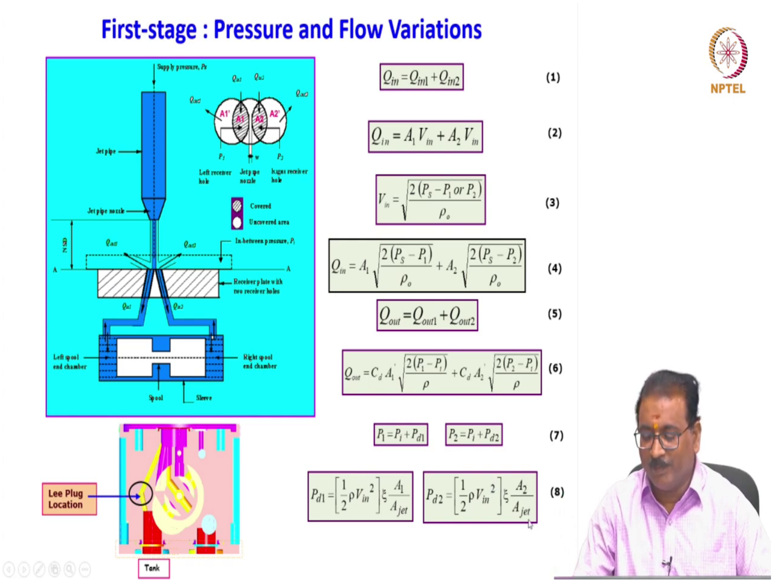
Oil Hydraulics and Pneumatics
Prof. Somashekhar S
Department of Mechanical Engineering
Indian Institute of Technology, Madras

Modeling and Simulation in Hydraulic Components
Lecture - 95
Part 2: First-stage: Pressure and flow variations, Parameter affecting the first-stage pressure recovery, Role of solid modelling and Finite element method

(Refer Slide Time: 00:22)



My name is Somashekar, course faculty for this course. Please see here friends in this figure I have shown the first stage pressure and flow variation. See figure here area modulation the left receiver, right receiver nozzle A1 is a covered area where the Q in 1 fluid will come. After pushing the spool it will go out through the opened orifice A1 dash, similarly in the right receiver Q in 2 is comes; the area A2 after pushing the spool it will come to A2 dash.

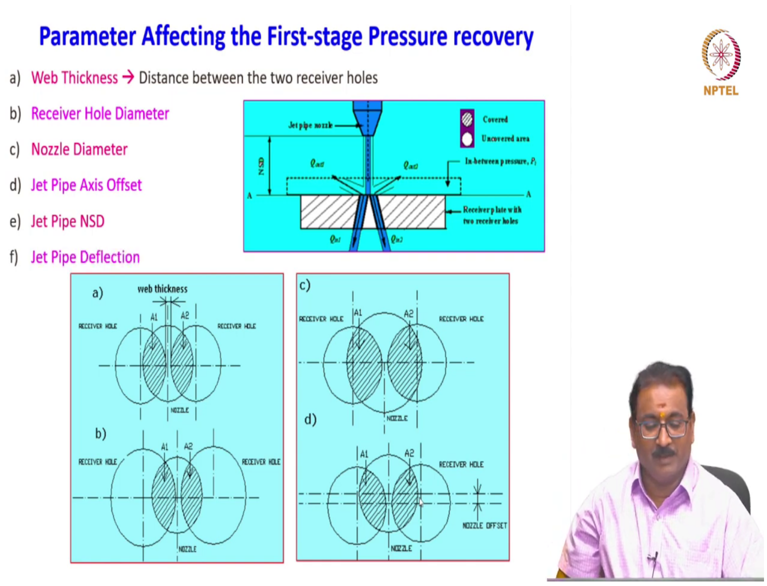
Please remember if you know very very carefully A1 A2 A1 dash A2 dash you are able to model the area modulation in terms of pressure modulation which is a very important input in the finite element model, I will show you later. Now, we will see how to calculate this all Q in equal to Q in 1 plus Q in 2, then how to calculate Q in 1? Q in 1 equal to using the pressure recovery principle A1 V in plus A2 V in. How to calculate V in? V in equal to square root of 2 P s minus P 1 or P 2 by rho naught; rho naught is the density of oil.

Q in equal to substitute these values here A1 root of 2 times P s minus P 1 by rho naught plus A2 into square root of 2 P s minus P 2 by rho naught. Q out opened orifice equation I am using. Here you will get C d A1 dash root of 2 P 1 minus P i very important P 1 minus P i; P i is a in between pressure.

Please note here friends after fluid is coming to the in between there is no passage always they will maintain the in between pressure to divert the complete flow to the tank through the lee plug; lee plug design is also very very important. Next, P 1 is P i plus P d1, P i is a in between pressure; P 2 equal to P i plus P d2; P d1 and P d2 are the dynamic pressure when the jet pipe will move over the receivers, which is given by 1 by 2 rho V in square into zeta A1 by A jet.

Similarly P d2 equal to 1 by 2 rho V in square zeta into A2 by A jet, zeta is a distributor loss coefficient. Generally they are taken these value is above 0.98 to reduce a leakage effect on this. After knowing all these equation only parameter you will see I already know the supply pressure and areas. Substitute all the values you will get the dynamic pressure and other pressures which are very essential to input in the finite element modeling.

(Refer Slide Time: 03:51)



Now, we will see the important parameter affecting to the pressure recovery or web thickness. Web thickness is the distance between the 2 receiver holes, you will see the 2 receiver holes are drilled in the receiver plate by the manufacturer. That time this distance between these 2 hole and alignment should be very very important.

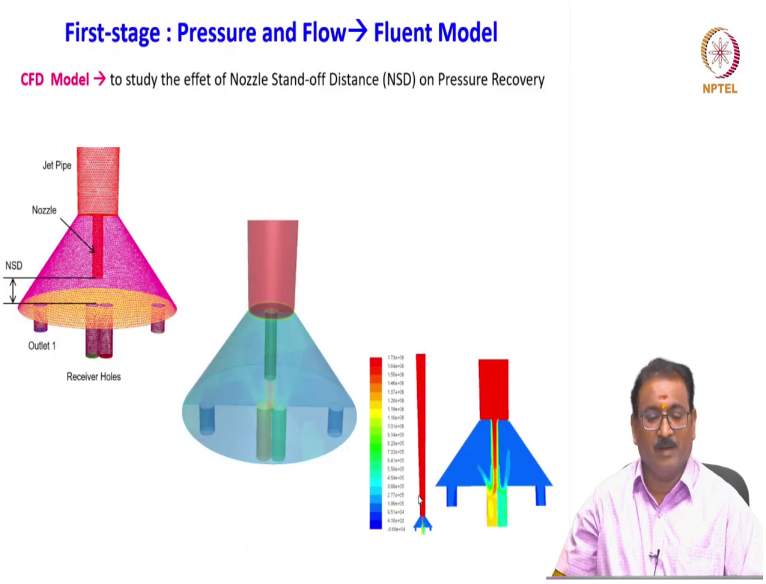
For example, if the web thickness is as close as possible for example, 0.001 very good for pressure recovery because this area will be more, but generally they will vary this 0.01 mm to 0.02, 0.03 mm, beyond that the pressure recovery may not be there in the receiver holes.

