should not be more than the thickness of the diaphragm mall or diameter of the pile in this case the diameter of the pile is one meter, thickness of the diaphragm mall is eight hundred so we are keeping very close to one meter. There is a procedure to calculate the spring constant that will discuss in another class, so we idealise the soil support.

Here these piles are connected with the spring, main diaphragm mall also connected with spring, the middle row pile also connected by spring and the last anchor wall that is also connected by spring than we have a boundary condition that is a roller boundary condition is given at the bottom to idealise the vertical load that is transfer directly below the pile or diaphragm mall and the horizontal support is given by anchor wall.

This is a plain frame analysis only two D idealisation; in this case there will be any load perpendicular to this plain of the frame loads will be in this directions, different loads are dead loads, live loads, berthing force mooring force etcetera, now we will see the three D rendering.

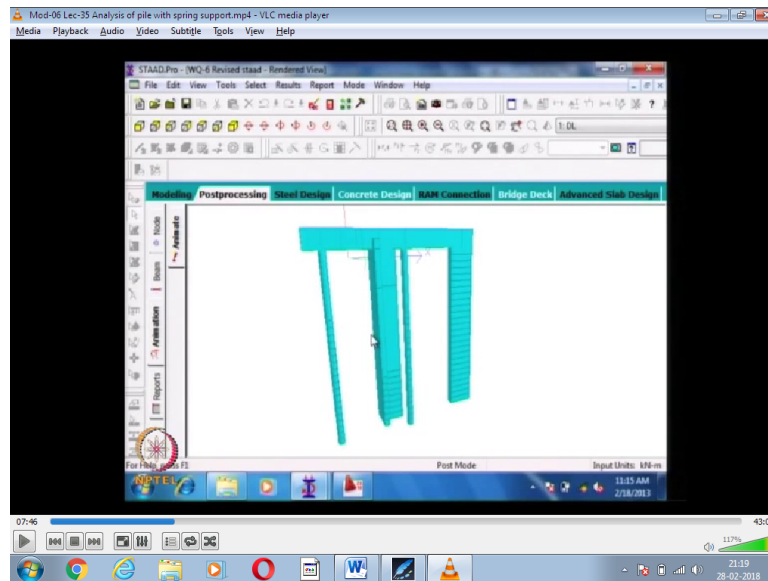
(Refer Slide Time: 04:34)



You routed this figure so what we will be doing is we will be generating this by giving this inputs systems one is node coordinate next is the element connectivity data here will be giving two elements one is a beam element another is a spring element, the beam element is to idealise the structure the spring element to idealise the soil then will give the geometric properties, material properties than will give the boundary condition, so boundary condition is very important if you do not give the boundary condition your results will not be proper then you have to give load data then load combinations.

So we are now doing only static analysis but the program can be use to do a dynamic analysis also, so these are the input data's which you have to give in sequence so this shows the three D rendering of the structure as you see here.

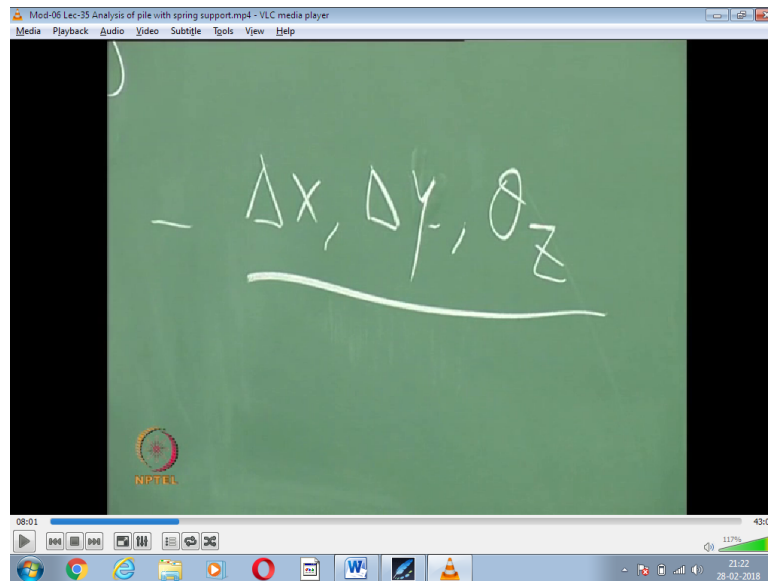
(Refer Slide Time: 06:29)



As you see here the piles are idealise by geometry property which is a circular cross section giving the movement of inertia and area, this is a T diaphragm mall this also have to give the geometry properties calculated for T section, this is for middle row pile this also circular cross section, this is anchor wall which is rectangular cross section. For these sections you have to correspondingly you have to give the geometric properties or give the sections dimensions then it will automatically calculate the properties.

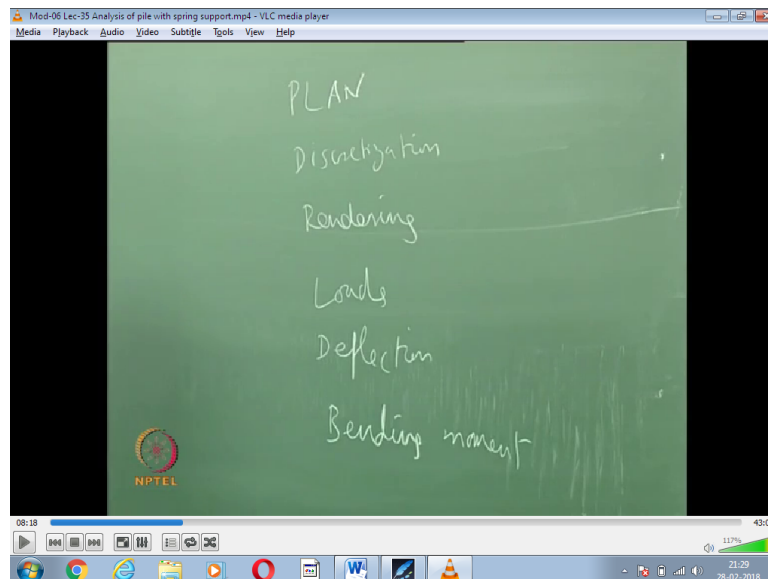
The material properties are Young's modules and Poisson's ratio; here it is concrete that can be given. This is the beam which is connecting the all the structures of the top the boundary condition are applied at the founding level not allowing the structure to move in the (()) (07:25) direction. So this coordinate this is X axis, what is this one, this is the Y axis and perpendicular to the plain of the board is Z axis, so what will be having is, will be having the boundary conditions.

