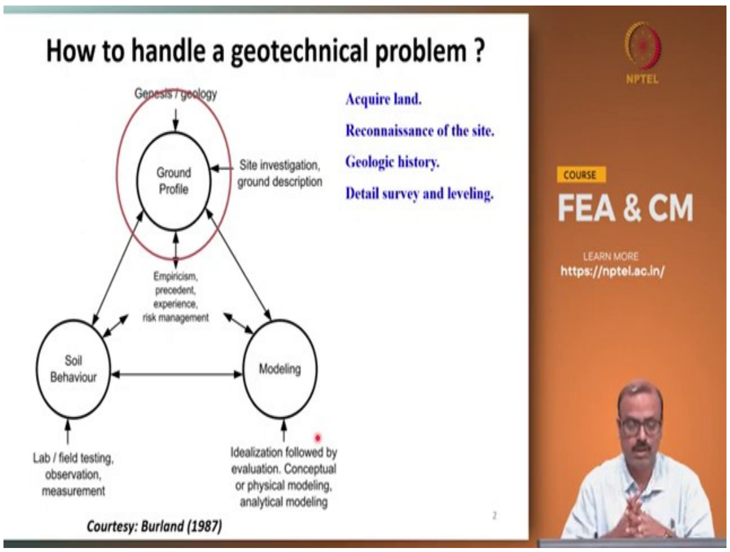
Finite Element Analysis and Constitutive Modelling in Geomechanics
Prof. Subhadeep Banerjee
Department of Civil Engineering
Indian Institute of Technology – Madras

Lecture – 41
FEM in Geotechnical Applications

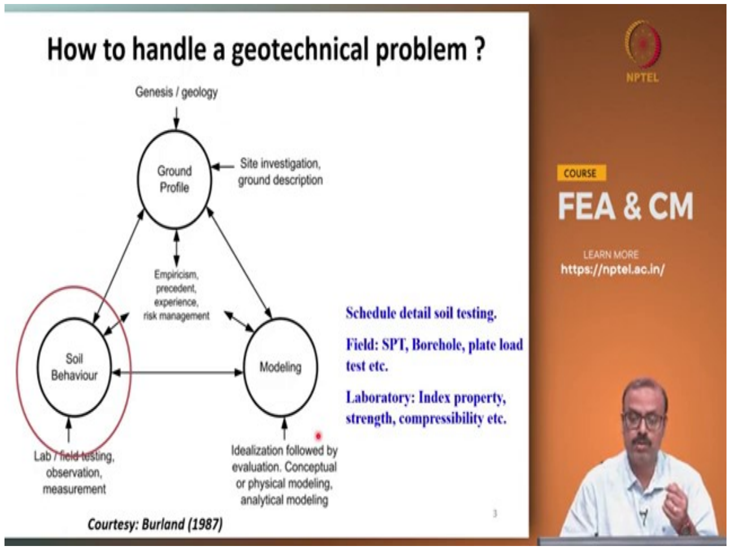
We will have today's lecture in the FEA and Constitutive Modelling in Geomechanics course on FEM in Geotechnical Applications.

(Refer Slide Time: 00:29)



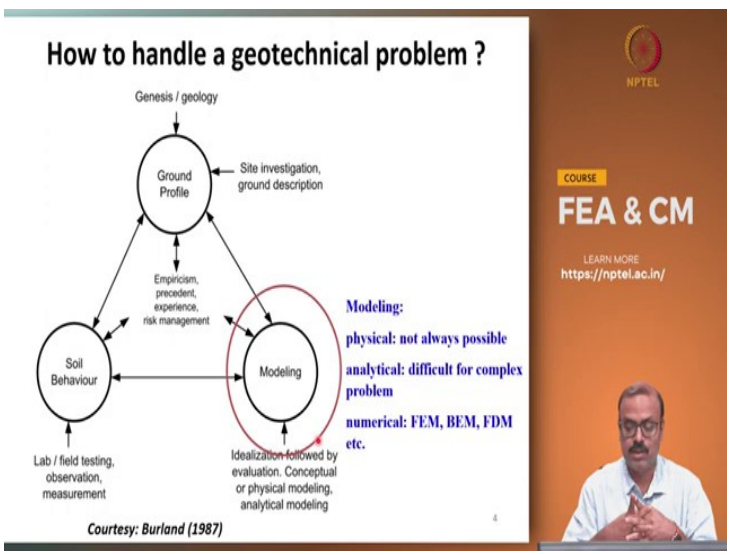
So, first question we have to answer here is how to handle a geotechnical problem before going into the application in FE. So, as you can see, the first one will be to analyse the ground. So, you have to acquire land then reconnaissance of the site you have to understand the geological history, the formations then detailed survey and the levelling. So, overall genesis of the site has to be done.

(Refer Slide Time: 01:02)



Then next one is come that the very important step. How the soil behaviour is happening? Like what should be the ideal schedule for the soil testing? What are the different types of tests you should do field or laboratory in inter spectrum of the soil investigation?

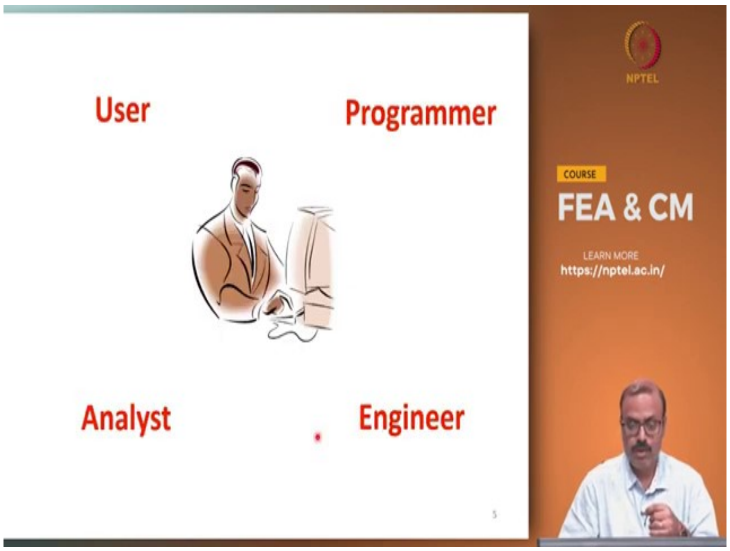
(Refer Slide Time: 01:24)



Then with all this input you have to go to the modelling. Now, modelling can be different types one of course, is the physical modelling which is not always possible because you cannot every time create a model to test it. And before execution then you can do the analytical which is obviously the preferred one. However, the problem is for difficult, complex problem it will be extremely difficult to come up with analytical model.

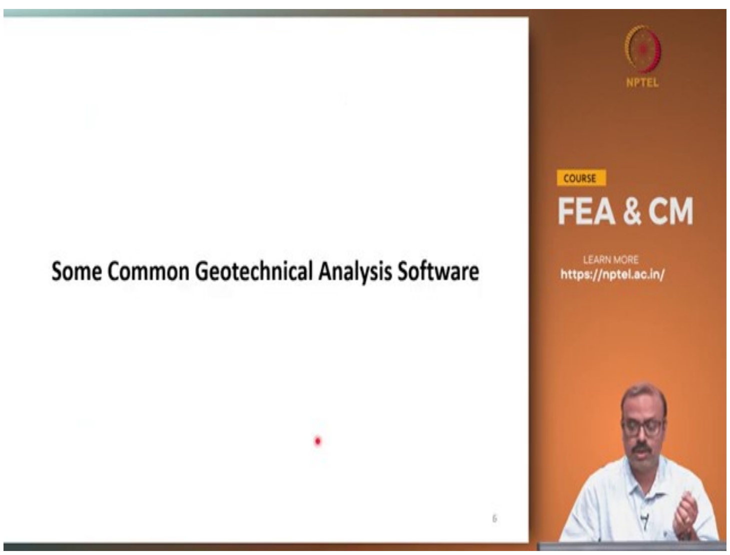
So that is where the comes, the final one the numerical analysis which can be finite element, vending, finite element, boundary element, finite difference and so on. Now, the particularly this numerical tools are very effective in a sense they give may not be the 100 percent accurate. But however, they can handle complex situations and relatively easier and within the time frame given for a particular project.

(Refer Slide Time: 02:30)



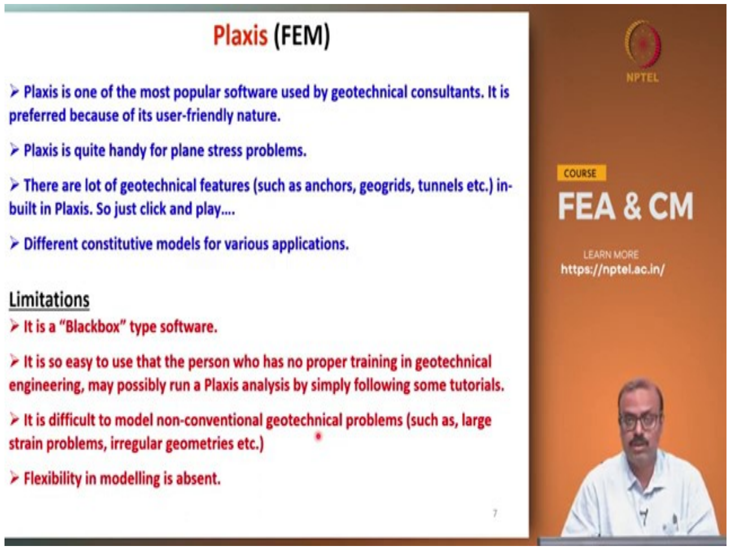
So, there are four players, user, programmer, analyst and the engineer in any of this kind of finiting when modelling.

(Refer Slide Time: 02:42)



So, what are the some common geotechnical analysis software's which we generally see.

(Refer Slide Time: 02:48)



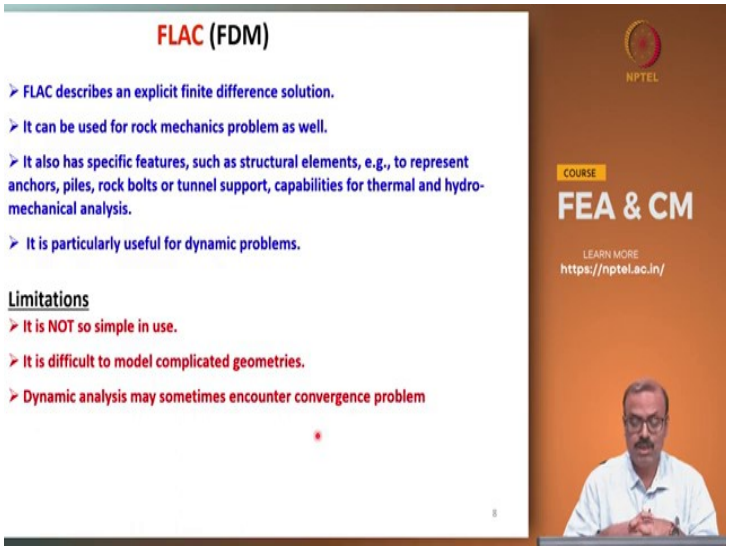
The most popular I mean I am there are many more I am just listing some of the most popular ones. We start with plaxis which is a finite element based software is the most one of the most popular because it is extremely user-friendly. And it has a certain number of geotechnical features like anchors, geogrids, tunnels inbuilt. So, that you can just simply click and play.

There is no it is not a general purpose, it is meant to be the geotechnical applications, so that is why it has various types of constitutive models, for which is suitable for the soil. However, there are certain limitations of plaxis one of the major limitations it is take all so, called the blackbox type software. That means it is so, easy to use that which the person which is not properly trained in geotechnical engineering.

May possibly run a plaxis analysis by simply following some steps. However, he or she may not be able to understand whether the results whether the analysis forms which are coming, are accurate enough to be used in practice. Some of the technical issues related that certainly some of the non-conventional geotechnical problems which we are every day facing nowadays is such as large deformations irregular geometries.

It is little bit of difficult in nature. So, in short, the flexibility which is the essence of this kind of metal modelling is not so much there in plaxis.

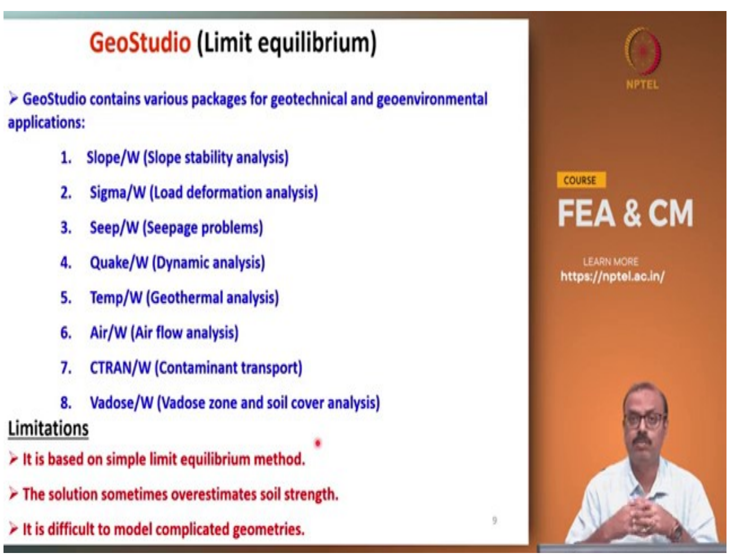
(Refer Slide Time: 04:31)



So, next one in the line is FLAC which is a basically finite difference based software finite difference with package. And it can be used for particularly the complex geometries with the dynamic loading. It also has light plaxis it helps us a certain structural elements which are very good anchors, piles, rock bolts. It is very popular for rock mechanics applications as well when, if you want to do a complete rock model.

However, the limitations are it is not as simple as plaxis. So, you need to know a bit more than what you expect to know in for the plaxis it is extremely difficult to model complicated geometries dynamic analysis may sometimes encounter convergence problem.