Then, complete dynamics of the jet pipes or valve will be get affected, but you have to study these parameter in the modeling. Similarly, next you will see receiver hole diameter these 2 receiver hole diameter. Generally they are the 0.3 mm if it is a 0.38 mm what happens? If it is a 0.2 mm what happens? These diameters are very very important in the pressure recovery you have to vary these parameter study the sensitivity of these parameter on the pressure recovery.

Similarly, nozzle diameter again receiver diameter and nozzle diameter generally they will keep it as a same, if they will vary drastically what is a dynamics we have to study. Then jet pipe axis, you will see here jet pipe axis is also very very important friends. Because you will see here the torque motor is manufactured separately.

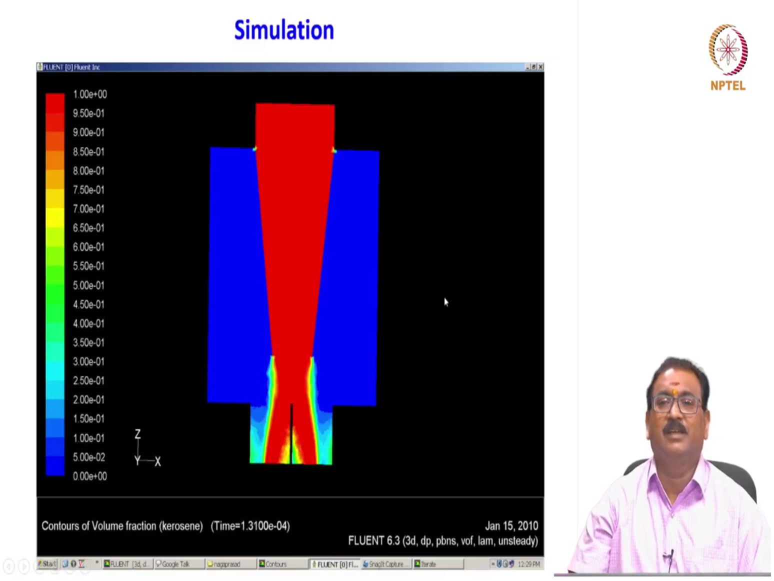
Valve body is manufactured separately later they will assemble over the valve assembly. That time if the jet pipe is accept that time pressure recovery get affected you will see the areas then you will study all these parameters. Next I will show you one more very important thing is jet pipe nozzle stand of distance, the receiver plate and jet pipe. How much we have to hold this? We shall we keep it away or shall we bring it near, to study these we developed then jet pipe deflection is also the important parameter.

(Refer Slide Time: 06:22)



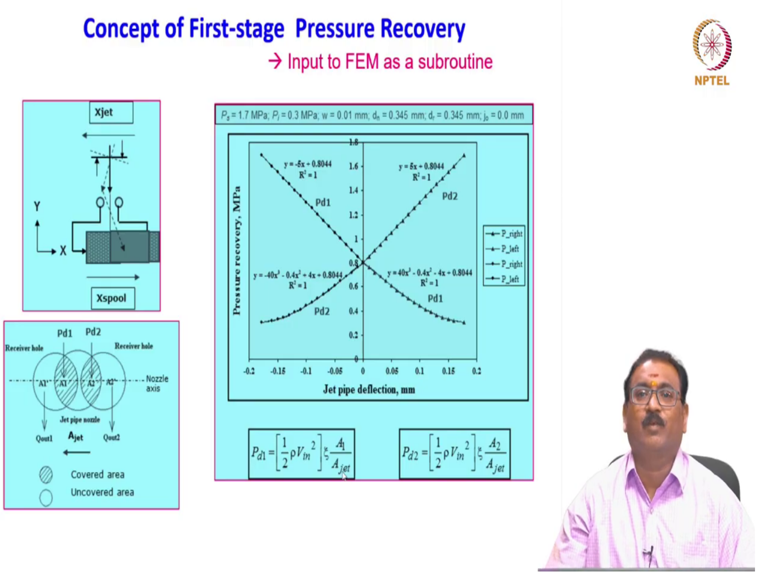
Now, we will see this CFD model is developed to study the effect of nozzle standoff distance see here from here to here how much? 1 d; 1 d means one times the diameter of the nozzle or two times the diameter of the nozzle, three times the diameter of the which are goes on increasing the nozzle standoff distance see here. Then we are studying the effect on the pressure recovery.

(Refer Slide Time: 06:50)



You will see here how it is this is the only way possible to study the nozzle standoff distance in the CFD simulations.

(Refer Slide Time: 07:00)



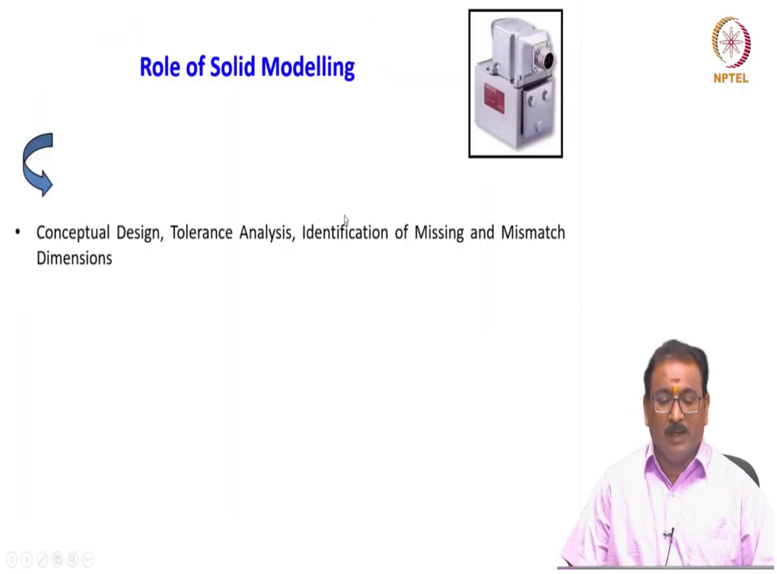
Now, quickly I will come to the concept of first stage pressure recovery. As I have told you this is the area modulation correct? Now, I am showing you here the pressure recovery versus jet pipe deflection using these equations. Every parameters I know I want to know A1; A1 is a area here A2 is a area here how to calculate using your solid model or your simple mathematical equation possible to calculate these. These things you have to input here, rho is a density of fluid correct.