(Refer Slide Time: 07:48)



Delta X, delta Y and thetaEz that means displacement in the delta X, X and Y direction and rotation about the Z axis for a plain crane two displacement and one rotation this is a boundary condition.

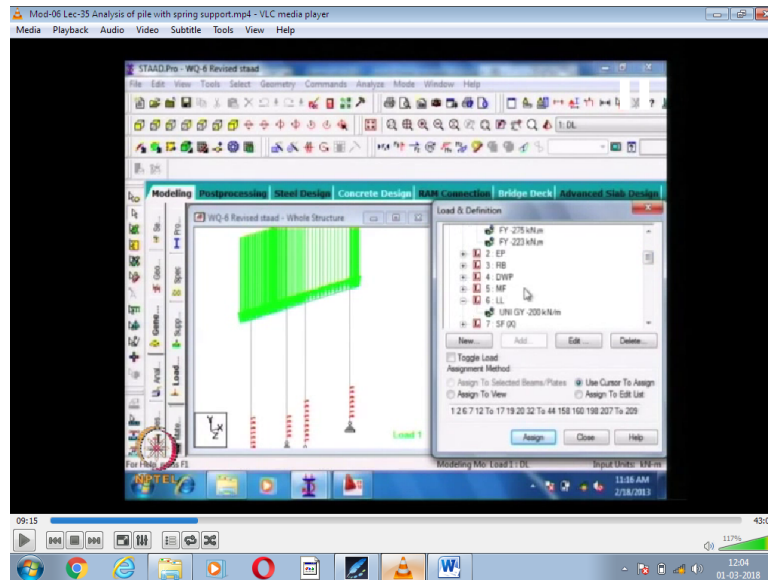
(Refer Slide Time: 08:10)



Next we will see the so here what we are doing is initially given the plan then give the discretization now what you seeing is rendering now will apply the loads and see how the loads are applied for each case separately the loads that will be applied are dead load live load berthing force mooring force active earth pressure differential water pressure seismic

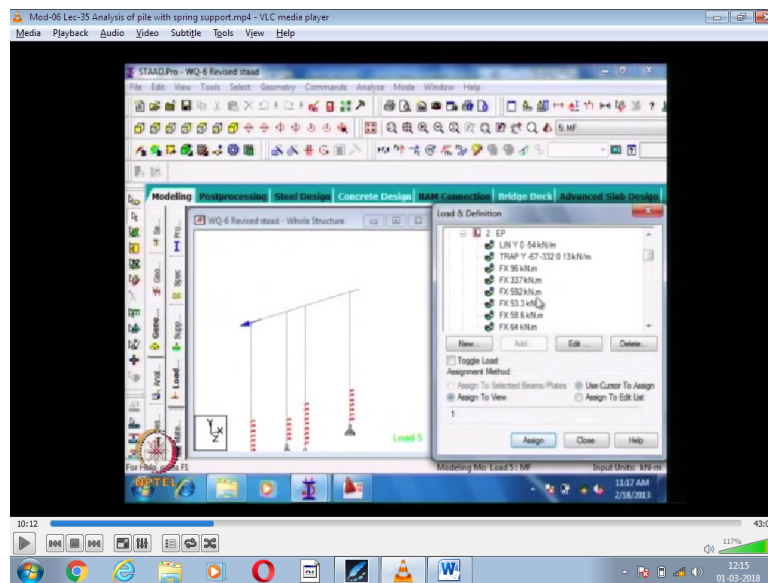
force, these are the typical forces these load will be applied individually and you can get the results individually also for analysis then we will be doing it for combinations also.

(Refer Slide Time: 08:52)



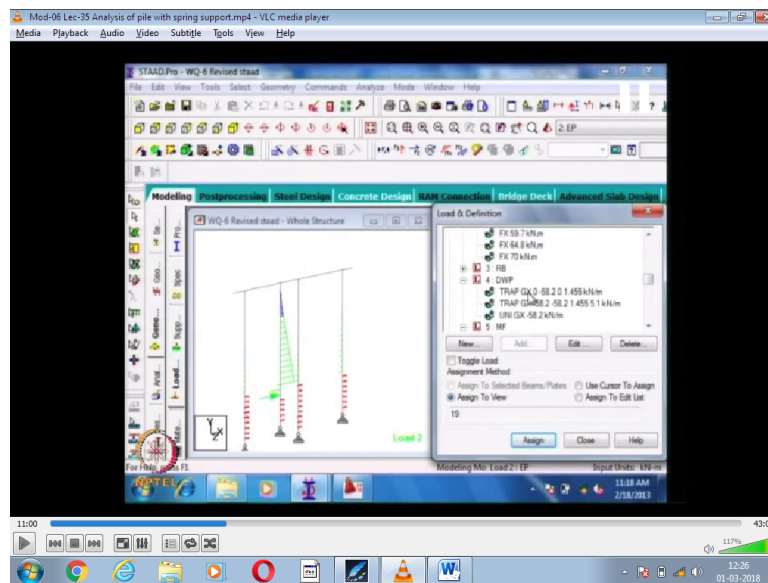
Now we will see the loads one by one how it is applied, so now first we will see the dead load as you see here dead load is applied on the beam this is a weight of the beam plus the weight of the slab which is for a distance of four meter that is two meter on other side that will be applied a uniformly distributed load on the structure, next we will see the live load.