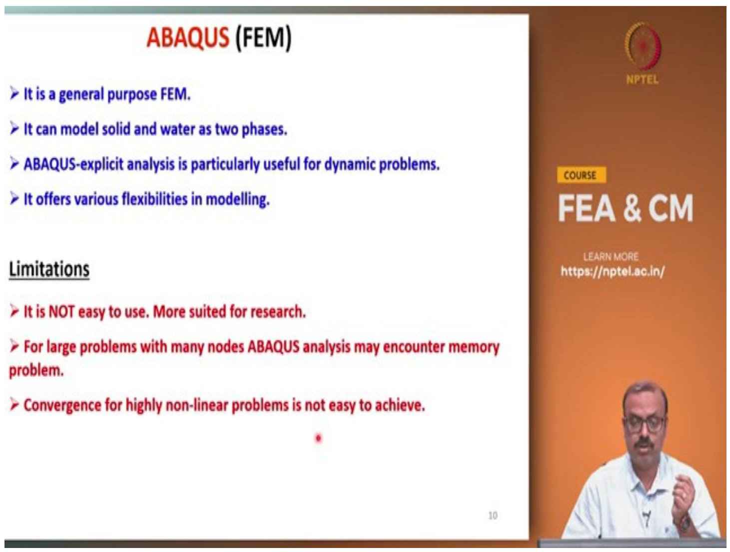
(Refer Slide Time: 05:22)



Then comes the GeoStudio which is which is not a single software. It is a collection of various softwares and it is principally they are based on the limit equilibrium type of approach. This class of softwares having wide variety of explications packages meant to be for geotechnical or geo environmental usage, such as slope W is for slope stability seep W is for seepage problems.

There are, something called dynamic analysis it is a quake W temp W is for geothermal models and so on. How the one of the major limitation is solution, sometimes overestimates the soil strength. It is again it is not suitable for complicated geometries.

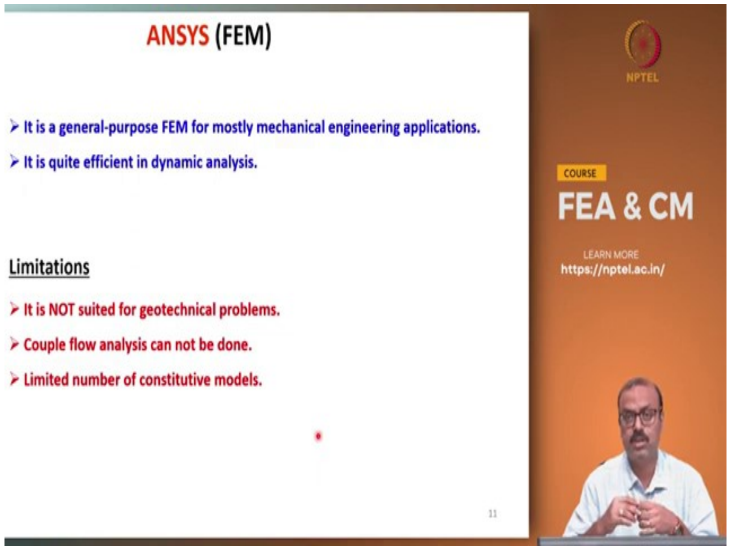
(Refer Slide Time: 06:16)



There are couple of general purpose finite human packages are there, such as ABAQUS is one of them. It is, as I said, it is a general purpose. So, it is quite flexible in nature. However, it is not so much useful I mean it is not that that people can simply plug and play type. It is you have to really learn a lot inside of the finite element to able to use this variety of softwares such as ABAQUS.

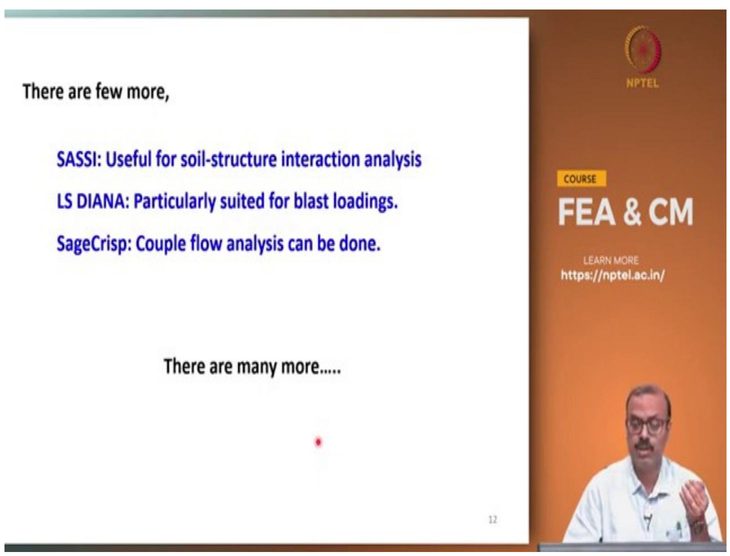
It can model solid and water as two phases, so that is one of the major I think. So, it is a; you can do effective stress analysis but one of the drawback of ABAQUS is where the memory runtime memory if you require more. Then sometimes ABAQUS analysis are not so, efficient in terms of the time running time and the computational cost. So, that is one of the reason why ABAQUS is not so, popular in industry?

(Refer Slide Time: 07:25)



The other one is the ANSYS. This is also a general purpose finite domain package. It is quite efficient again in dynamic analysis, compared to geotechnical side ANSYS is more suitable in structural engineering at that in certain areas. Because there are lot of requirements for geotechnical analysis which it is not there in ANSYS.

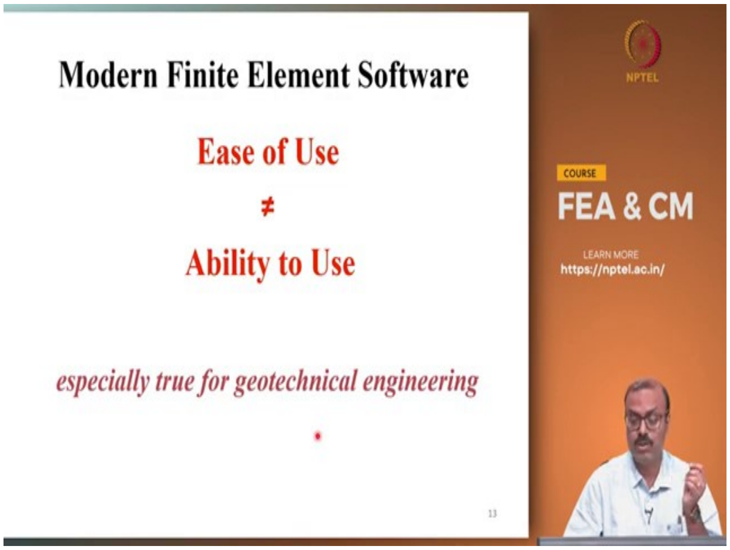
(Refer Slide Time: 07:54)



There are few more few more very specialist application type softwares one of them is SASSI is mint for soil structure interaction analysis, LS DIANA which is particularly suited for someone is working in the blast loadings. SageCrisp which is one of the earlier version of the software where finite domain package, where couple flow analysis can be done, there are many more.

And there are so much development is going on every day. So, you expect that for a specific problem, you will also have a specified package.

(Refer Slide Time: 08:31)

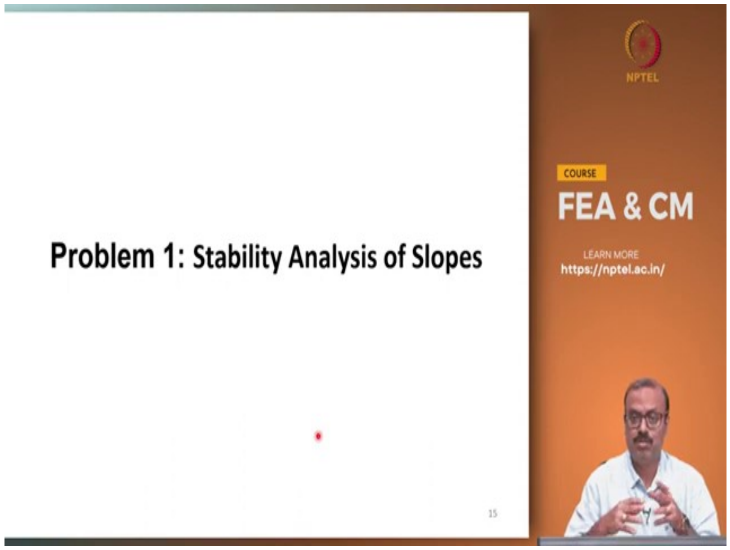


So, in summary, modern finite element softwares are generally very easy. They are scaled based on the ease of use. How easy to use it? But that does not necessarily mean that ability to use. So that is where we have to draw a balance. It is a ease of use to the ability to use, so, there should be a balance. Otherwise, this is particularly true for geotechnical engineering.

(Refer Slide Time: 09:02)

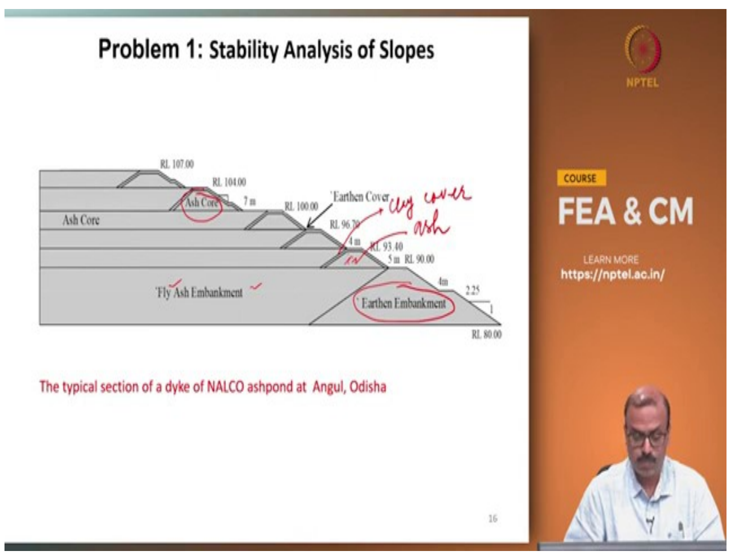
Otherwise, you will have this kind of consequences. I mean which is maybe a simple mistake but consequences it is long term.

(Refer Slide Time: 09:15)



So, let us example where we can use finite element, this type of analysis, to study the geotechnical problems. We start with the simple one where we will analyse slope stability problem.

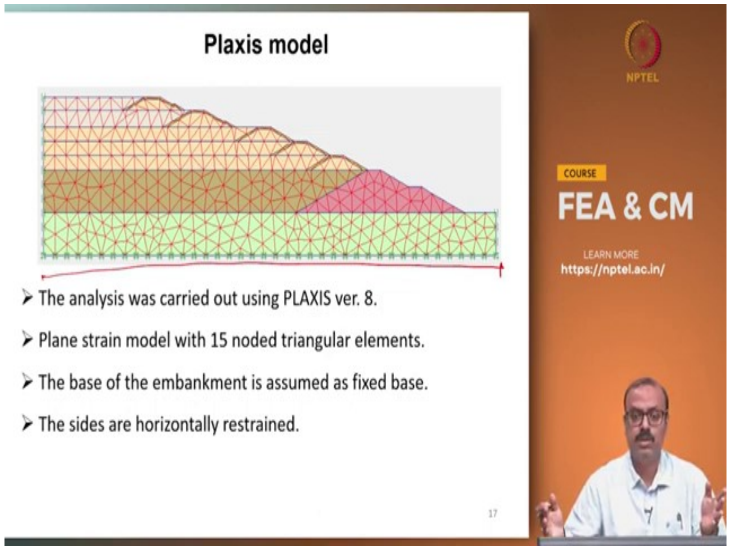
(Refer Slide Time: 09:35)



So, this is a typical slope which we use in for any kind of embankment purpose. It can be like ash pond type of thing. So, basically, this is a ash pond you can see there is a ash core and this is earthen embankment. So, this was the actual earthen embankment starter dyke as we call it and then we have the fly ash getting stored in this kind of areas. And as we as we cross the initial embankment, we need to raise the height of this particular dyke.

So, how it is raised? It is the again inside we fill up with the ash, so, this is again the ash. However, this is the outside there is a clay cover. So that you do not want your ash to fly around and it is not cause any environmental issue. So, this is an actual dyke in one of the aluminium company industry ash type.

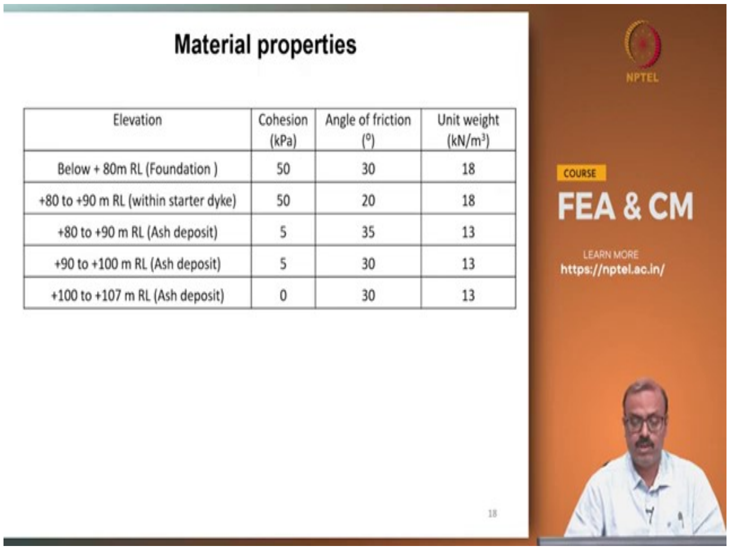
(Refer Slide Time: 10:50)



So, we did the analysis in plaxis as you can see that it has been done in. When did not the latest version of the plaxis and then it is plain strain model with 50 node triangular elements. The base of the embankment is assumed as the fixed phase the sides are horizontally restrained. We have taken the one of the major issue of finite element is how to decide that the width of this model this dimension?

So, there are certain rules are there but I would say that it is more like a we have to do a sensitivity analysis to understand. How big or what should be the preferred width of the model? You have to do a trial and error method to fix up these boundaries.