A jet is the diameter of the jet pipe pi by 4 you know d n square. Diameter of the nozzle is 0.3 as I have told you here some of the parameter considered further analysis is as shown here. Whole objective is here when the jet pipe is at the center positions the equal amount of oil will go here.

P d1 and P d2 equal that is I am getting the 0.8 MPa or 8 bar when the jet pipe is deflecting in the positive directions jet pipe will move here you will see here one side pressure goes on increasing, other side pressure goes on decreasing proportionately. Similarly, when the jet pipe will deflect in the other direction negative direction due to the current polarity in the torque motor.

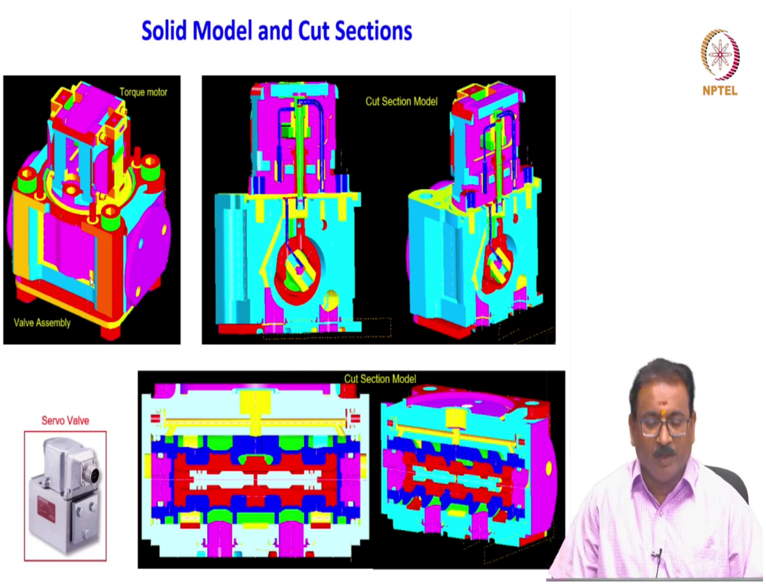
The same pressure you will see goes on increasing and another pressure is goes on decreasing. You please observe here friends small non-linearity is there this is due to the web thickness provided between the 2 receiver holes.

(Refer Slide Time: 08:43)



Now, we will see the role of solid modeling which helps you conceptual design tolerance analysis identification of missing and miss match dimension.

(Refer Slide Time: 08:56)

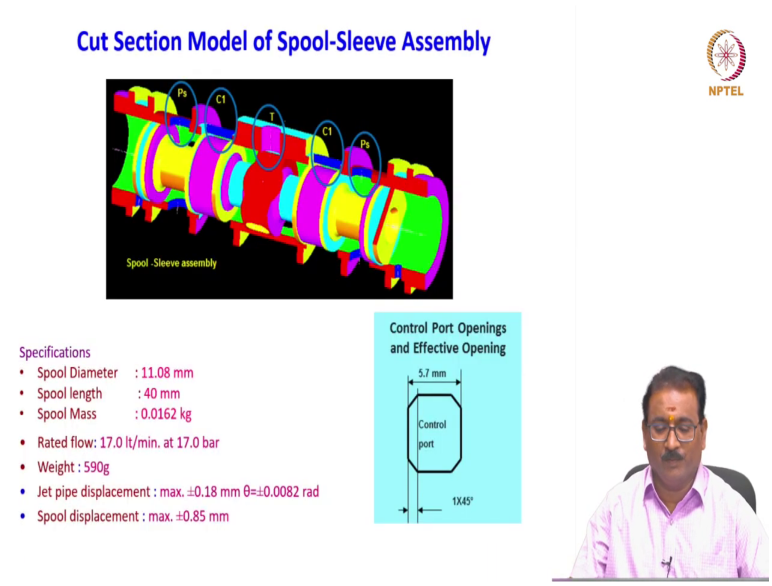


Now, I will show you here the complete solid modeling of torque motor parts as well as a valve parts. In between there are the so many precise delicate components are there. Similarly, they are having the various cavities cut in the valve body. The beauty of solid modeling is as we know all the parts are designed separately and then we are assembling.

Once the solid model is ready friends here you will see you will cut the model in any direction to see the alignment very very important. You will see now here torque motor jet pipe how it is coming over feedback spring position you will see and then I told know lee plug design. How it is diverting the flow everything you know supply pressure, how it is going everything you will see once the solid model is properly designed and drawn.

Cut the model and see the spool whether ports are blocked or not everything you will see this is the beauty of this. Once the solid model is ready friends the same thing I am using in the finite element model that is the beauty how to do it? But, first and foremost thing is you have to model this all the component hydraulic component and solid model check it all the dimensionality tolerance analysis interferences all then you will use for the finite element after knowing the theoretical analysis.

(Refer Slide Time: 10:34)



Let us I will show you now we will see the clearances whether we are given clearances ports C1 C2 tank port is middle supply port at the end whether they are properly assembled or not possible to see here because directly. I am using this thing in the finite element model.

(Refer Slide Time: 10:54)

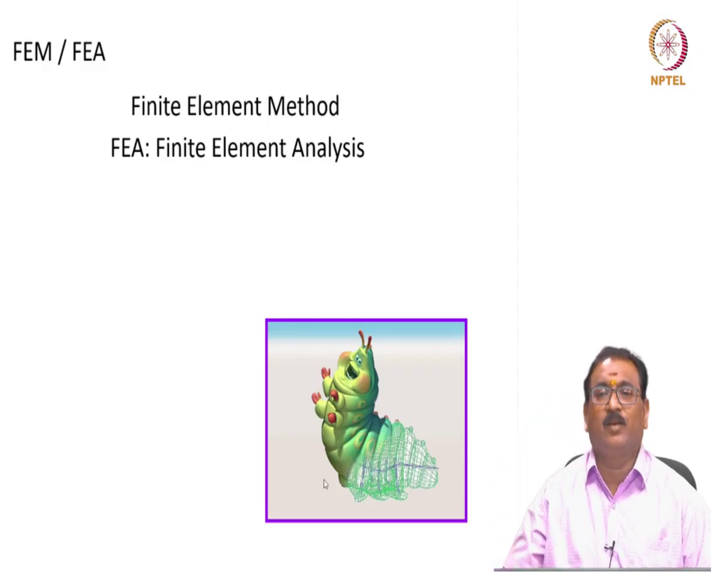
Main Valve Specifications

No	Parameters	Value
1	System pressure	1.7 MPa
2	Tank pressure	0.1 MPa
3	Max. jet pipe deflection	0.2 mm
4	Max. spool displacement	0.85 mm
5	No load flow rate	17 lt/min. for 1.7 MPa pressure drop
6	Spool diameter and length	11 mm and 40 mm
7	Jet and receiver diameters	0.3 mm
8	Piston diameter	54 mm
9	Piston stroke length	100 mm
10	Working fluid	Aviation kerosene