(Refer Slide Time: 09:17)



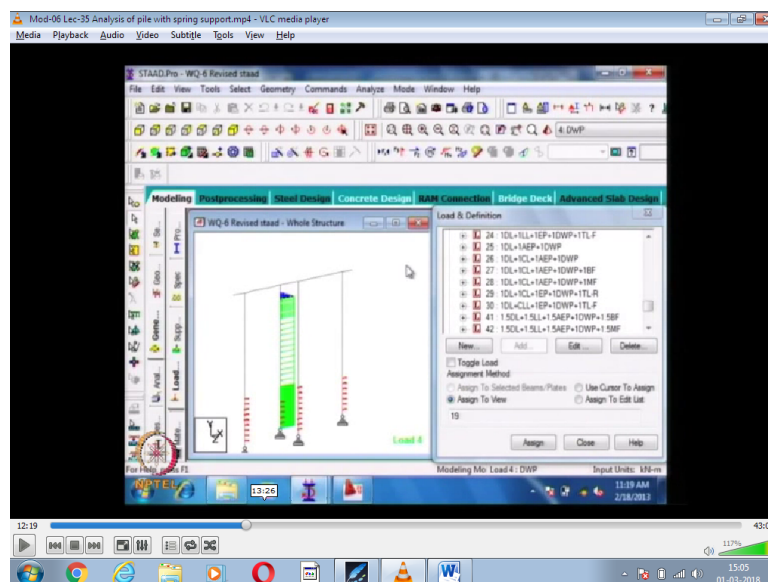
There are different types of live load one is uniformly distributed live load berthing than we may have the crane load also crane loads are applied as points loads which are applied in this direction the crane loads can have (())(09:36) in the lateral direction also than we will see the berthing force, berthing force is when the ship is coming and getting is a berthing force then mooring force, this is a mooring force this is one of the critical combination is the mooring force along with the active earth pressure and differential water pressure we will see the active earth pressure.

(Refer Slide Time: 10:15)



So this is a diaphragm wall main diaphragm wall which will be subjected to the active earth pressure that is shown here then we will also have a we remove some springs here when you do the analysis that why we out the passive earth pressure in the opposite direction because there will some plastic deformation of the soil so we cannot put a spring with in connect stiffness with the stiffness with infinite displacement so we are replacing the spring with a equal force on the opposite side. Then differential water pressure.

(Refer Slide Time: 11:01)

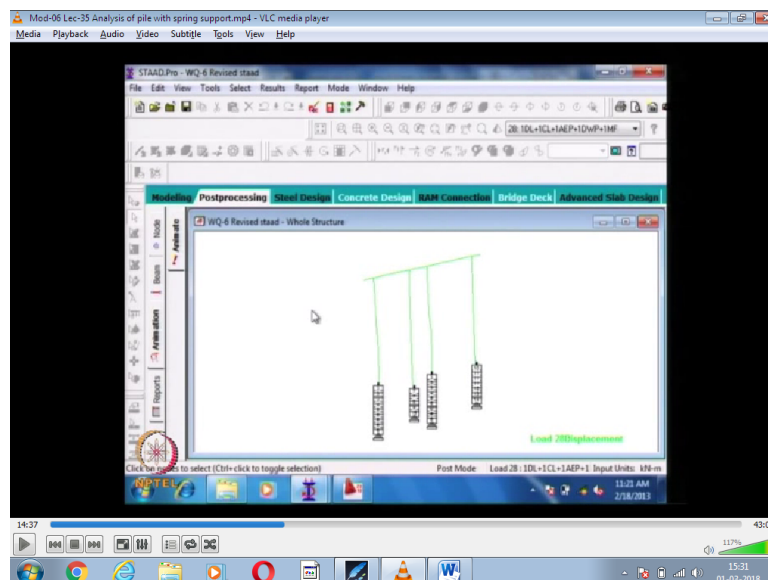


So this is a differential water pressure, the water level here on the land side and water level at a lower level on the sea side this is a differential water pressure, applied up to the bottom. Once you apply the loads and perform the analysis we have to see certain results the results are deflections binding

movement and axial force on the structure, share force on the structure, these are the results which you have to see we will see one by one further that is the deflection, bending movement, axial force and share force also these are the results once you perform the analysis will be getting these results deflection, share force, bending movement the axial force as you see in the slide if you see the load combination number twenty six.

We see the load combination number twenty six so this is a dead load, crane load, active earth pressure and differential water pressure with a load factor of one this is for a serviceability limit state condition. You can also do it with berthing force or mooring force this load combination number twenty eight will most probably critical we will see the results for load combination number twenty eight.