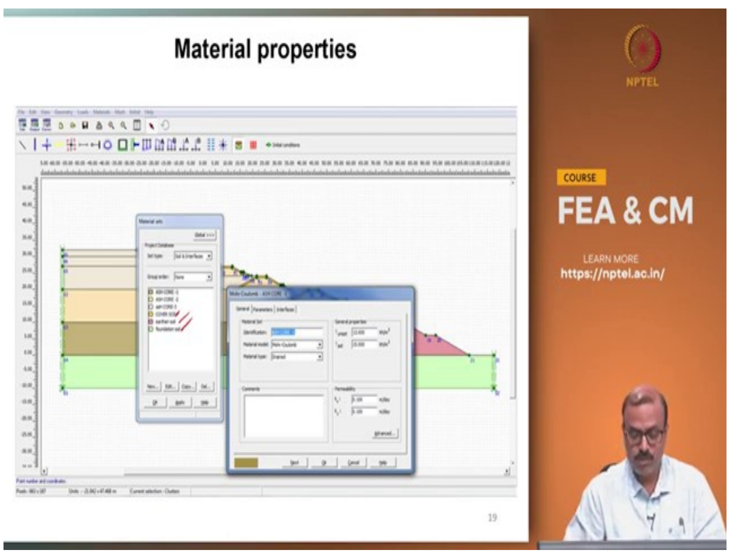
(Refer Slide Time: 11:40)



Elevation	Cohesion (kPa)	Angle of friction (°)	Unit weight (kN/m³)
Below + 80m RL (Foundation)	50	30	18
+80 to +90 m RL (within starter dyke)	50	20	18
+80 to +90 m RL (Ash deposit)	5	35	13
+90 to +100 m RL (Ash deposit)	5	30	13
+100 to +107 m RL (Ash deposit)	0	30	13

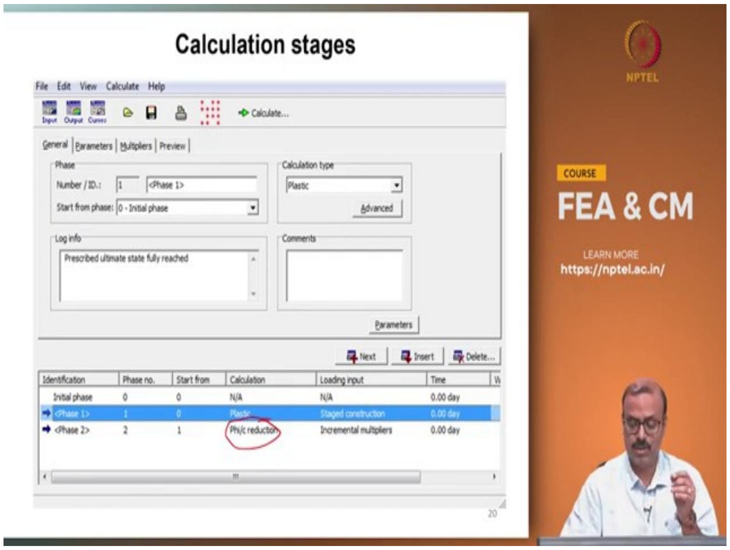
So, we got actual soiled it has oil and the fly ash properties from the test done at the side. So, we use that.

(Refer Slide Time: 11:49)



So, we incorporated the material sets in plaxis, so, we have ash core, ash core 1, 2, 3. There are three levels of raisings and then that is earthen cover original soil and then the foundation soil and then we have in given the input properties.

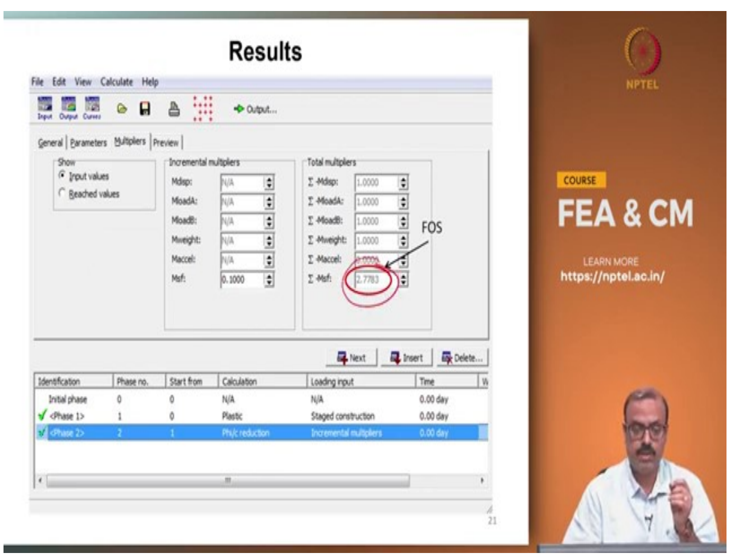
(Refer Slide Time: 12:10)



So then we went for the calculation stages, we did the plastic analysis, we allow the soil to or soil and the ash to deform in plastically manner. Followed by this phi c reduction this phi c reduction is basically the stability analysis. That means what in plaxis we it is the program will do is they will forcefully reduce the c and phi of the soil property? So, as the c and phi soil property reduces.

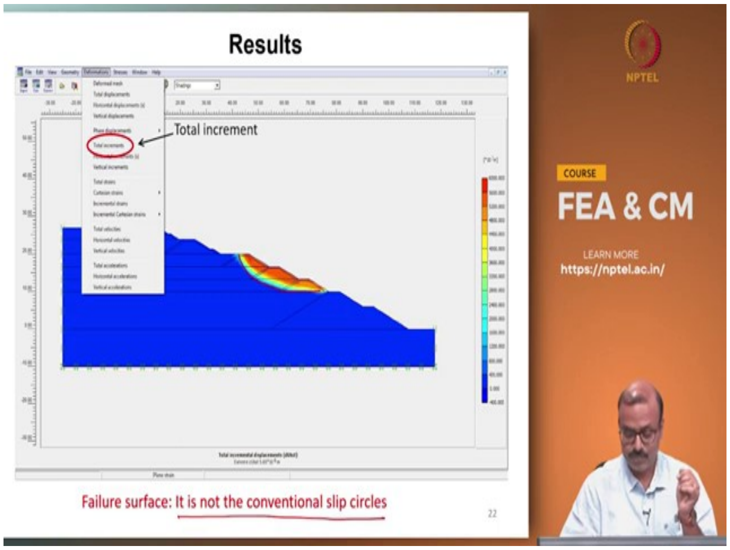
They will see at which point of mobilized cohesion and the friction the failure happens. So, the ratio of mobilize to the original strength properties will give you the factor of safety. So, either they call it a safety analysis or a phi c reduction. So, name itself suggests that we are reducing the shear strength of the soil.

(Refer Slide Time: 13:04)



So that is where the factor of safety develops in the cumulative multipliers.

(Refer Slide Time: 13:13)

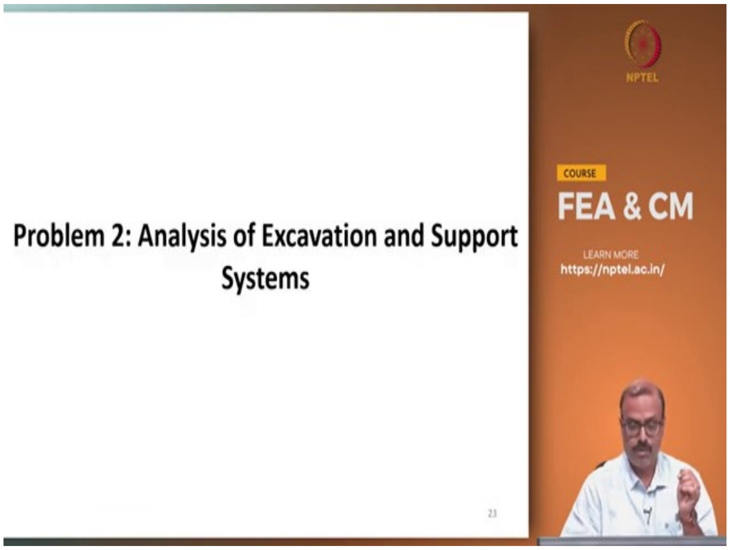


So, we can actually see the slip circle so, here this contour will tell that. What? When is the critical failure circle? So, you have to choose the total increments contour plot and then you will see the failure surface. So, you have to keep in mind that this is kind of a representative of the slip circle but principally it is not the actual conventional slip circle which we generally do for Bishops method or similar liquid equilibrium method.

This is gives you the failure I would say that this can be considered as the part of the embankment which is getting fell or which is getting plastic mobilization happen. But despite that you can have an idea that where your slip is actually bound to occur. It may not be exact matching with the limit equilibrium but it will give a indication. So, this is a simple exercise for the slope stability.

Usually, the general understanding is that the factor of safety which you obtain from the finite element analysis for a slope, is generally slightly more conservative side compared to the factor of safety which you obtain from the limit equilibrium approach. Like your bishops method or similar limit equilibrium method or Morgenstern-Price approach and so on.

(Refer Slide Time: 14:49)

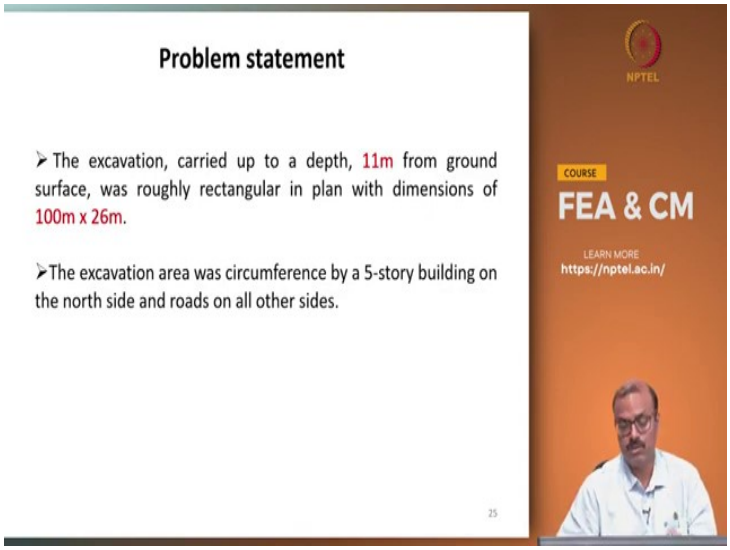


So, the next problem which I want to discuss here, is a slightly more complicated in terms of the analysis, it is the analysis of excavation and the support system.

(Refer Slide Time: 15:04)

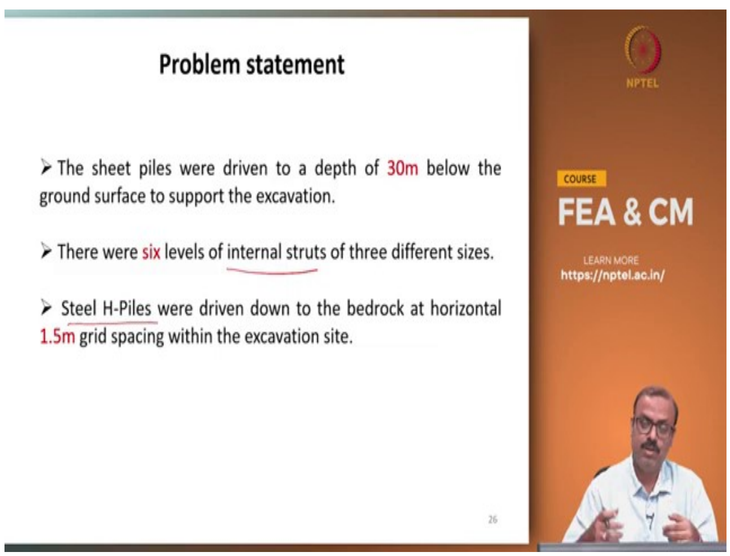
So, as you can see, this is excavation site in an extremely congested area. It is extremely congested urban area where we are going to do a excavation.

(Refer Slide Time: 15:19)



So, the excavation carried up to a depth of 11 metre from ground surface because top soil was a kind of a soft clay. So, you have to cross that soft clay and you have to put to have a support system for that. It is a quite a large plan dimension for the excavation 100 metre by 26 metre. The excavation area was circumference by 5-storey, building on a north side roads on all other side. So, again, as I said earlier, it is a very congested urban environment.

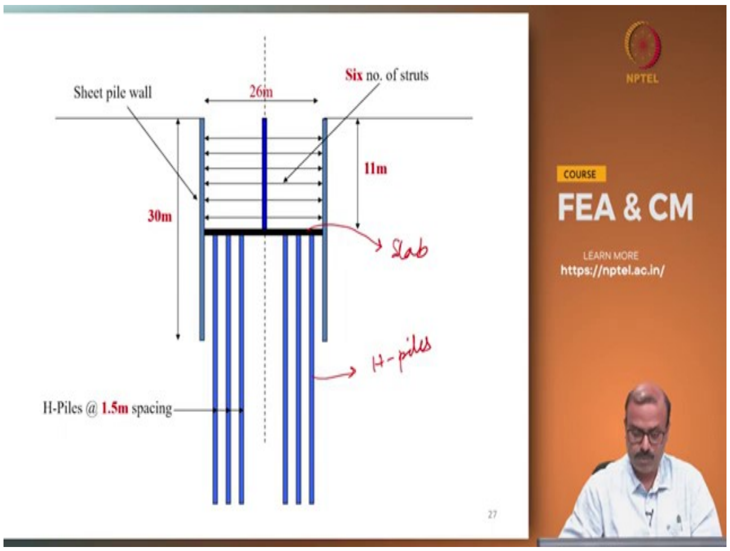
(Refer Slide Time: 15:55)



So, the sheet piles were first driven to a depth of 30 metre below the ground surface to support the excavation. As I said, the top layer is a soft clay. So, you have to really go deeper to get a proper anchoring of the sheet pile. There are six levels of internal struts. So, it is you have a started excavation at three different sizes. On top of that there will be a steel piles driven down the bedrock.