These are the wall specification system pressure, tank pressure maximum jet pipe deflection is 0.2 mm to get the maximum pressure recovery. The maximum spool displacement is 0.85 mm no load flow one I am showing you 17 liters per minute for 1.7 MPa pressure drop. 17 bar pressure drop or 1.7 MPa pressure drop.

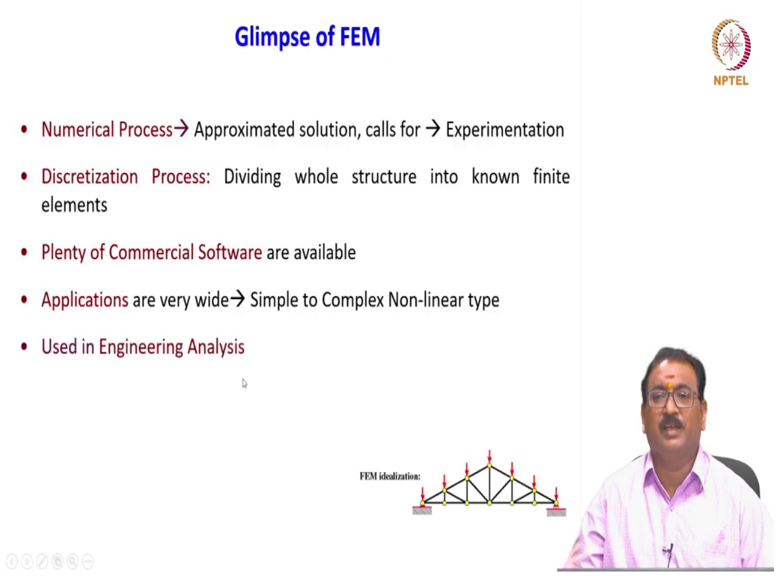
Spool diameter is 11 mm and length is 40 mm jet and receiver diameter you will see 0.3 and piston diameter 54, piston stroke length is 100 mm working fluid aviation kerosene. Once your model is ready finite element everything changing these parameter is very easy friends.

(Refer Slide Time: 11:46)



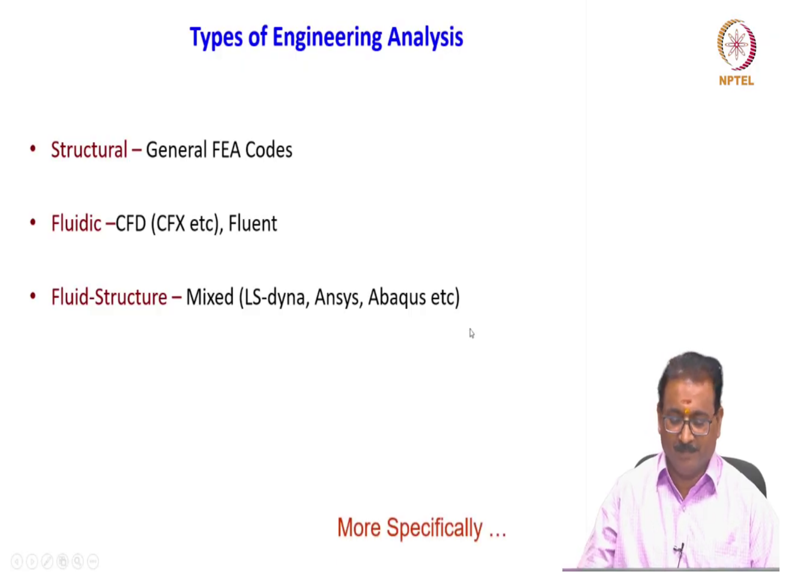
Quickly I will show you the finite element modeling or a finite element analysis already you know this very quickly I will show you in one slide to motivate you how to use this in the hydraulic component analysis very quickly.

(Refer Slide Time: 12:00)



It is a numerical process friends approximated solution calls for experimentation always. Discretization process it is friends, dividing the whole structure into known finite element. Please remember that is a beauty in finite element it will take actual geometry and then discretize and then you apply the boundary condition loading all these thing to study this is the beauty here. The plenty of commercial software's are available.

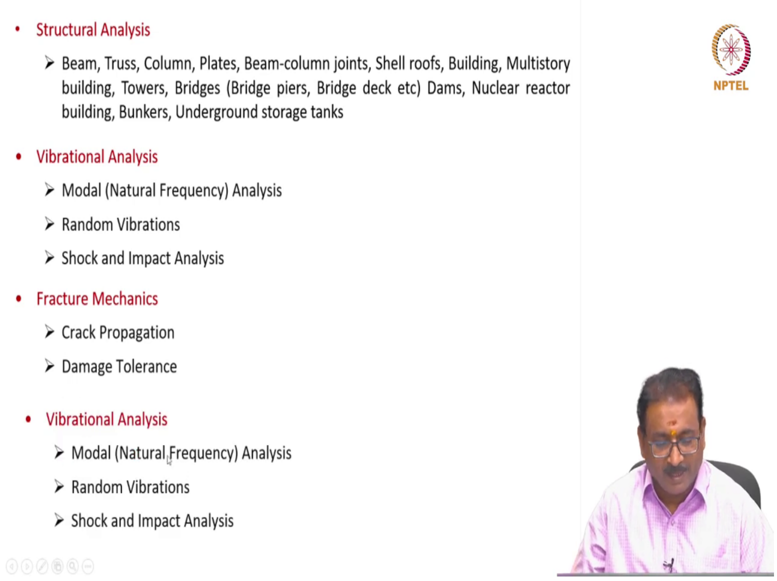
(Refer Slide Time: 12:39)



Applications are very wide simple to complex non-linear analysis used in engineering analysis. Quickly already we know that so many engineering analysis can be solved using the finite element method quickly I will show you structural analysis general FEA codes are generally people are used.