(Refer Slide Time: 12:34)



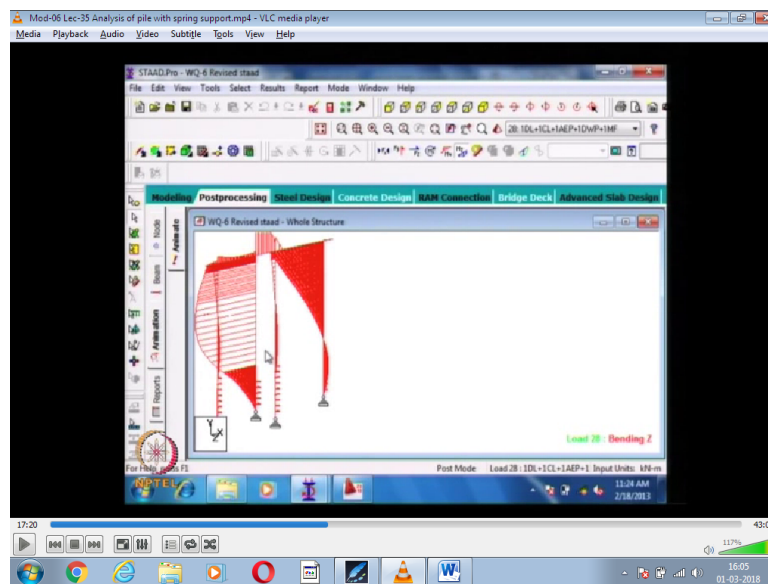
When we do this load combination number twenty eight for this deflection we have to see that the deflection is less than about hundred mili meters we will see the animation of this just you given idea hoe the total frame deflects, so the values are written there the maximum deflect is at the top which is about hundred mili meter you have to graphically see the deflection to see whether the deflection is properly obtain or otherwise, so once you give the results then it will be very clear and see the output results you can feel that the results are correct or otherwise, now we will see the animation of the deflections.

Animation is the deflections will be some time frame as a load is applied how the structure is deforming then the load is removed the structure will come back to the original position. You have to proper skyline than only you can see the deflections, are you able to see the structure or sanity. See here the top of the structure most were this side then comes back to rest, the soil is not allowing the structure to deform below this level.

If you see the diaphragm mall very carefully diaphragm mall deforms like this you go to the deflections diagram. In large, as you see here the top deflection is more here and as you see here the deflection is not reducing it is going to zero only here that is for the main wall main diaphragm mall, here the deflection is zero at much higher level for the piles both for the main diaphragm mall as well as the anchor wall the deflection is zero at much lower depth whereas here it is much higher depth.

They something like free earth support this diaphragm malls the founding level is not very high just giving a point support here whereas for the piles it gives a fix it support you will have the deflection going on the other side also, now we will see the binding movement diagram. What I explain now it will be clear if you see the binding movement diagram.

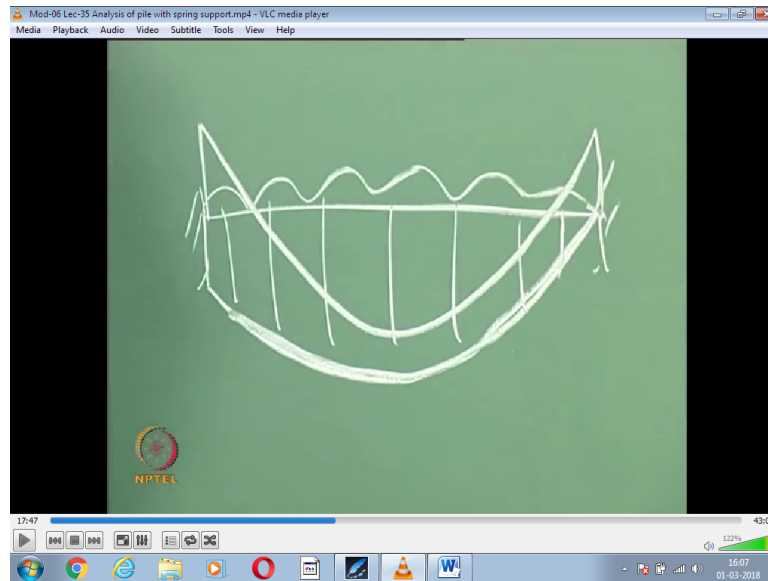
(Refer Slide Time: 16:14)



This is a higher scale now we will reduce the scale so the results will be very clear leave it as it is. See as you see here this is for main diaphragm mall the binding movement is very high this is of the order of the thirty thousand kilo newton meter as you see here that binding movement is thirty thousand it may increase to thirty five thousand here than it comes back and become zero here, for the anchor wall it may be about thirty three thousand here, than it goes to the negative side then become zero, whereas this is only on the positive side only one direction.

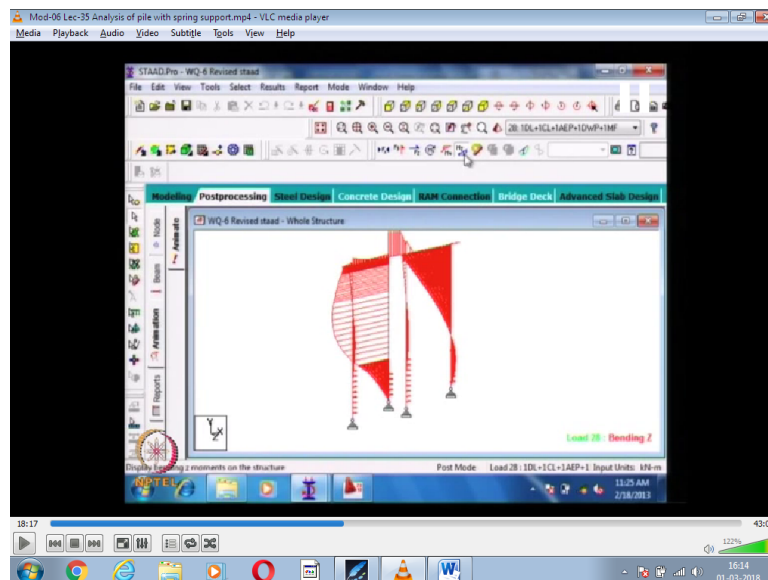
Where is the maximum movement for the main diaphragm mall that is at this level because this subjected to here uniformly distributed earth pressure there is a load acting on this. Anchor wall there is no load acting on the length of the anchor wall the load is acting only on the main wall once you have the load acting on this the binding movement goes like this, there binding movement does not go to the other side it goes only to one side since there is a load here acting on this see something like this.