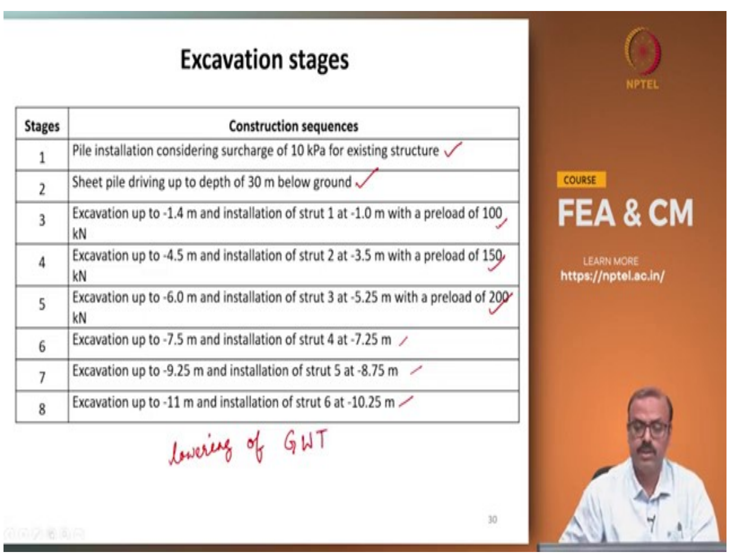
Because at the excavation floor you have a slab to be cast for that slab will it is not the soil is not good enough to support that slab. So, you really have to have a piles supported slab at the excavated level.

(Refer Slide Time: 16:36)



So, this is typically the case. So, as you can see, this is the excavation depth of 11 metre and this is the slab which is supported by this H-piles. So, what should be the step of the or stages of construction?

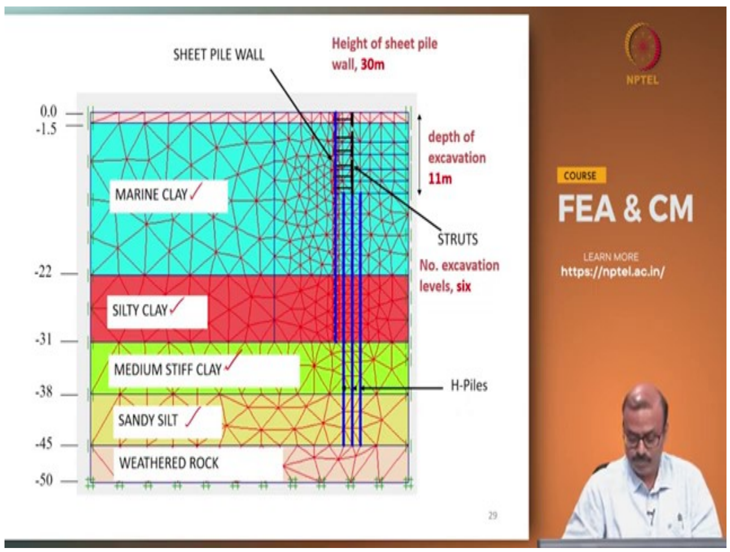
(Refer Slide Time: 16:56)



So, I just explain the stages. Then we will go back to the soil properties. So, we have to first install the pile, pile installation, so, we just simply insert a hammer in the H-piles. Of course later we will cut it up to the excavation level. Then we will drive the sheet piles. Then we do the excavation up to –1.4 metre. Then we install the strut and preload it and then similar thing we repeated up to the first up to the 6 metre where strut 1, strut 2 and strut 3 installed.

As you can notice that the preloading amount increase again these are the thing data came from the actual site. Preloading increased because to control the preloading of the start, increase to control the deflection of the sheet piles and then beyond 7.5 metre you simply install. One of the major issue here is the for each state you have to have a lowering of groundwater table so that accompanied with this cutting, so, we will discuss that also.

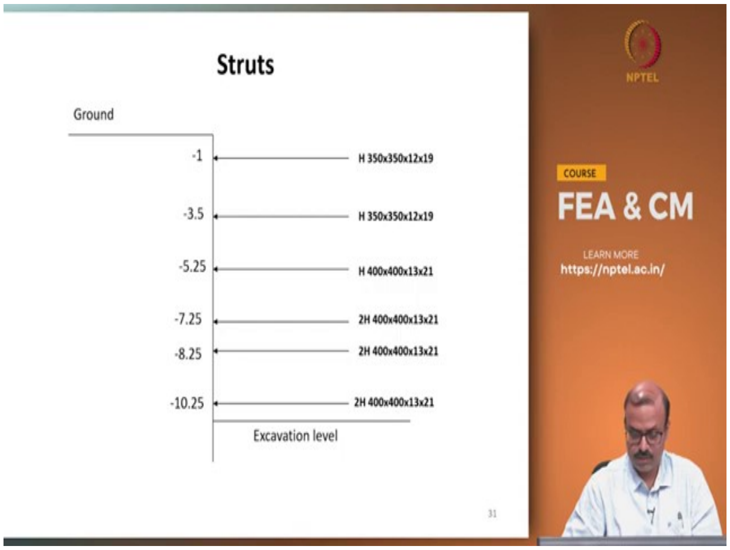
(Refer Slide Time: 18:09)



So, if you see the soil profiles, so, there are explained with this itself. So, there is a thick layer of marine clay and then there is a silty clay which is not bad in terms of string. There is a medium stiff clay which is good which is slightly over consolidated, followed by sandy silt and almost close to 45 metre. You will have the weathered rock starting. So, this H-piles are resting on the surface of the weathered rock.

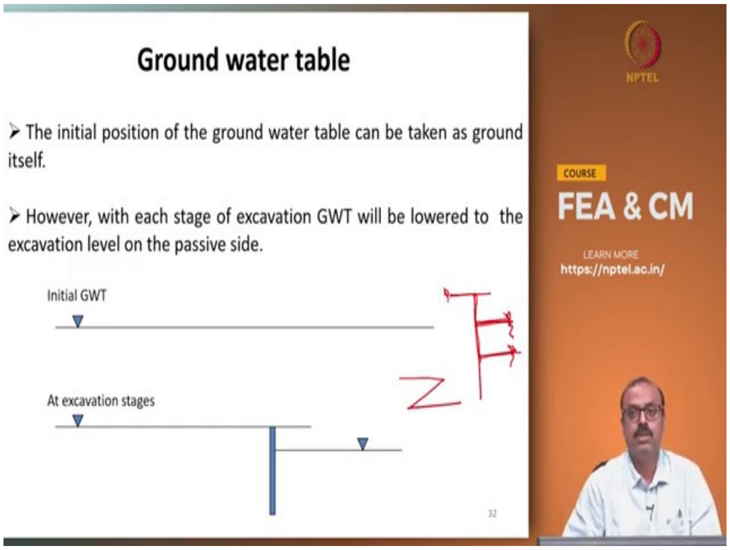
So, excavation is 11 metre is that means excavation done within the marine clay itself. So that is why you need so many I mean different levels of strut. So, as you can see again in the stages that we have six levels of strut but only top three level, we need are preloading to control the pile the sheet pile deflection.

(Refer Slide Time: 19:07)



So, this is the different start locations with different strut sections.

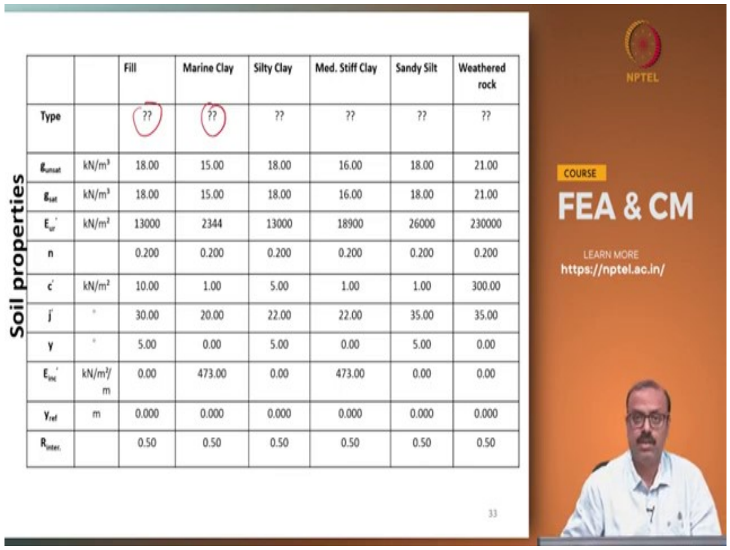
(Refer Slide Time: 19:14)



So, as I said the initial position of the ground water table, it is almost at the ground, it is a coastal area. So, we can take it as a ground. And then each excavation states that ground water table lowered. So, we call it typical as zip type arrangement, so that means you have this, so, you excavate up to this initially ground water table is like this. So, once you have the excavation up to this. So, you lower the groundwater table like this.

So, it is like as kind of staggered arrangement. Then you go to the next step, so, you further lower the ground water table this. So that is how we model and that is how it will be exercised in the field as well.

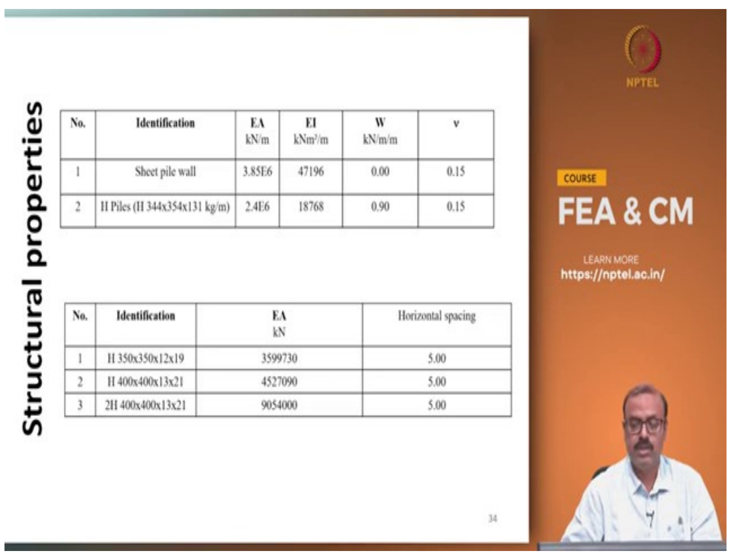
(Refer Slide Time: 20:02)



		Fill	Marine Clay	Silty Clay	Med. Stiff Clay	Sandy Silt	Weathered rock
Type		??	??	??	??	??	??
g_{unsat}	kN/m³	18.00	15.00	18.00	16.00	18.00	21.00
g_{sat}	kN/m³	18.00	15.00	18.00	16.00	18.00	21.00
E_{ur}'	kN/m²	13000	2344	13000	18900	26000	230000
n		0.200	0.200	0.200	0.200	0.200	0.200
c'	kN/m²	10.00	1.00	5.00	1.00	1.00	300.00
j'	°	30.00	20.00	22.00	22.00	35.00	35.00
y	°	5.00	0.00	5.00	0.00	5.00	0.00
E_{inc}'	kN/m³/m	0.00	473.00	0.00	473.00	0.00	0.00
y_{ref}	m	0.000	0.000	0.000	0.000	0.000	0.000
$R_{inter.}$		0.50	0.50	0.50	0.50	0.50	0.50

So, this is a typical property set which came from the soil investigation. Please remember this question, so, we will come back with this the type is strain and underneath type of analysis which will discuss very shortly.

(Refer Slide Time: 20:18)

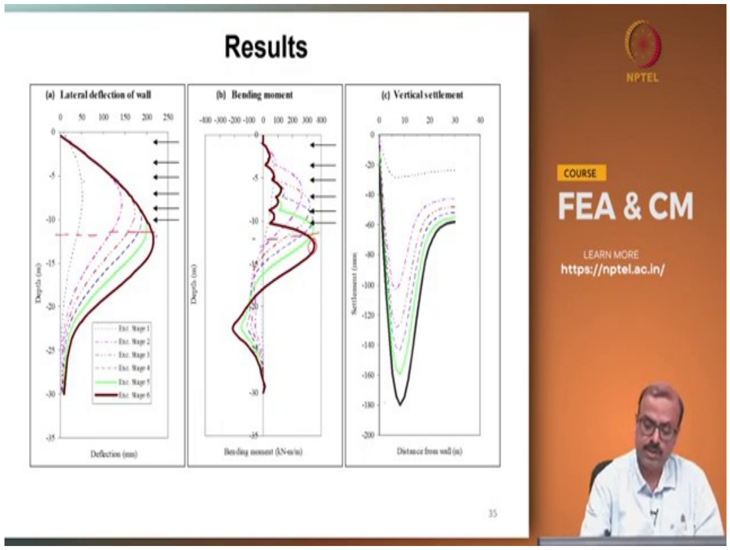


No.	Identification	EA kN/m	EI kNm²/m	W kN/m/m	v
1	Sheet pile wall	3.85E6	47196	0.00	0.15
2	H Piles (H 344x354x131 kg/m)	2.4E6	18768	0.90	0.15

No.	Identification	EA kN	Horizontal spacing
1	H 350x350x12x19	3599730	5.00
2	H 400x400x13x21	4527090	5.00
3	2H 400x400x13x21	9054000	5.00

So, this is the structural properties of the sheet pile, H-piles and the struts. Essentially, the struts are actual members, so, we have given the actual rigidity. Sheet piles and H-piles we have given both actual rigidity as well as the flexural rigidity.

(Refer Slide Time: 20:35)



So, this is the typical, results obtained. As you can see, we have the lateral deflection, winning movement and the vertical settlement lateral deflection is and for this, each dotted line represents that. How we monitor the deflection for each excavation stage and after installation of the strut? So, you can see that lateral deflection, gradually increases and maximum after state six that means when we reach the excavation of 11 metre.

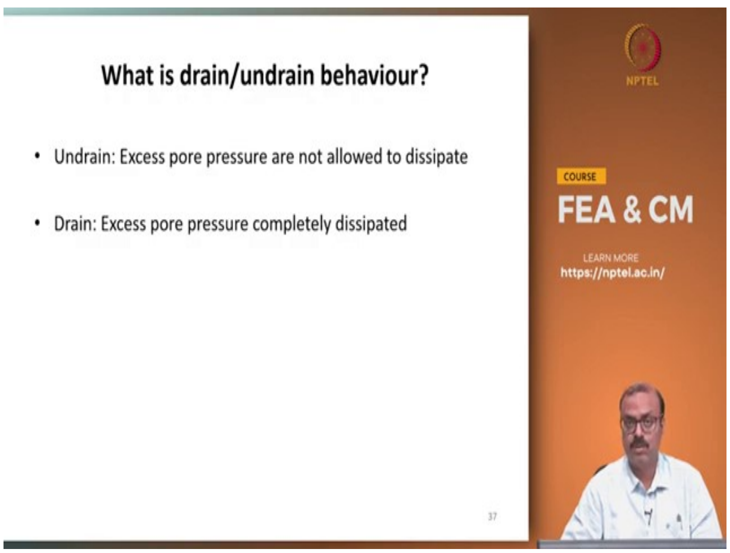
So, 11 metre is somewhere here. So, you are getting the maximum deflection slightly below the excavation depth. Same thing observed in the bending moment diagram also, you can see that it is maximum bending moment occurred, some place below the little below the excavation depth. We also measured the, what is happening in the vertical ground in the ground surface?