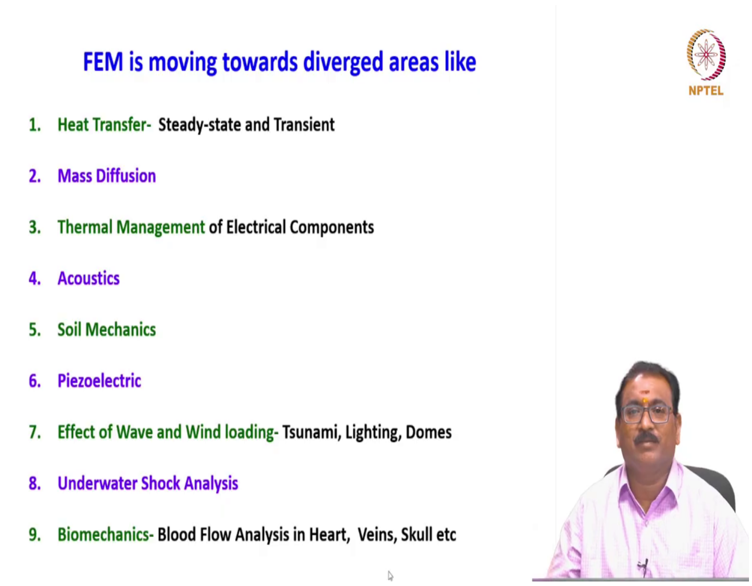
Fluidic here there are CFD CFX, fluent many software's people are using fluidic. Here fluid is meshing here structure is meshing the fluid structure interaction. What I will call the mixed analysis; the limited software's are available to solve the fluid structure interaction. Also a very limited functions like LS dyna, Ansys, Abaqus etcetera. They fail to do the powerful closed loop simulations.

(Refer Slide Time: 13:31)



More specifically structural analysis includes all these things you know already this, then vibration analysis includes so many sort of analysis people are doing fracture mechanics many crack propagation, damage tolerance analysis, vibration analysis.

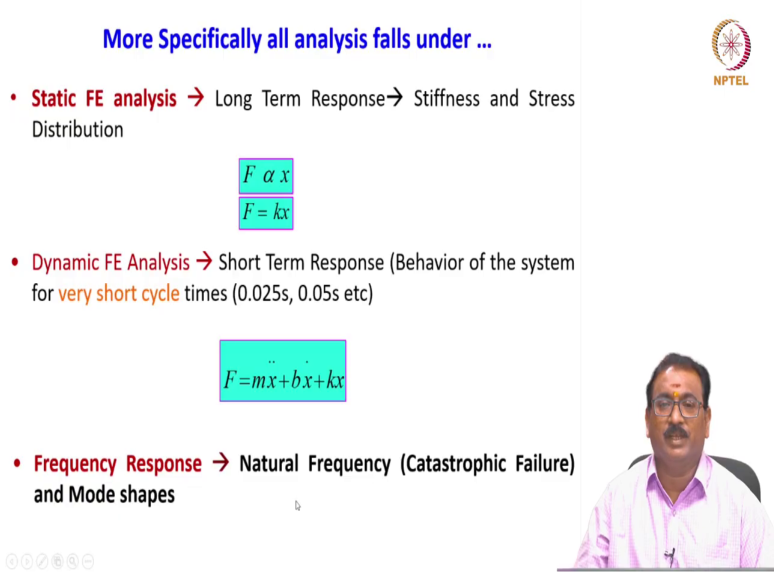
(Refer Slide Time: 13:50)



You will see this finite element is making the large application in the civil engineering, mechanical engineering, biomedical engineering, electrical engineering and much more. So, FEM is moving towards now diverged areas like a heat transfer, mass diffusion, thermal management, acoustics, solid mechanics, piezoelectric, effect of wave and wind loading- tsunami, lighting dome. How they will subject for the wind loading underwater shock analysis.

Biomedical also now people are using the fluid flow analysis in the hearts, veins, skulls ok. Then the you know teeth analysis many where finite element playing a major role.

(Refer Slide Time: 14:43)

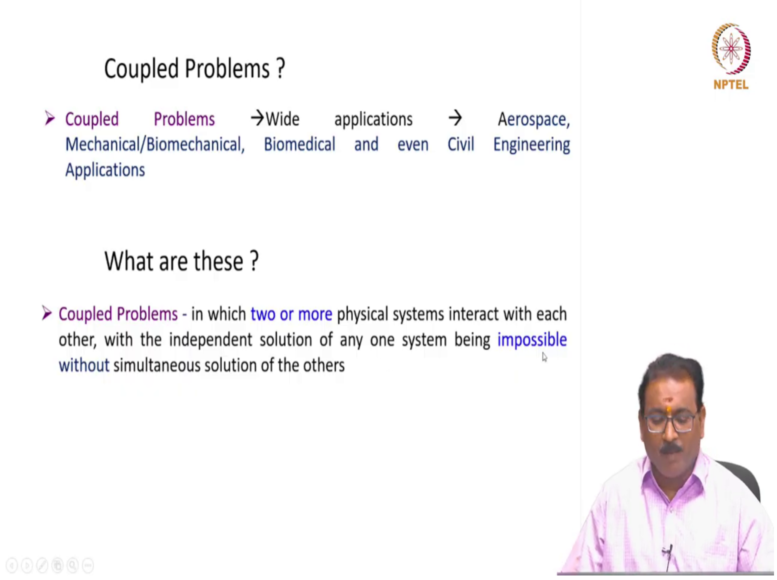


More specifically static analysis already we know that for the long term response here stiffness and stress distributions are the main motto. Here, F is proportional to x F equal to kx, k is the stiffness F by x very simple it is here it will not considered any density, mass nothing.

Now, we will see the dynamic analysis for the short term responses, correct. That is a very short cycle times like a 0.025 seconds to 0.05 seconds. Here it will solve the second order equation F equal to mx double dot plus bx dot plus kx. It is complicated as compared to static analysis.