(Refer Slide Time: 17:23)



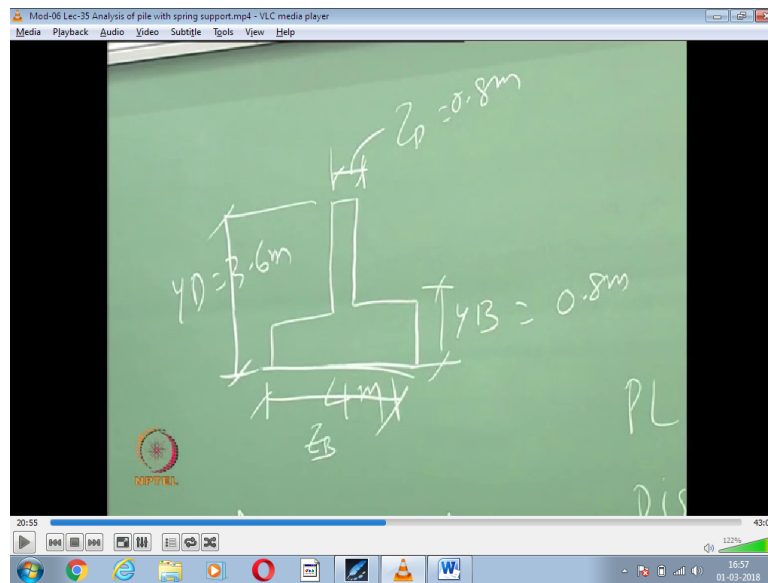
Suppose you have a beam like this subjected to UDL you will have bending movement diagram like this all of you know for a fixist beam the bending movement diagram is like this but if the support seeing the bending movement goes like this so this is what you are so this what you are seeing for the main wall, main anchor wall you have the bending movement diagram only on the one side only.

(Refer Slide Time: 18:00)



So you can see the beam bending movement also if you take a joint a particular joint, the joint whatever the movement in the diaphragm wall on the beam on this side, on this side if you sum up all the movements the net movement will be zero that's what you have to check once again. Next we will see the axial force.

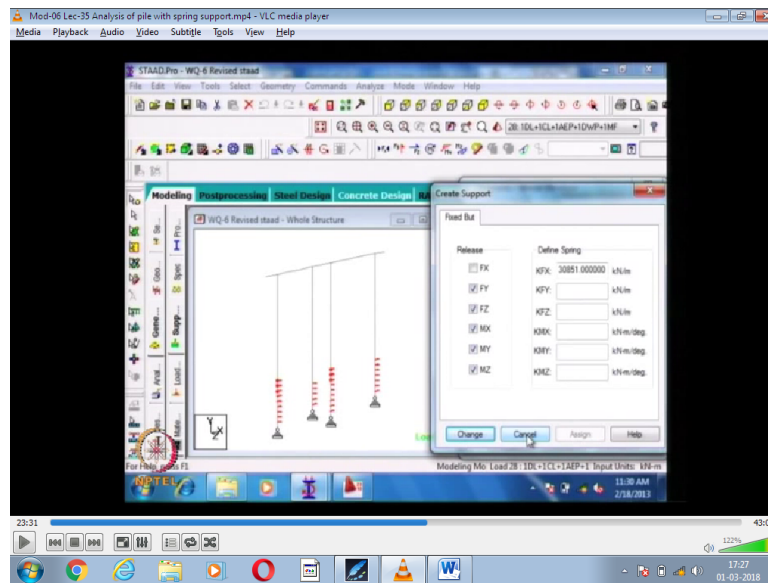
(Refer Slide Time: 19:18)



We will first see what are all the properties of this element particular element you go to the pile this is the pile you just this pile is this is the pile R one is the pile what you are seeing here we have given circular section of one meter than we will see R three you see here this is a cross section of the pile I am sorry diaphragm wall main wall (1944) depth YD is about three point six meter and this ZB actually this structure is slightly different what shown here is like this but what we have modelled is the other way around like this we have model you do not see that figure where it will be little bit confusing so this section is correct this value is YD this YD is three point six meter this is also correct what is written here is ZD is point eight meters next is ZB next is YB this is your YB water plant is point eight meter this is four meter because our T section is ultra.

This figure and that figure what is shown as to be redrawn like this so this is your ZB which is four meters then we will go to the anchor wall.

(Refer Slide Time: 21:11)



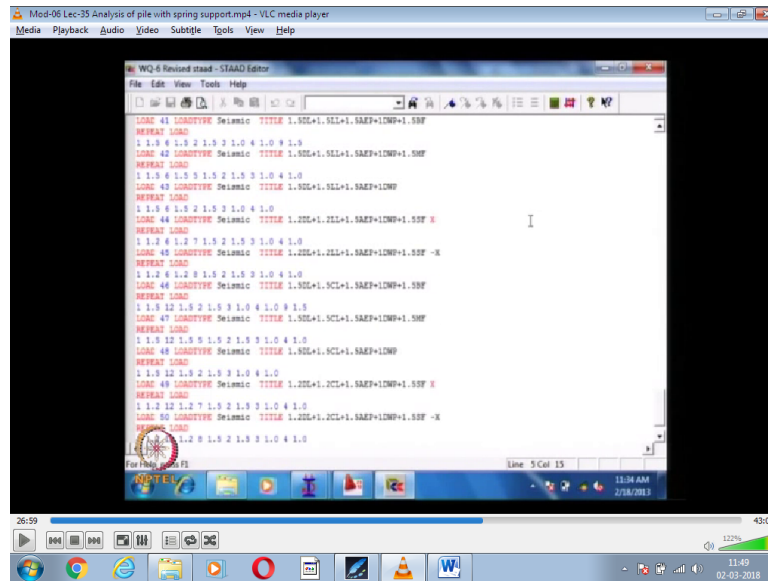
This anchor wall this dimensions may not be proportional because this is given here so YD is about four meters that is the depth of the anchor wall and ZD is one meter go to the section now there is one more section circular section of one meter this is also for concrete there are four sections that are given sections (())(21:21) is for one meter that is the middle row pile one meter dia meter pile then we have the rectangular section two point seven by one point two meter there are five sections this two point seven by one point two meter is nothing but the beam element what is shown here.