Because as I said that this is a congested environment, so, to track how the ground around the excavation is settling also is a key issue. As you can see, there is a very significant settlement of 180 mm expected at the ground level.

(Refer Slide Time: 21:56)

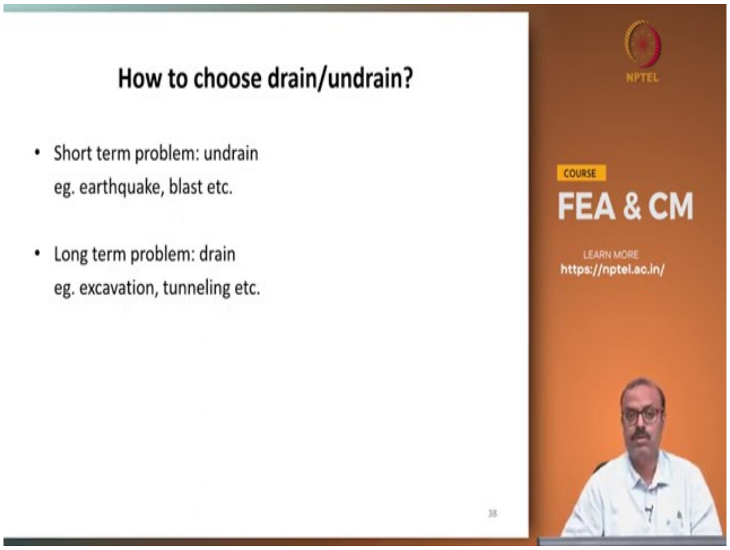
So now, as I said that there is one of the major issues about is the type of analysis whether drain or undrain.

(Refer Slide Time: 22:03)



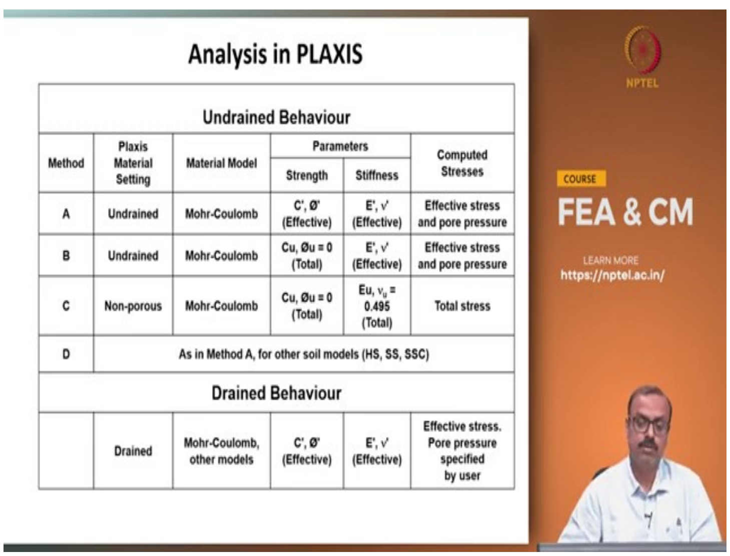
So, what is drain and undrain behaviour? I know that this course has been discussing this drain and undrain in quite a number of times. So, in we will just see how this, when we use any software, how they are treating it? Now, before going to that undrain generally excess pore pressure are not allowed to dissipate as we all know. Drain excess pore pressure is completely dissipated. That is a simple way of looking at it.

(Refer Slide Time: 22:30)



So, how to choose drain and undrain? So, drain and undrain generally, we call it short term problem, earthquakes, blast where you are not allowing the excess flow pressure to receive it. Long term is excavation, tunnelling and so on because here long-term performance is key issues, particularly if you talk about the cohesive material.

(Refer Slide Time: 22:53)

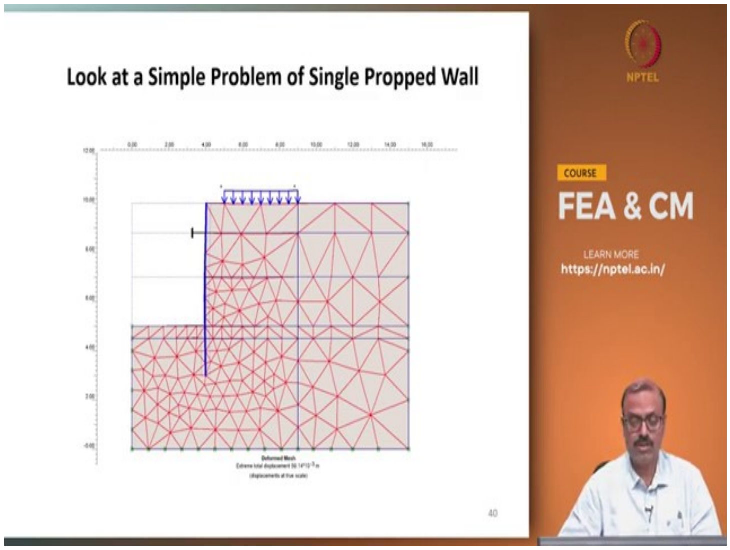


Undrained Behaviour					
Method	Plaxis Material Setting	Material Model	Parameters: Strength	Parameters: Stiffness	Computed Stresses
A	Undrained	Mohr-Coulomb	C', Ø' (Effective)	E', ν' (Effective)	Effective stress and pore pressure
B	Undrained	Mohr-Coulomb	Cu, Øu = 0 (Total)	E', ν' (Effective)	Effective stress and pore pressure
C	Non-porous	Mohr-Coulomb	Cu, Øu = 0 (Total)	Eu, ν_u = 0.495 (Total)	Total stress
D	As in Method A, for other soil models (HS, SS, SSC)				
Drained Behaviour					
	Drained	Mohr-Coulomb, other models	C', Ø' (Effective)	E', ν' (Effective)	Effective stress. Pore pressure specified by user

So, in plaxis you can do it in a various way there is they call it method, A, B and C. So, you plaxis material setting should be undrained. If you choose the material model Mohr-Coulomb then you have to in A. You have to use all the effective strength and the effective stiffness property. So that the computed stresses or effective stress and the pore pressure. In B undrained method is where you use strength as total stress but stiffness at effective stress.

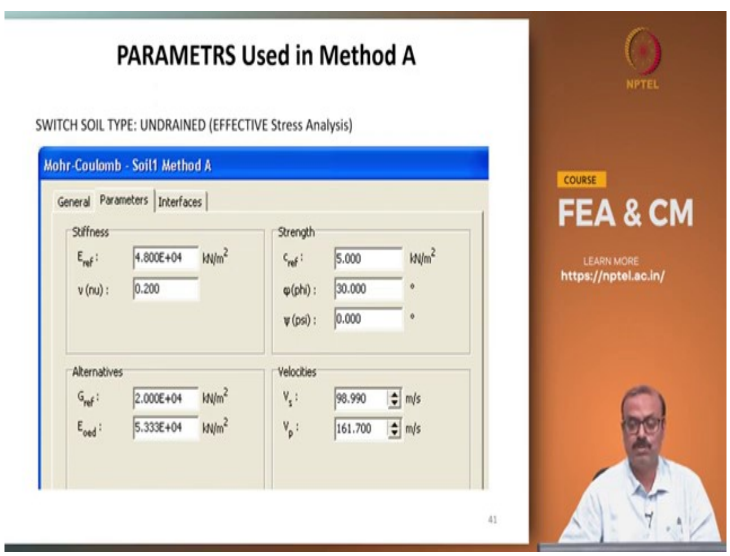
So, you again you get effective stress and pore pressure as the computed response. If you choose non-porous there also you can, I mean, do undrained analysis, so, there it is everything will be the total stress both the total stress, stiffness strength, as well as the output.

(Refer Slide Time: 23:48)



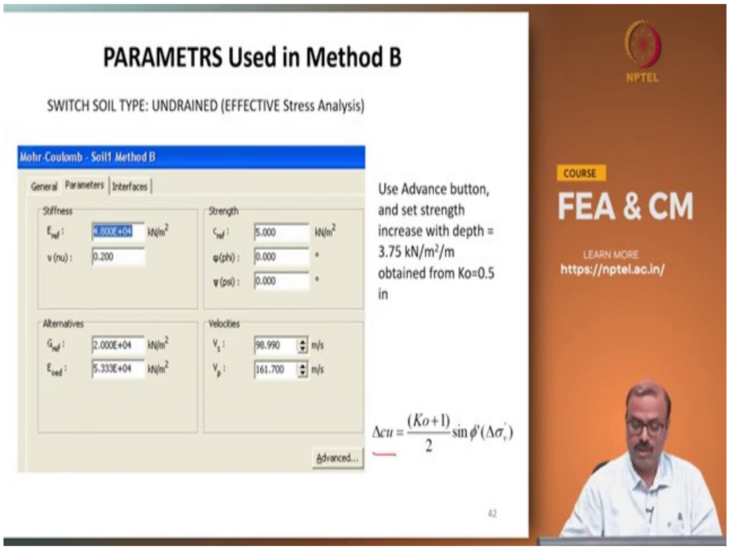
So now, we will do a simple cross check that what we said is correct or not? So, if we use the same at same problem, same a trial problem and do it in these three different methods. Let us study.

(Refer Slide Time: 24:06)



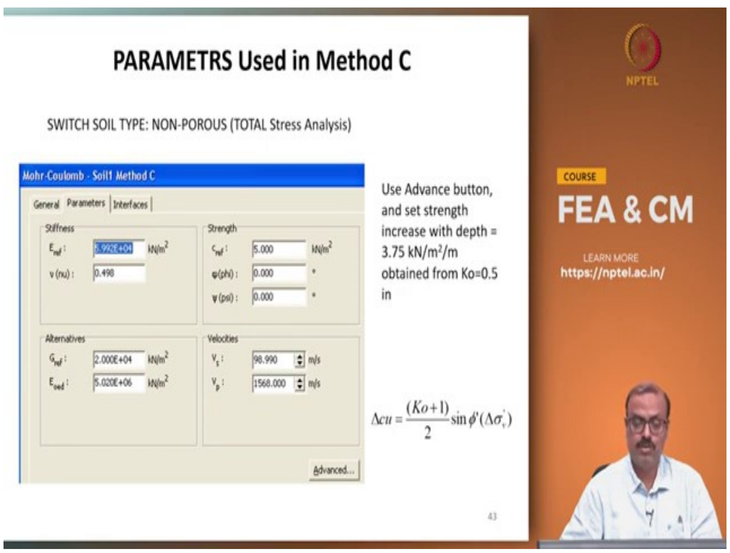
So, this is a simple excavation problem which we just analysed a similar line. So, for method A we used as you remember, it should be all effective stress properties. So, we have effective strength and the effective stiffness properties.

(Refer Slide Time: 24:19)



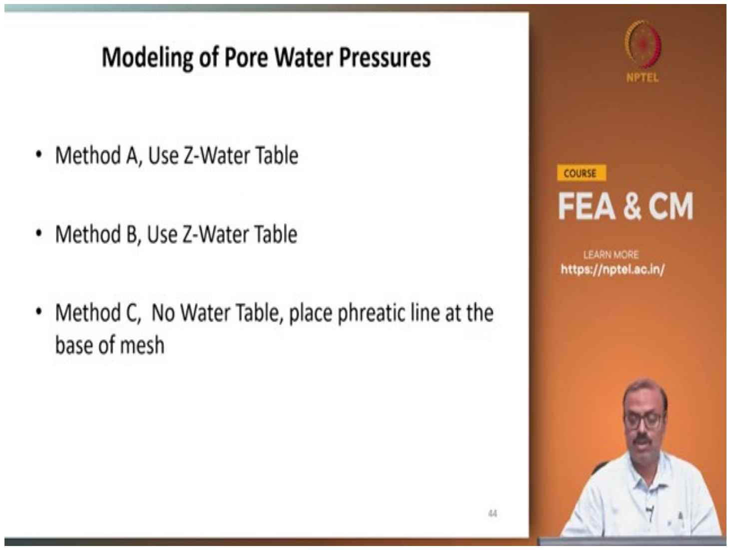
Method B it is the effective strength and the total stiffness. So, as you can say that we, it is still, it is effective stress analysis. We have a delta cu increase. How we have done?

(Refer Slide Time: 24:41)



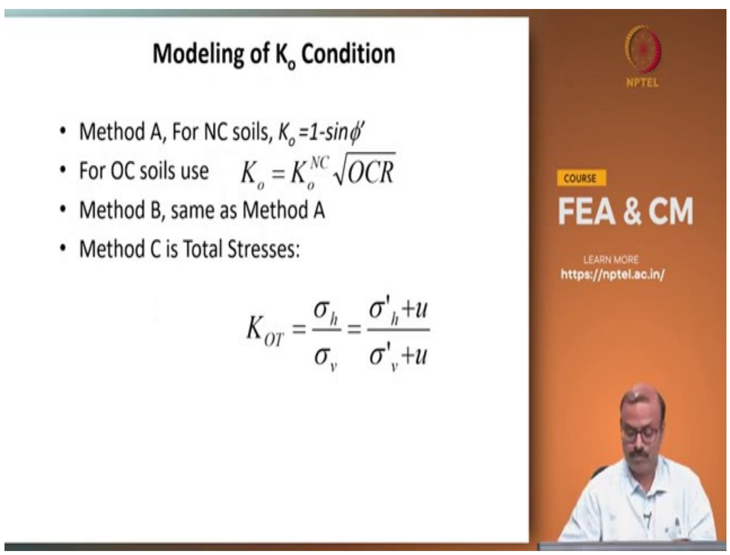
And then if it is method C everything total so, we have again use the total stress or total string properties.