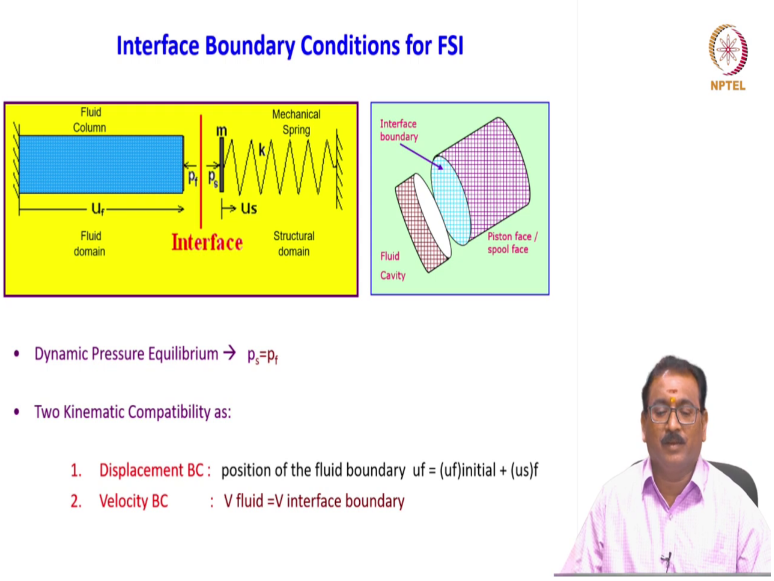
Each is having own advantage and disadvantage when I want to know the shorter responses dynamic analysis is best. I want to know the long term responses that time a static FE analysis is good. Next one is a frequency response to for predicting the natural frequencies which is very essential to know the catastrophic failure of the critical components and a mode shapes.

(Refer Slide Time: 15:58)



Coupled problems also known as mixed problems. Coupled problems the wide applications in aerospace, mechanical and biomechanical, biomedical and even civil engineering applications. What are these? Coupled problems in which two or more physical systems interact with each other. With the independent solution of any one of the system being impossible without simultaneous solution of the other.

(Refer Slide Time: 16:34)



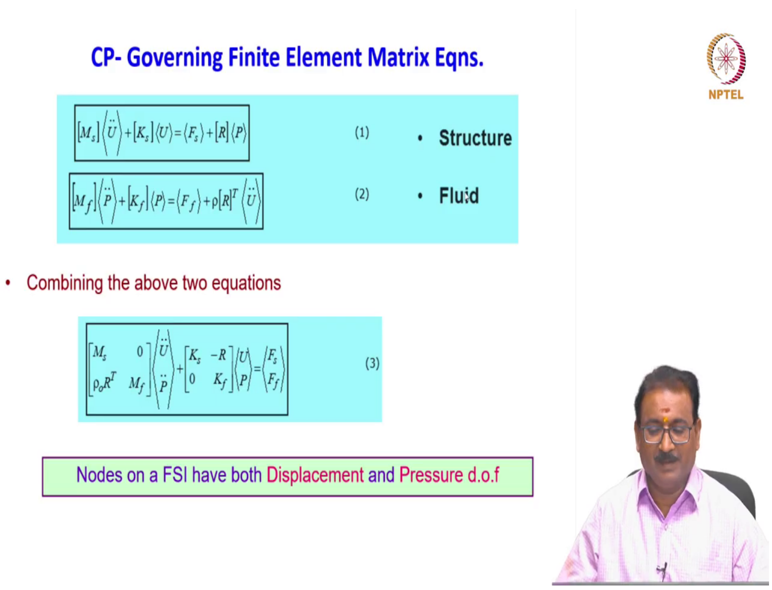
Now, we will see here I have considered the fluid is one side fluid column and another side is a spring in between these two here it is what I will call the spring mass correct here. The interface boundary condition is very important which establishes the interaction between the fluid and the structure at the mechanical spring. Here very very important thing is you have to establish the interface boundary conditions for the interface modeling for the fluid as well as the structure.

As because when the structure will apply the load on it will move again it will in turn it will move the load on it meaning interface boundary conditions are very very important. What we have to do friends for this? If this is the fluid domain it is a structural domain for this dynamic pressure equilibrium should be there on both side that is why I am telling you p s equal to p f dynamic pressure equilibrium. Similarly, 2 kinematic compatibilities are very very important to ensure the fluid structure interaction.

Displacement boundary conditions, what is this position of the fluid boundary uf is equal to uf initial plus us final. Similarly the velocity boundary condition is equal to v fluid equal to v interface boundary condition. If you are able to solve these two boundary condition at the interface you are establishing the coupled problems.

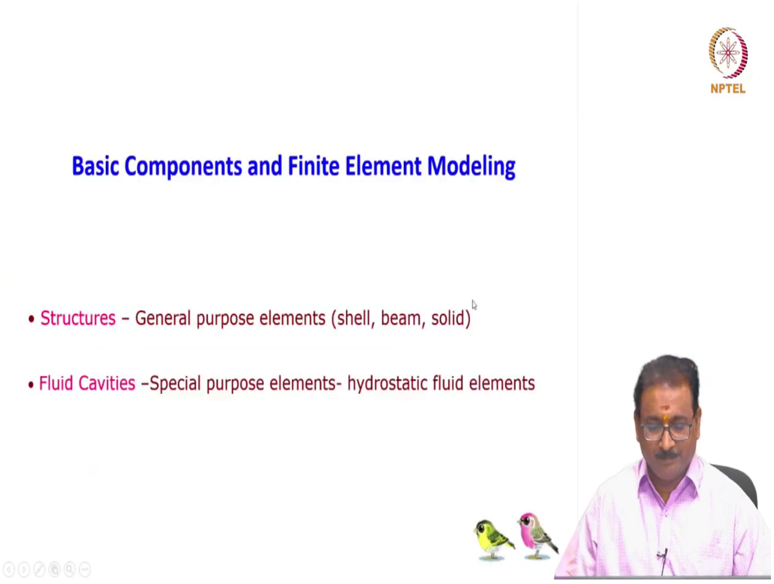
How to do it? You will see here this is the fluid cavities as I have told you surface, correct surface elements it is. Here it is what you will call it is a piston and cylinder what when I will model. Then you will see here friend these are the elements and nodes present here. These elements and nodes should be common to fluid as well as the structure you will make such a compatibility then only you are ensuring the fluid structure interactions.