Then we have a prismatic T section that is R three that is for the main diaphragm mall then we have a rectangular anchor wall that is four meter by one meter then we have a circular section for the pile that is one meter in dia meter. Now we will see the material properties, so material properties is a concrete. That is also concrete for which it will take a default members whatever is a value you have you can stain less still, aluminium or concrete now we have chosen concrete default value is to be given there.

Now we will see the boundary conditions, how the boundary conditions are applied so one by one we will see, what are the boundary conditions, we will just see these are the boundary conditions, this is the first boundary condition so her ewe are giving some spring constant values that is X direction in this direction it is allowed to displace but there is a spring stiffness here.

In the Y direction displacement is zero, EZ MX MY since it is a plain frame these value will be zero that is why you given a FZ MX MY is zero they also allow MZ to be not zero ok next level. I will explain in another class. So you have to apply the boundary conditions, then you have to perform the analysis now we will go to the input data pile where all these details are given in a text format.

(Refer Slide Time: 23:50)



The first line is start pile it gives some joint name then the date of the thing is given then what is the input width is given, the units for which the analysis is to be carried out is given this is meter and kilo newton just enlarge the size control and enlarge control screen is not working. Ok, then we have to give the joint coordinates so here you have to give units in meters and kilo newton that is the length unit should be given in meter and the force unit to be given in kilo newton.

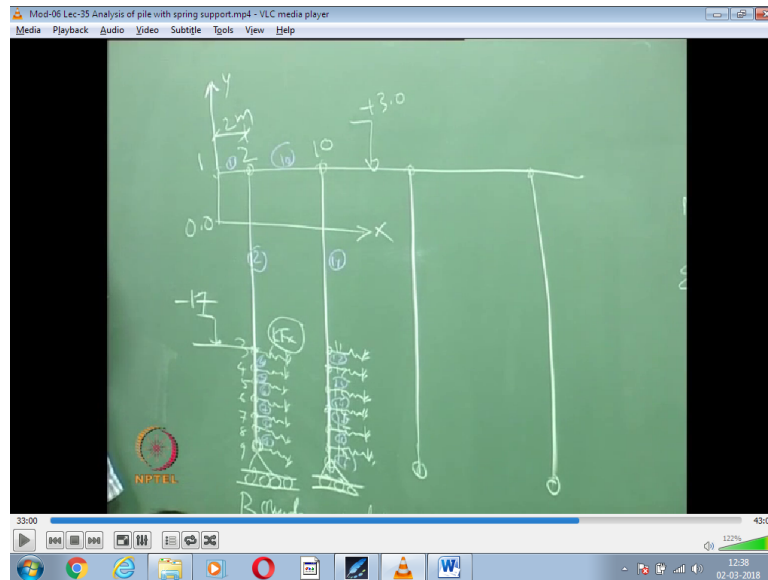
If it is Young's Modulus you have to give it in kilo newton per meter square that you have to strictly flow movement of inertia you have to give it in meter power fur you should not change the units. Then there are the joints for which you gave the XY and Z coordinate so Z coordinate is zero for all the joints X and Y coordinates are written here then we have to give a member incidences that is member connectivity between which note it is connected then you have to give the material properties so we are giving isotropic concrete we give the Young's Modulus Poisson's Ratio Density Alfa is a temperature expansion coefficient Damping this is use for dynamic analysis.

Then we have to give the member property you have to assign it to each member how it is there then we have to give the constants that is the supports we have to give we are giving all the supports as pinned and some supports we are giving fix but releasing some certain degrees of freedom so then we do to the loads how the loads are applied, so we are applying the sulphide so we are applying the dead load live load joint load that may be for berthing force or mooring force, go down we give a linear variation at saddle variation so these are the various loads that are applied then we have to give the load combination.

So each load we have to give some combinations if you see load forty one load type seismic we give the load combination one point five dead load, one point five dead load, one point five active earth pressure, one differential water pressure one point five berthing force then go to the end of the text file

then we have to do the analysis then print all, then finish so this are the so you can either graphically input all the model points or you can type it also but generally now a days it is performed in graphical things only.

(Refer Slide Time: 27:06)



Now I will summaries what we are giving so we have a beam connecting the piles diaphragm wall another row pile, another row pile, here these are the loads which are used to define the points for various elements these coordinates you have to give so you have to first define the axis you can define the axis like this this is your X axis this is your Y axis wherever that chart data level is there you can take it as zero point zero so centre line of the beam you have to model then you have to give the nodal coordinate for each points that you can give suppose you are defining by different springs that springs are one meter centre to centre you have to model the spring elements here for each pile suppose you join this one two three four five six seven eight nine nodels here we have to give a roller support and give the boundary conditions.