(Refer Slide Time: 24:49)



So, all the for top two cases A and B use the conventional way of handling water table which I just explained it is a Z-type water table. In method C there is no water table, it is a nonporous, so, place phreatic line at the base of the mesh.

(Refer Slide Time: 25:06)



And you have the modelling of the K naught conditions so, method A for normally consolidated soil you should use cannot, as a as for the Jackie's rule 1 – sin phi dash. For over consolidated soil you can use, K naught NC with a factor under root, OCR. Method B same as method C in terms of K naught conditions. Method C is a total stress, based approach.

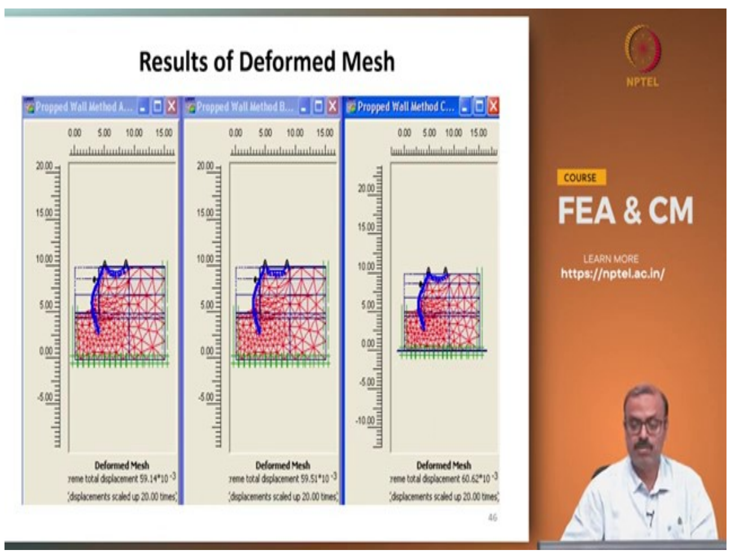
Modeling of K_o Condition

- Method A, For NC soils, $K_o = 1 - \sin\phi'$
- For OC soils use $K_o = K_o^{NC}\sqrt{OCR}$
- Method B, same as Method A
- Method C is Total Stresses:

$$K_{OT} = \frac{\sigma_h}{\sigma_v} = \frac{\sigma'_h + u}{\sigma'_v + u}$$

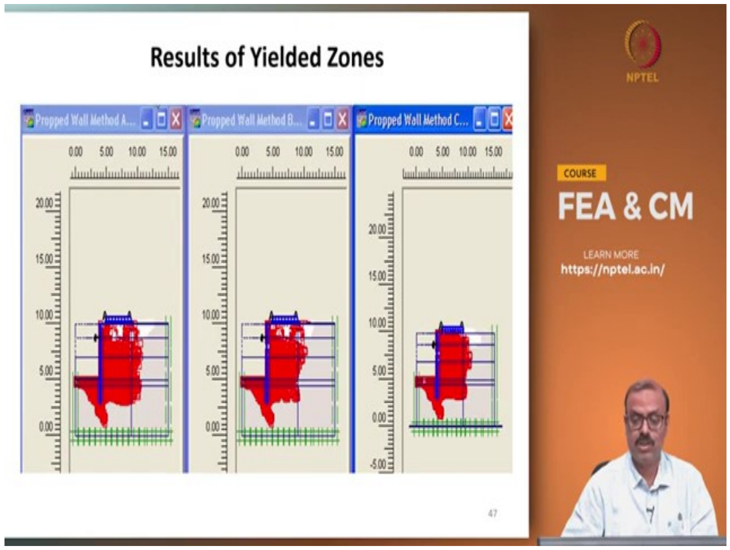
So, here you have to consider that it is the total stress K naught condition. So, use the sigma h over sigma v not sigma h dash over sigma v dash.

(Refer Slide Time: 25:43)



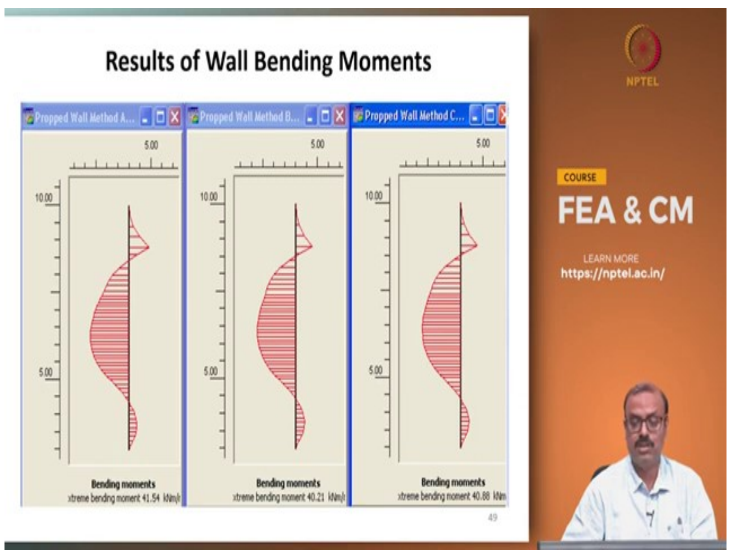
So, if you use properly, you can see that the deformations are very similar.

(Refer Slide Time: 25:49)



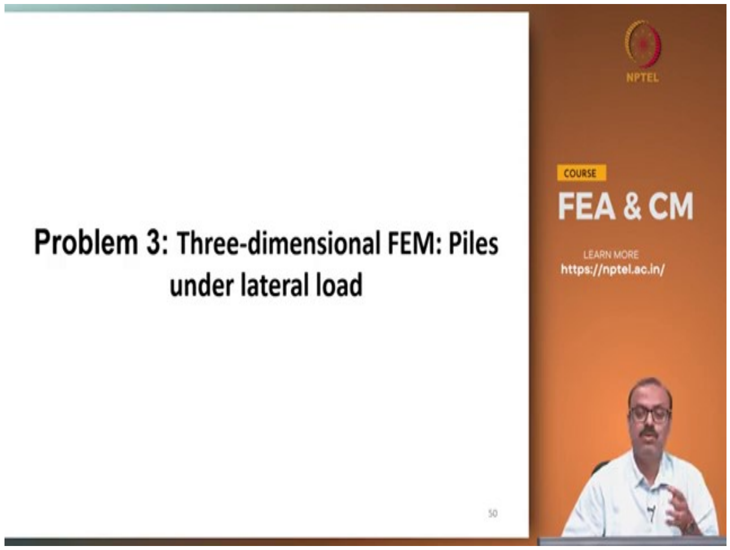
The plastic zones or the yield zones are quite close.

(Refer Slide Time: 25:55)



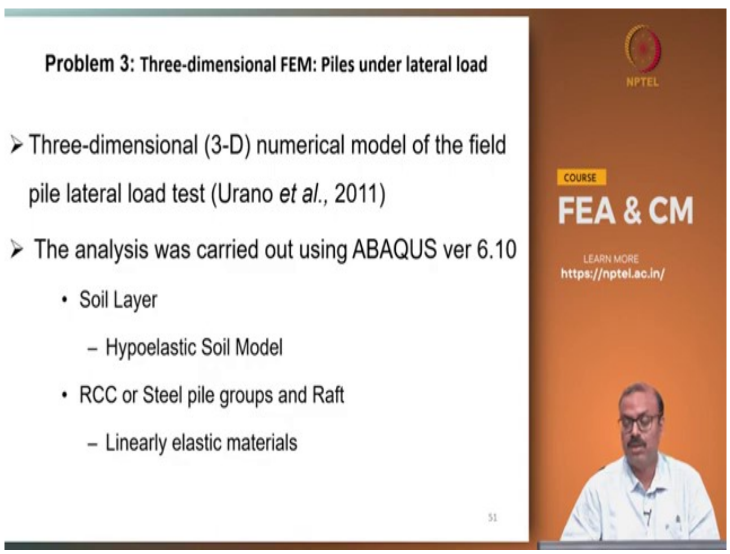
Even the bending movement wall deflections winning moments, everything is are coming within a reasonable values. It is not totally different, I would say.

(Refer Slide Time: 26:06)



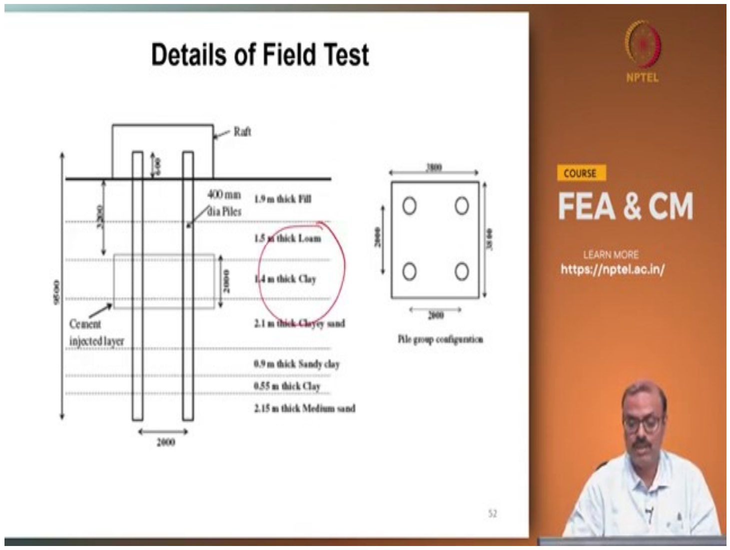
So, with this two simple plane strain problem will go to that problem three. Where we will study the 3-dimensional FEM for we will study a simple response of a pile under lateral load.

(Refer Slide Time: 26:23)



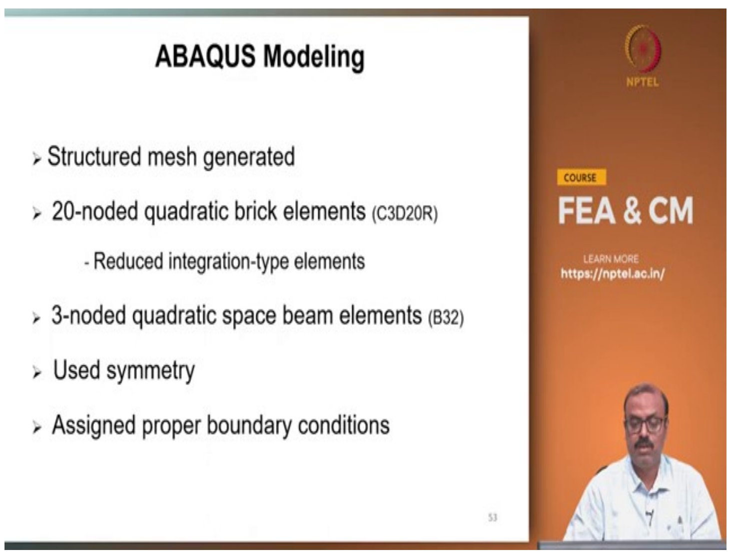
So, this is a 3-dimensional numerical model of a field pile lateral load test reported by Urano in Japan. The analysis was carried out in ABAQUS. The soil layer used as a hypo elastic soil model. Steel piles and the raft pile groups everything all the structural elements are considered as linear elastic material.

(Refer Slide Time: 26:46)



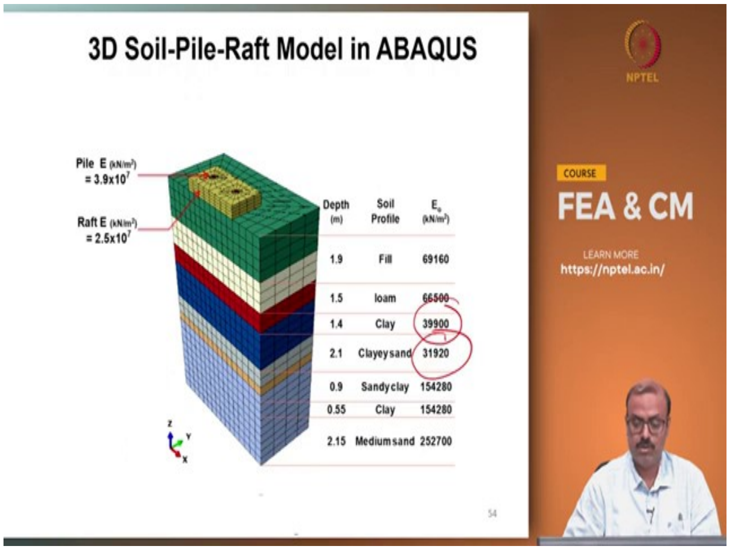
So, this is the typical soil layer is a major your attention should be in this particular layer where there is a loam and clay layer is there which is relatively soft.

(Refer Slide Time: 26:58)



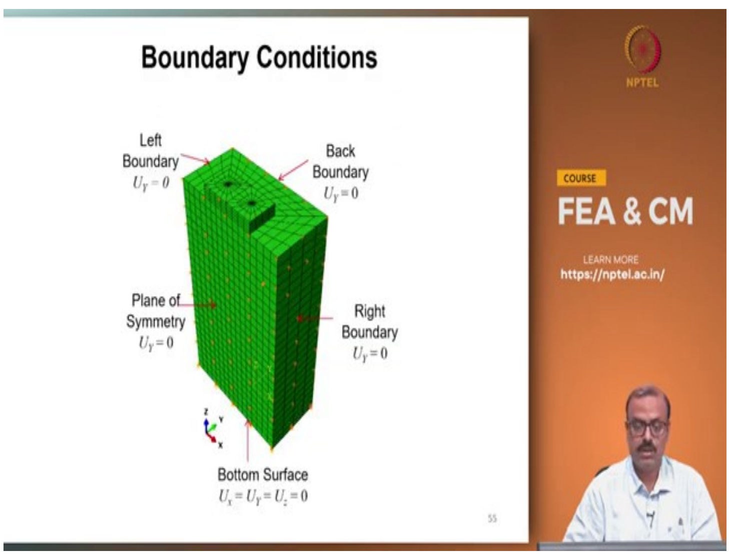
So, we use 20-noded quadratic brick elements with reduced integration. Then 3-noded quadratic space beam element to model the piles.