(Refer Slide Time: 18:58)



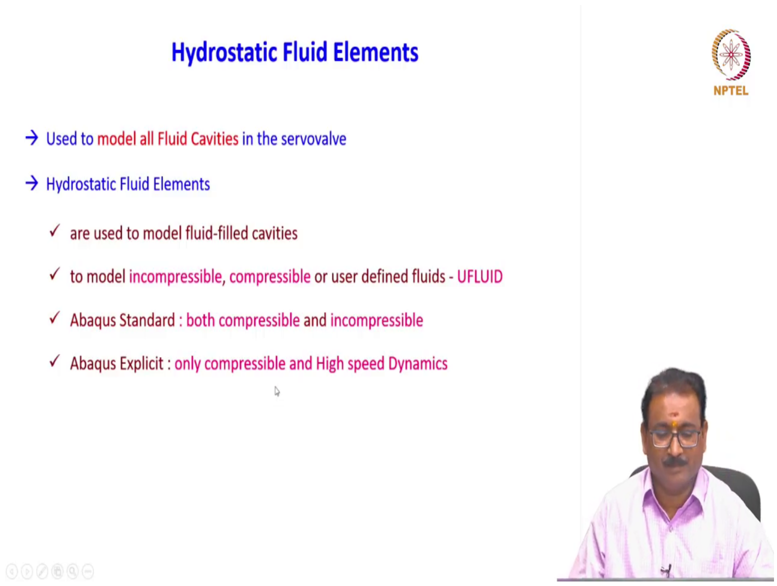
Now, we will see the structural equation set of equation fluidic interactions combining both it will give you the coupled matrix. Here you will see R is a coupling matrix which includes both structure as well as a fluid. Please take care here nodes on the FSI have both displacement and a pressure degree of freedom.

(Refer Slide Time: 19:28)



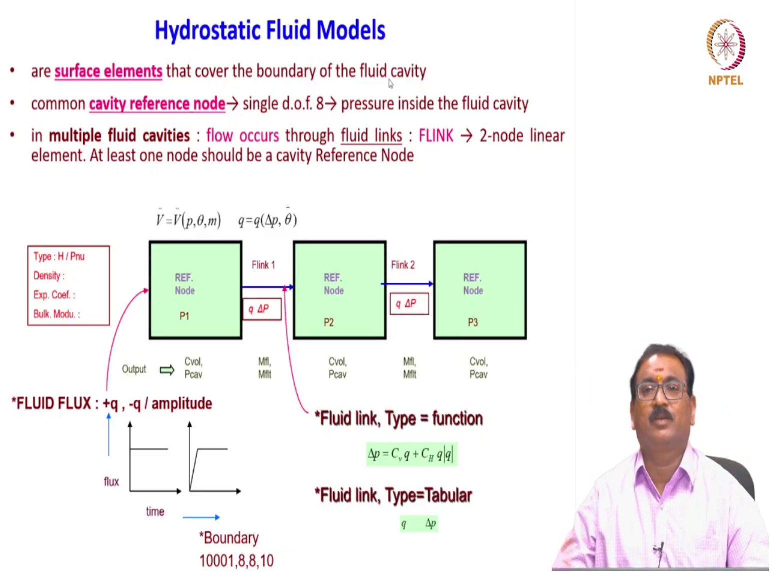
Now, quickly I will show you basic components and finite element modeling. Here all the basic structural elements are meshed with the basic purpose elements like a shell, beam, solid based on the action during the operation. Fluid cavities all you know left side cavity, right side cavity, spool left cavity, right cavity, actuator, left cavity, right cavity. All are created using the spatial purpose element hydrostatic fluid elements.

(Refer Slide Time: 20:13)



What are these hydrostatic fluid element? Used to model all fluid cavities in the servo valve, hydrostatic fluid elements are used to model fluid filled cavities to model incompressible, compressible or user defined fluid also you have to do it in the abaqus UFLUID. There are 2 things in abaqus; abaqus standard and abaqus explicit. Abaqus standard you may model both compressible and incompressible, meaning oil as well as a air you have to model, but an abaqus explicit only for compressible and high speed dynamics.

(Refer Slide Time: 21:03)

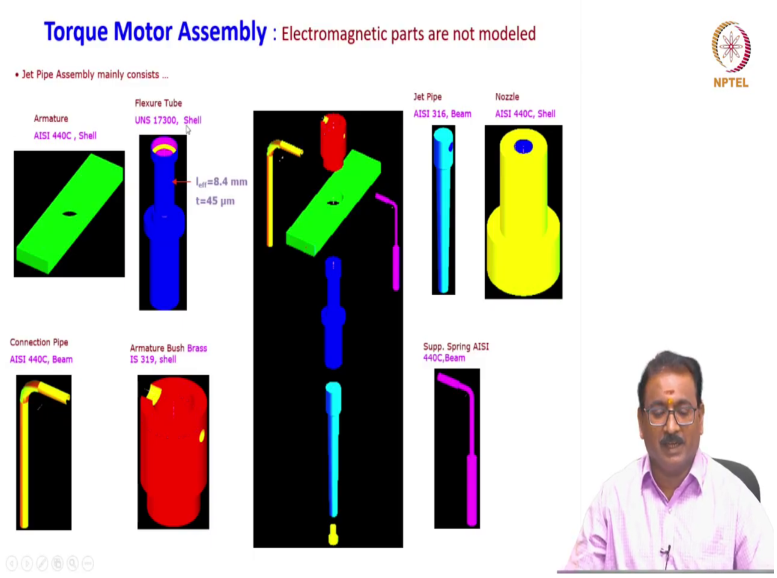


Now, will see very quickly I will tell you to understand this how fluid will takes place when the jet pipe will move spool will move. Hydrostatic fluid elements or a surface elements that cover the boundaries of the fluid cavities. If you are taking the piston and cylinder piston side left cavity all boundaries are covered with the hydrostatic fluid elements. The common cavity reference node is there which has a single degree of freedom pressure inside the fluid cavity you have to assign to this.

In a multiple fluid cavities flow occurs through the fluid links FLINK, you will see this is the Flink 1 Flink 2. What is this? It is nothing, but the P fluid will takes place from one cavity to another cavity how it is due to the varying the delta P. Then you have to define q versus delta P q is a flow rate delta P is the pressure.

This you have to input in terms of the function after getting how fluid varies with the delta P or else you have to type it is a tabular form you adopt any method. Please remember friends here the fluid flux q plus minus q minus you will vary like any conditions; stepped input variable input whatever you will give, also we will define all the fluid property hydraulics or pneumatics density, expansion coefficient, bulk modulus all you will define for each cavities.