This is the place where we give the boundary conditions; boundary condition means at this particular point this delta Y deflection in the Y direction is zero that is a boundary condition what we are giving. At this particular point what we are giving is we are giving some stiffness for the pile there is a procedure to calculate the KFX you seeing the equation called as basic equation based on which you can calculate the spring stiffness will discuss in some other class about the spring stiffness you can connect one and two by element number one this can be element number two, two to three, then this can be element number four five six seven eight nine then you can continue here nine is over then you can ten then you will have a one element here this is to idealise the soil support then we have the boundary condition here.

Then eleven twelve thirteen fourteen fifteen sixteen then you can have the spring here connecting this elements, spring elements use to be define separately numbering on the type should be define separately then you have up to nine then this can be the tenth element this can be the eleventh element this is twelve this is thirteen this is fourteen this is fifteen sixteen seventeen.

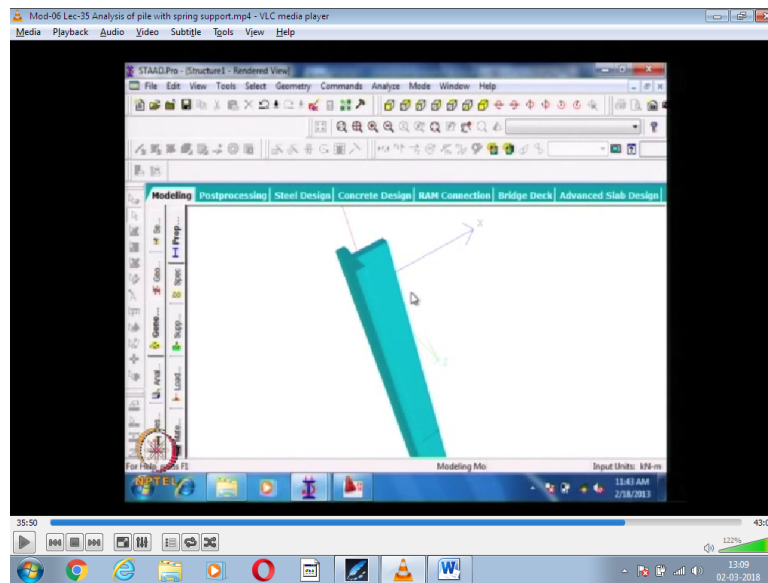
The nodel coordinates when you want to see nodel number one the X coordinate is zero point zero suppose the centre line of the bema is plus three the Y coordinate is three point zero Z coordinate is zero point zero point is it clear. X is zero Y is three point zero and Z is zero point zero so the perpendicular to the plain of the broad is a Z axis when you go to the nodel number two suppose this distance is two meters nodel number two will be X coordinate will two point zero Y coordinate will be zero three point zero and Z coordinate will be zero point zero.

This is your dirige level which is minus seventeen in that case nodel coordinates three is X coordinate will be two point zero Y coordinate will be minus seventeen point zero Z coordinate will be zero point zero. This how we have to give the coordinates, any doubts in this. You can graphically do this also please do graphically all this, when you go to the element connectivity data this element connectivity data two the element number two is connected between which node and which node two and three so what we are giving to the program is how the node numbers are connected.

What is the connectivity data for four, four is between you can give three and four or four and three but it has some implication there is a local coordinate system which it will define. Then we have the geometric properties suppose you take this member number two four five six seven eight nine all these members are circular piles are one meter dia meter so you can define one meter dia meter as a circular member that you can define it as index number one that geometric property you can assign to all this members that is the next one.

Then material property you can give it as concrete, you can give the Young's modulus, Poisson's Ratio, Temperature Coefficient and other things then boundary conditions you have to apply one two three four nodes that is at the founding level you have to apply boundary conditions here also because at this points you are attaching the spring, so once you define all the nodes for the elements all the nodes for all the structures then give the connectivity data for the beam element then you can go for a spring elements you can give the spring element values also.

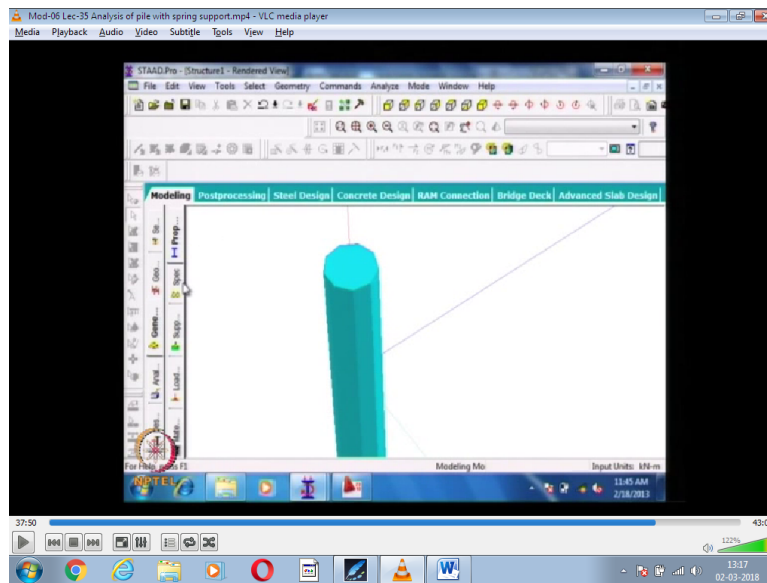
(Refer Slide Time: 33:25)



Now he will be discretising the structure just see how he is doing this the same structure will be discretised, you idealise only one main wall, the geometric properties and material property. The diaphragm wall, you give YD as three point six meter then ZD is four point zero meter YB is two point eight meters ZB is zero point eight meter, concrete give, show the three D rendering so R one is the anchor wall property that it will take, material property so now you can see the three D rendering of the element how the member is looking like so now we will go and change this property like what we have seen here so now the flange is on that side on the right side now we will change it to the left side now we will change YD as three point six like it remind ZD will make it as zero point eight YB as zero point eight ZB as four meters, now we will see the rendering.