(Refer Slide Time: 27:10)



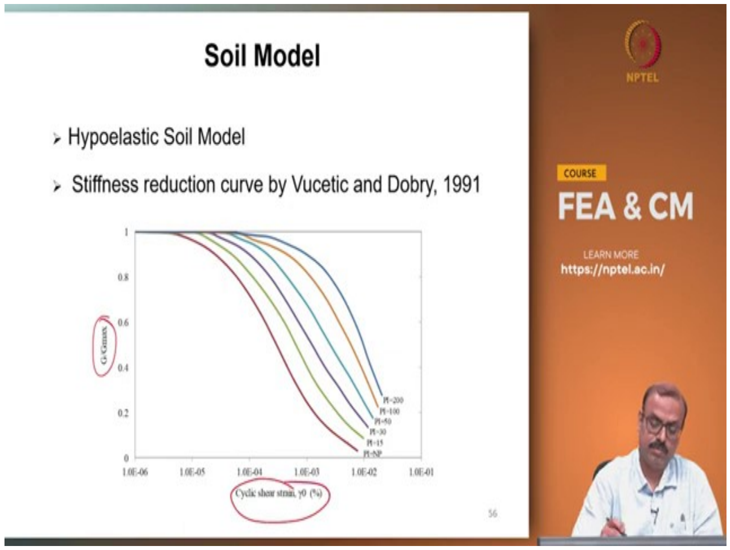
This is the soil property as I said, there is a clay layer of slightly lower strength and stiffness, even here both the layers having a problem. So, there will be a lateral load issue in those particular layers.

(Refer Slide Time: 27:30)



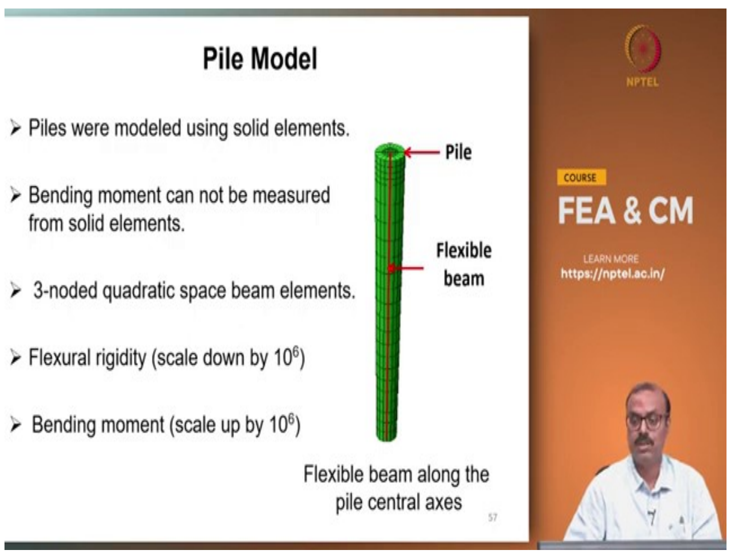
So, this is the boundary conditions.

(Refer Slide Time: 27:32)



So, this hypoelastic soil model where you it is basically a non-linear elastic type of model where you have a modulus varies with strain.

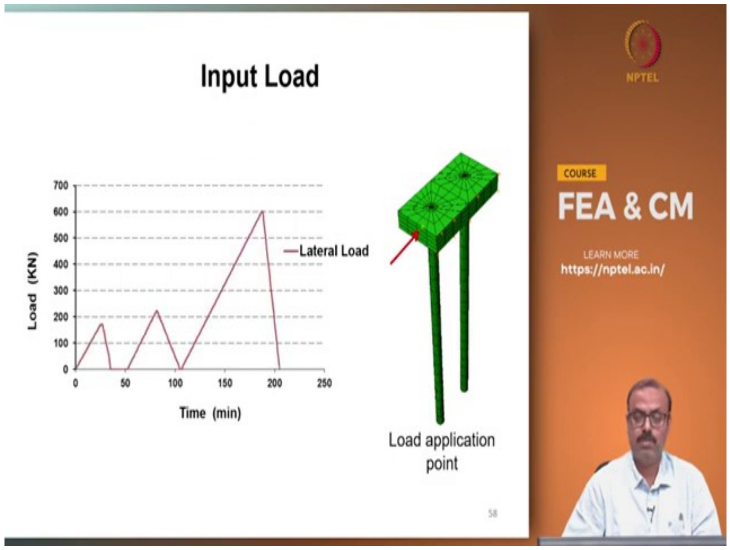
(Refer Slide Time: 27:47)



So, the pile there is a challenge of modelling pile because piles are generally modelled using the solid elements. Now, in solid elements bending on moments directly cannot be measured. So, we have to have a 3-noded quadratic space element inserted within the pile. And those space p minimum flexural rigidity it is scaled down by 10 to the power 6. And bending moment will be whatever the computed bending moment we have to multiply with 10 to the power 6.

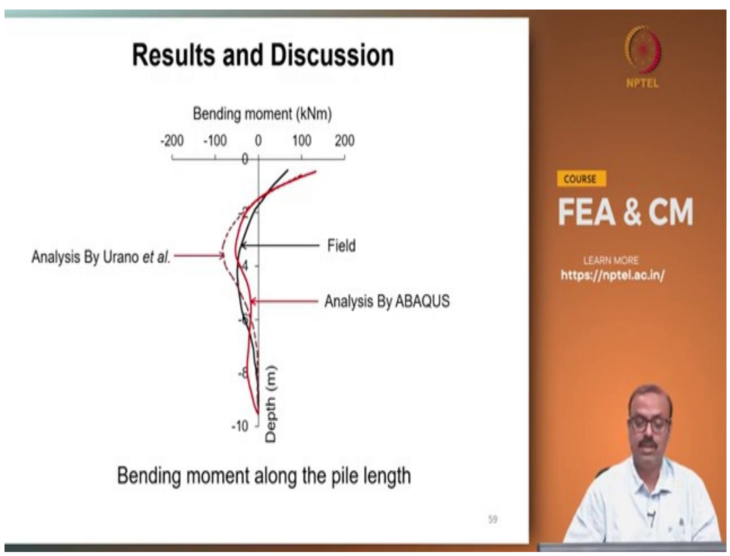
Now, one should keep in mind. This is possible if your structural elements are linear elastic. We assume here it is linear elastic, so that is why we are doing. So, if it is not in an elastic what we should do? We will have the next or the final problem there we will discuss that.

(Refer Slide Time: 28:35)



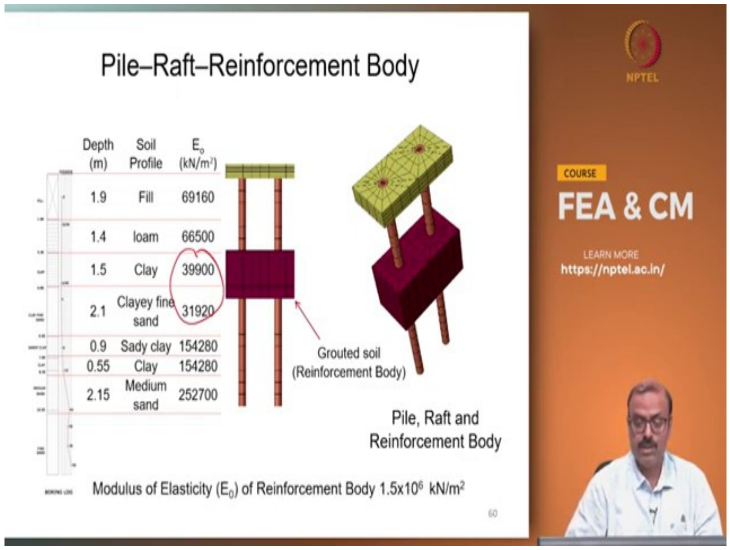
So, the input load is a basically a pulse load. So, it is a triangular pulse applied.

(Refer Slide Time: 28:43)



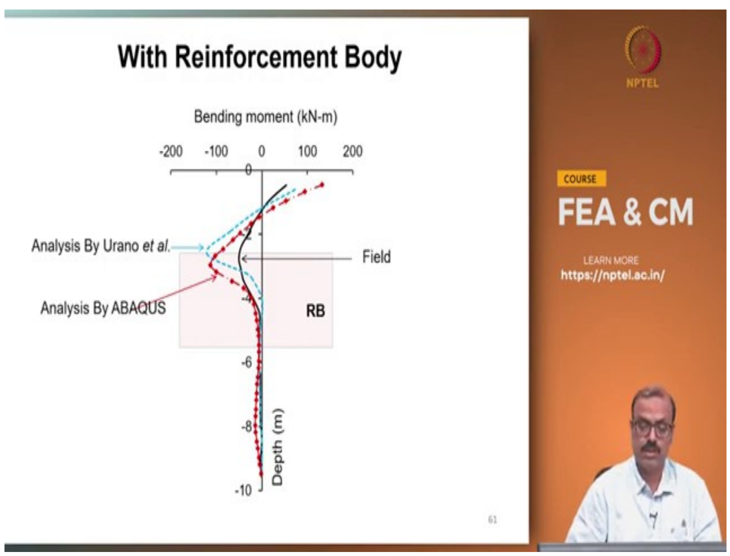
So, let us see how it compares with the field experiment? So, this is the field experimental data reported by Urano. And this is the analysis is a limit equilibrium type of analysis done by the Urano. Our ABAQUS analysis is somewhere falls in between the field and the Urano's analysis. As you can clearly see, Urano is quite grossly over predicted the field but our thing is slightly better.

(Refer Slide Time: 29:11)



So now, the problem is, as I said earlier that there is the lateral load carrying capacity was a question in this particular case. So, because this soft layer, so, what it is planned by the site is like? Let us have a grouted soil. So, we call it reinforcement body. So, we model that also in the abacus and then it is a linear elastic material we assume.

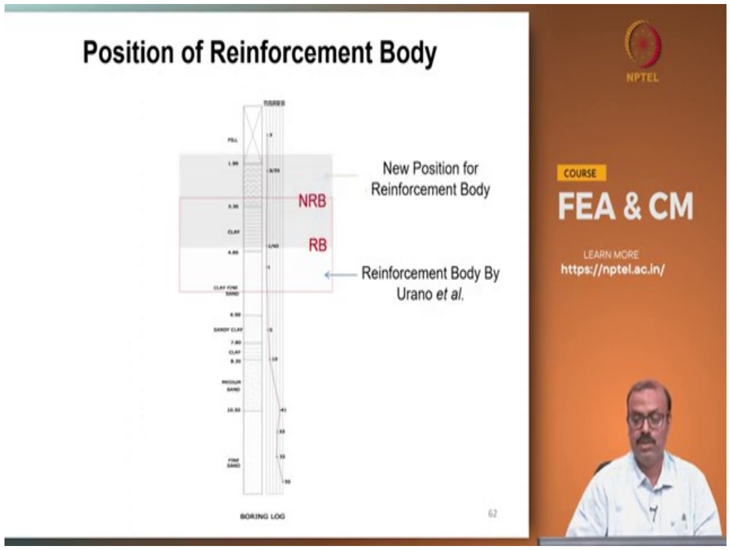
(Refer Slide Time: 29:38)



So, if you put that reinforcement body, so, this is the field test data. So, this analysis Urano around again it is over predicted. However, in ABAQUS also it is more or less close to the analysis by Urano. There are various reasons if possible, one of the major reason may be both the cases the ABAQUS and the Urano analysis analytical model assume this reinforcement body has linear elastic which may not be the actual case.

However, one of the things which gives a satisfaction is that the bending movement is slightly over predicted which is from the safety point of view. It is fine.

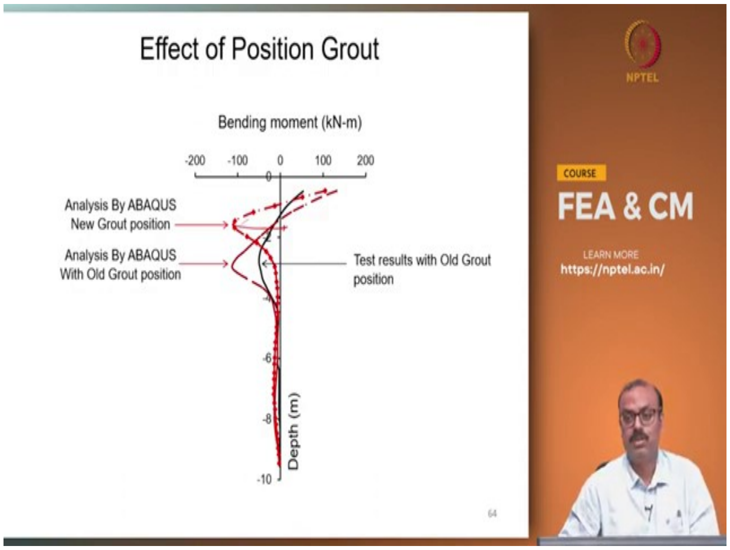
(Refer Slide Time: 30:19)



(Refer Slide Time: 30:25)

We did a bit of parametric study to see that if you change the position of the reinforcement body depending on the soil condition whether that makes any difference.

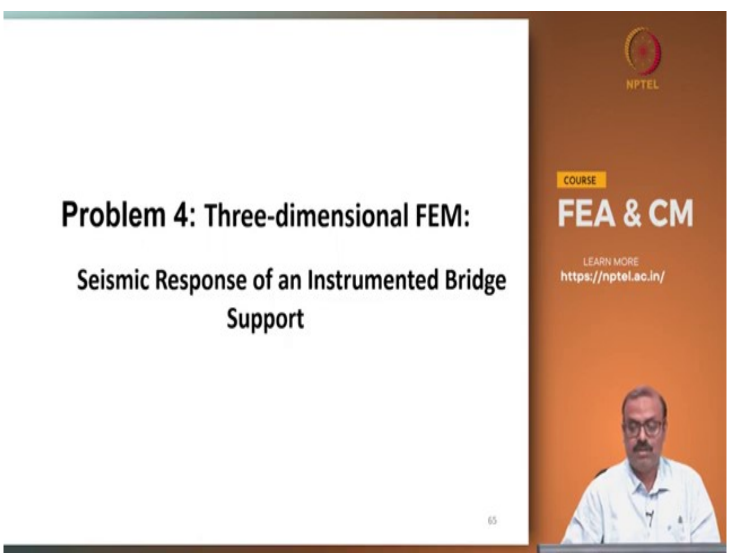
(Refer Slide Time: 30:28)



So, this is the old grout position. This is analysis by old grout position in ABAQUS. However, there is no change in the winning movement but being said and done, there is a change in the location of the maximum moment. So, it shows that the reinforcement body is giving a kind of a fixity around the pile. So, it is domaining moment development is localized to that region.