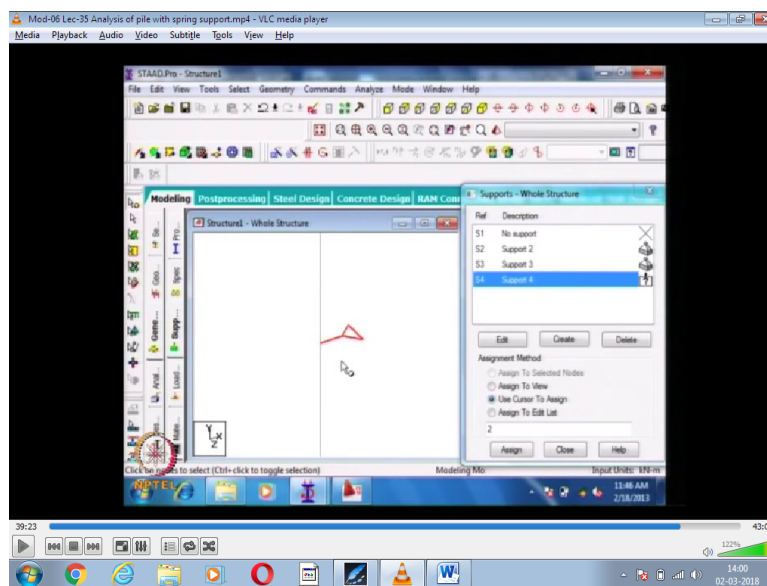
Rotated this so now the flange is on the other side earlier the flange is left side now the flange is on the right side now we will go and see is there change in anchor wall is a rectangular section will change this prismatic to some other member we will change it as rectangular member will make YD as four point zero ZD as one meter so now the depth of the section is along the X axis and thickness of the section in the Z axis this is a correct property of the anchor wall now go and change this X and Z go to the next section suppose you are changing the section this is one meter YD is one meter ZD as four meter so it is in the wrong direction. This is not the way anchor wall is placed so then we can correct this.

(Refer Slide Time: 37:21)



Next will change this into a circular section YD one meter you put, whatever dia meter you want you can put enlarge this it looks like a circular section now we will see how to apply the boundary condition go to the bottom most node, hinged condition pinned you have given.

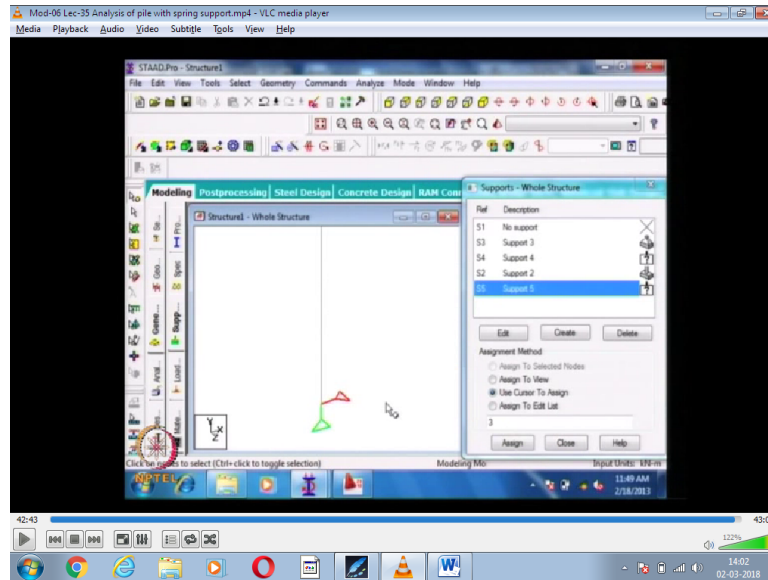
(Refer Slide Time: 37:55)



So now it has gone as pinned condition then will apply a spring slightly above one more node some value will give five thousand go one more node, not that node, you applied some other node one more node was there enlarge the figure now the spring has been applied this direction earlier there is a method by which spring element can be attached as a separate type of element now what we are doing is we are applying as a boundary condition there are two ways a soil idealisation one is to give a

spring element this is there earlier for a sharp and all in a start pro you can give it as a spring element itself suppose the soil stiffness is changing you go to the boundary condition.

(Refer Slide Time: 39:30)



What is one which is given earlier, earlier one you give, no no you have given is three on fixed but you have given earlier you have given spring support two, support two or support three support four I see ok fixed but you give, go to the spring element ok what here you are giving is the F_x we give a spring stiffness KF_x you give five thousand see this value five thousand what we are giving is kilo newton meter five thousand kilo newton per meter wait kilo newton per meter means if there is a one milli meter deflection the load that will be transferred will be five thousand into one by thousand that is five kilo newton this is for a soft soil.

Suppose if there is a rock then we can put one more zero fifty thousand that means fifty thousand means fifty thousand into one milli meter by thousand that will be equal to fifty kilo newton, so you can for different types of soil using basic equation you can calculate the KF_x and apply.

Suppose you put another spring in the Y direction, suppose you create one more value KF_y five thousand KF_y as fifty thousand, assign it at the bottom of node where we given hinch, this will assign to the hinch earlier we assume this is a hinch that will now change it, you go to the support edit that I think you have to give beside I think X and Y only give the value now why you release, X and Y only give five thousand and fifty thousand why the other springs are not coming, F_y give fifty thousand generally stiffness in the vertical direction is much higher than in the lateral direction.

So he now you are seeing that green colour it shows here spring in the, see to summarise you can give the boundary bottom was boundary ever as hinch or as a fixed condition or as a spring ok.