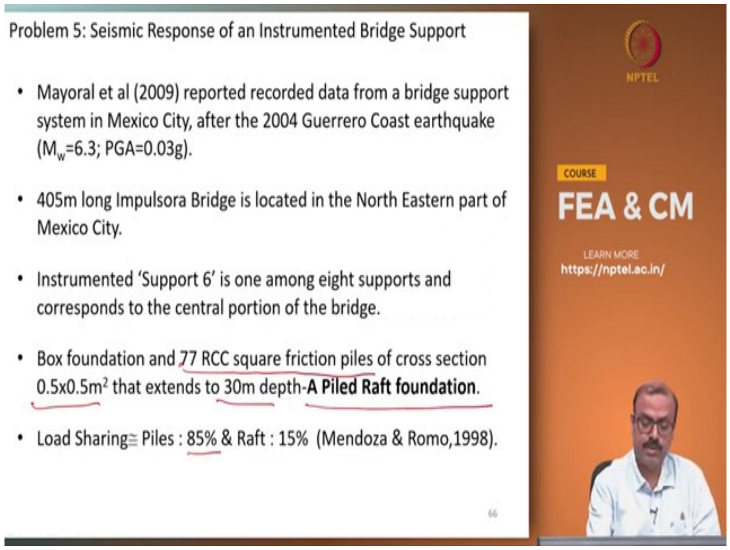
One of the major advantage of getting this results is shows that you can design the pile or structural design of the pile can be done based on this. So, it can be localizedly to avoid any kind of failure. We can flexural failure, we can have an additional steel.

(Refer Slide Time: 31:15)



So then we will go for the seismic analysis of our instrumented bridge support again this example of our 3D FEM.

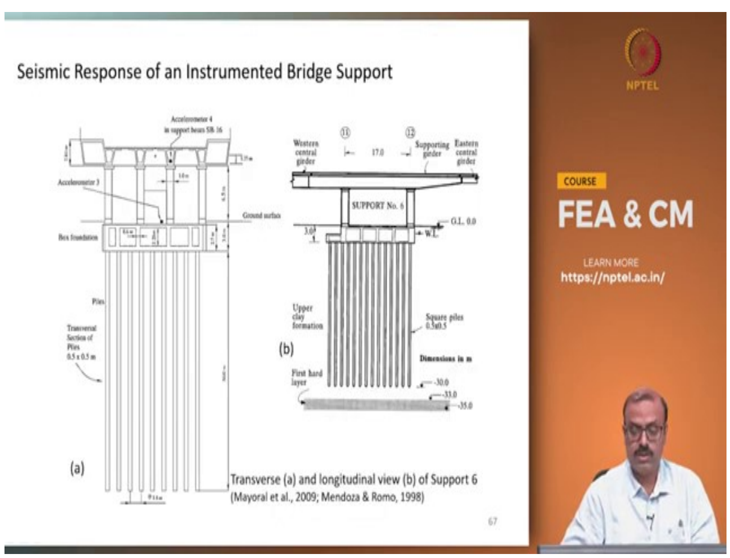
(Refer Slide Time: 31:23)



This is the last example problem, so, it is instrumented bridge support. So, it has been reported by a actual earthquake response, measured in Mexico City one of the overbroad over breach. And the construction happened just before the 2004 Guerrero Coast earthquake. So and the bridge luckily was the instrumented so, it is recorded some interesting responses.

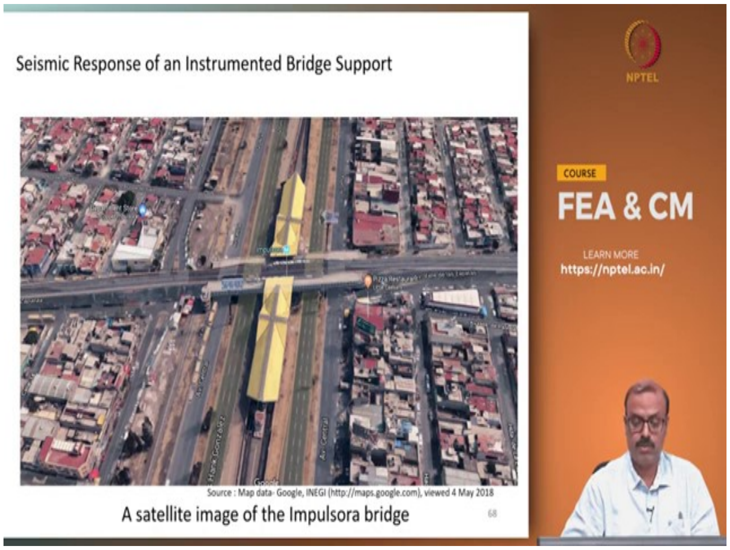
So, it is a 405 metre long Impulsora Bridge North Eastern part of the Mexico City. So, it is instrumented support 6 is the one that we will model. So, it is a box foundation and 77 RCC square friction piles of cross section 0.5 by 0.5 metre extended up to 30 metre depth. So, it is a predominantly it is a pile raft type of approach where piles are carrying 85 percent load raft is 15 percent.

(Refer Slide Time: 32:30)



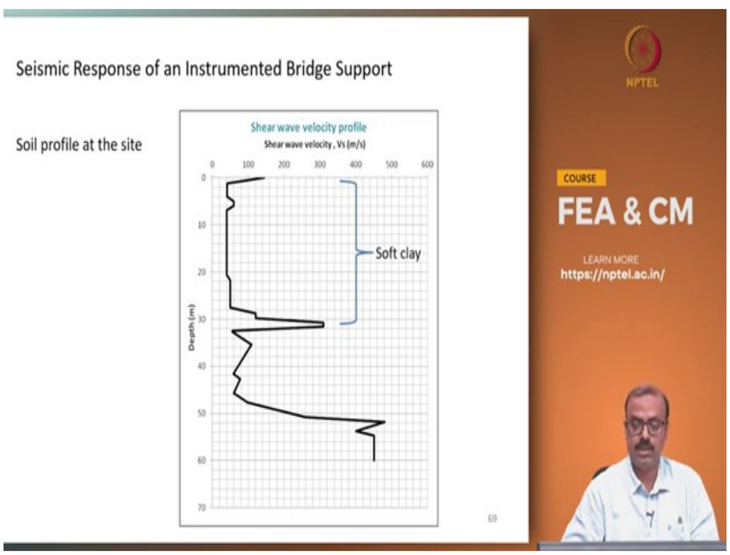
So, this is the typical diagram.

(Refer Slide Time: 32:33)



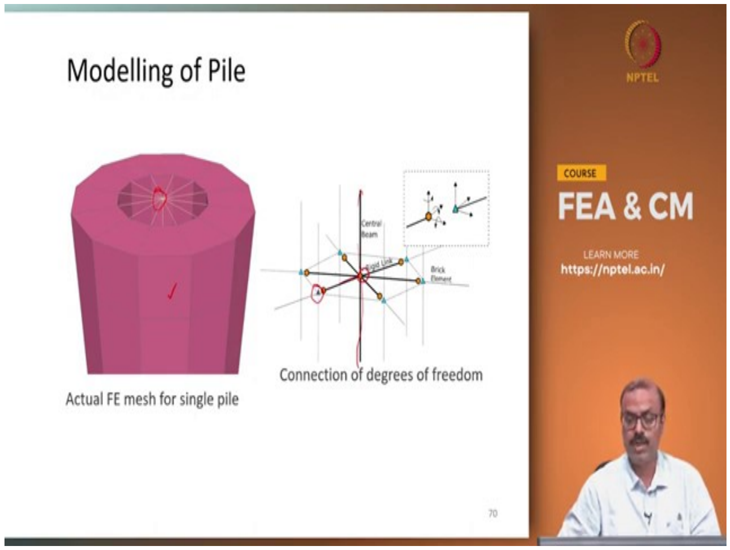
So, this is the Google Earth view of the Impulsora Bridge.

(Refer Slide Time: 32:40)



And shear wave velocity as we know in the Mexico City is a thick clear basin is there. So, top 30 metre is extremely soft clay.

(Refer Slide Time: 32:52)

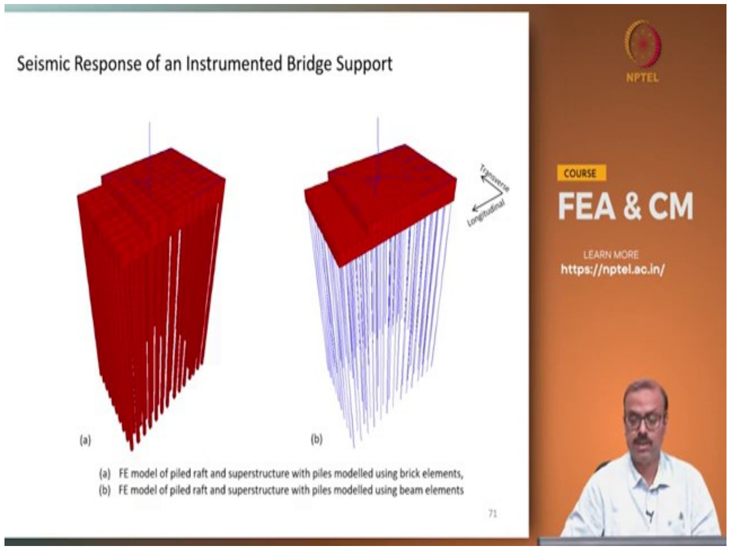


Now, the question about the modelling of the pile, as I said earlier that if you are fine with modelling the pile as structure linear elastic then there is no problem. However, if you want to have some kind of damage plasticity model type of thing, if you want to increase include into model the pile. To see is there any kind of damage happened. Then you cannot use that indirect method.

So, here what you can do is? You can come up with a kind of a hybrid type of finite, even modelling. So, here this is the continuum element. So, the each continuum element will be connected to a central beam. So, this is my central beam element to our rigid links again this rigid links can be modified later with like we can do a check whether it is rigid or flexible link is required or not.

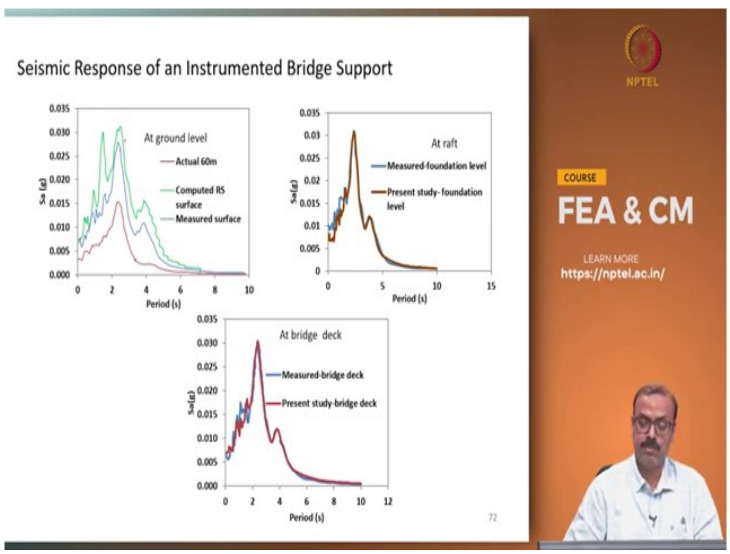
But for now, we have tried like so, there is this node is belongs to the continuum solid element and this node belongs to the central beam and this connected by the rigid link. So, this way we can have both continuum element as well as the beam without interfering each other properties.

(Refer Slide Time: 34:16)



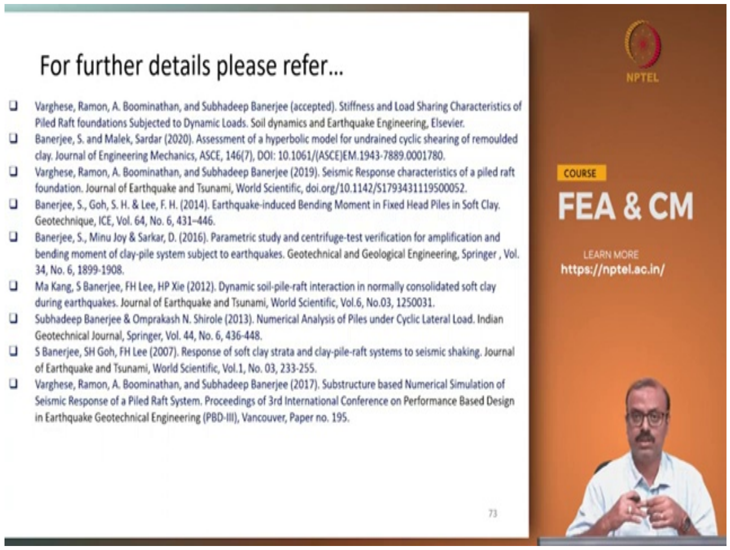
So, this is the way that bridge had been finite element model of the bridge peer structures as I said, it is support number 6.

(Refer Slide Time: 34:27)



So, as you can see that is a measured and the computed response at ground level. So, this is the actual at 60 metre. So, input so, this green and blue are the basically computed response spectrum at the surface and the measure at the surface level which is for all practical purpose. We can say it is pretty close the measurement and the computer results even closer at the raft level. We can see exact match even at the bridge tech level also, we can see exact match.

(Refer Slide Time: 35:04)



So, this is so, today's lecture we have seen the overall description of the finite elements usage of finite element softwares for a few geotechnical problems and how from plane strain to go for the complex seismic response with the different nuances of modelling from the meshing to the properties? And how to do the drain and undrain behaviour, where to use the pile, the regular pile elements or whether you can use a hybrid type of pile elements for different purpose.

For further details you may refer of these papers which have the detailed results of all these studies. Thank you.