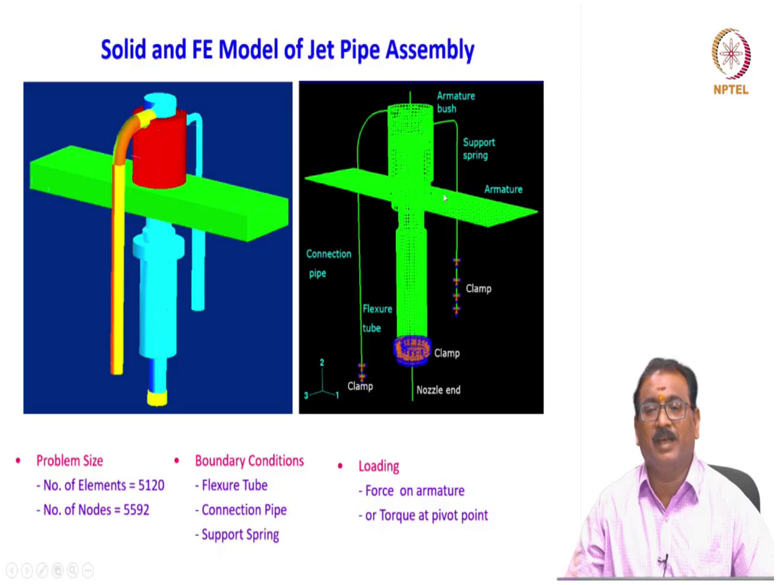
(Refer Slide Time: 22:44)



Now, quickly I will show you the finite element modeling of some of the critical part like a torque motor. Here for easy understanding now I have not considered the electromagnetic parts coil gial. I have not considered here, only thing I will considered all the parts like a armature.

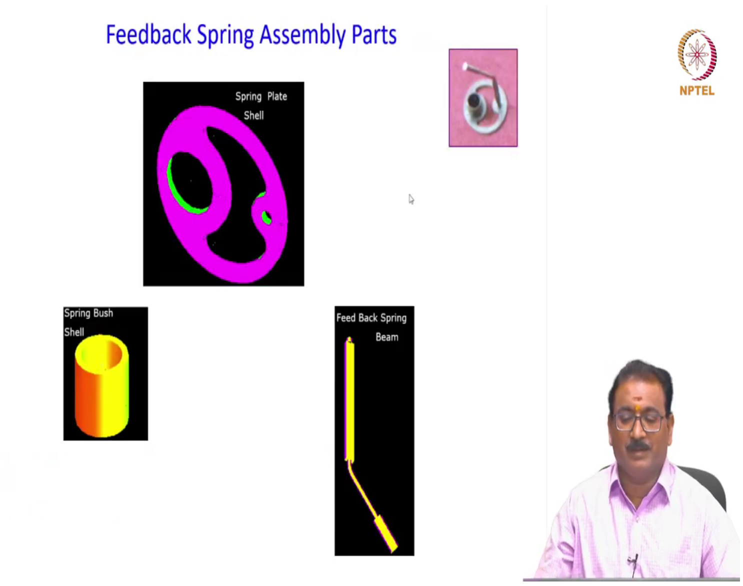
See here armature; armature bush flexural tube which will give the rotational stiffness then this is a jet pipe nozzle and this is a pipe where the fluid will come continuously. And this is to move in the one plane is to deflect you know only one plane. All these parts are modeled with a suitable materials and elements.

(Refer Slide Time: 23:34)



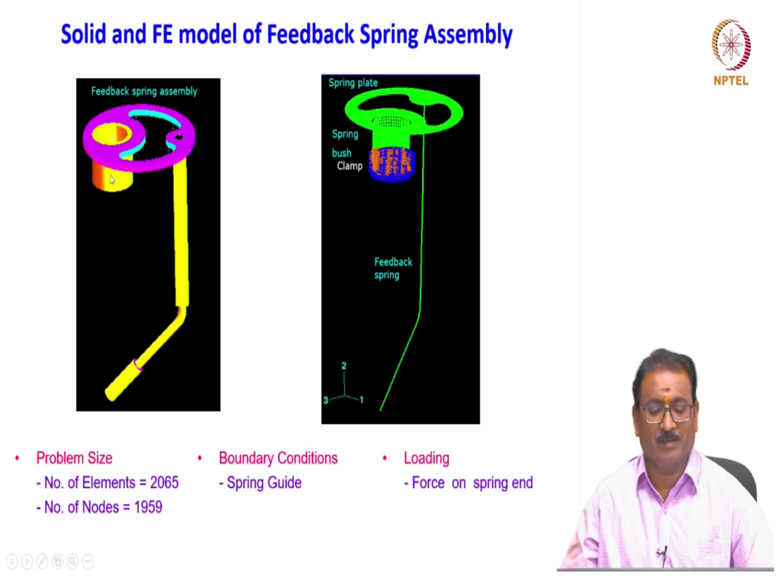
Now, we will see the one to one relationship of the solid model the beauty of solid model once you will do this and you will model the FE models.

(Refer Slide Time: 23:45)



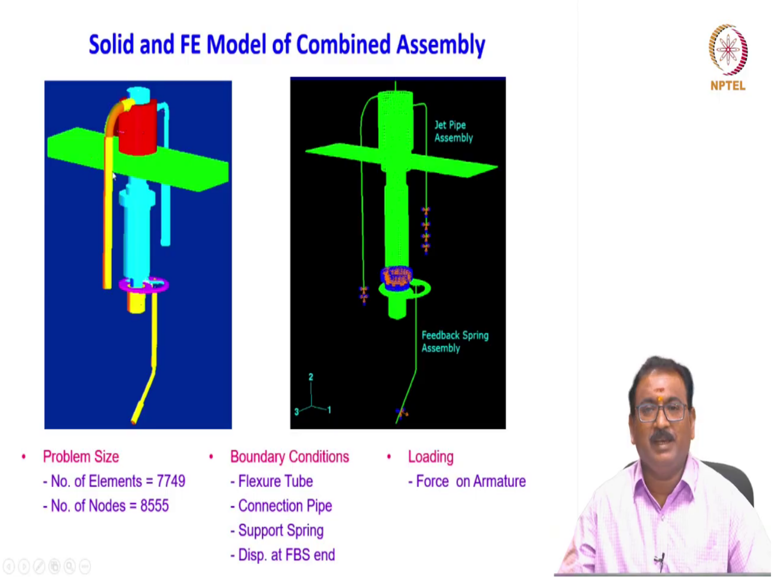
Similarly, I will show you one more part first feedback spring assembly. You will see the feedback spring assembly mainly consists of 3 parts one is a spring plate, see the dimensions and geometry, how it is. Then spring bush it will go and fit here and feedback spring it will go and sit here. This angle also subjected for the various types of analysis by varying the theta plus or minus 3 4 degrees. Then you will see here the this is the feedback spring which will stabilize the valve operations.

(Refer Slide Time: 24:22)



Now, we will see the proper boundary condition same it is, correct? Feedback spring finite element model, number of elements, number of nodes created, boundary condition loads on the feedback spring with the proper boundary condition, arrested in the proper place.

(Refer Slide Time: 24:42)

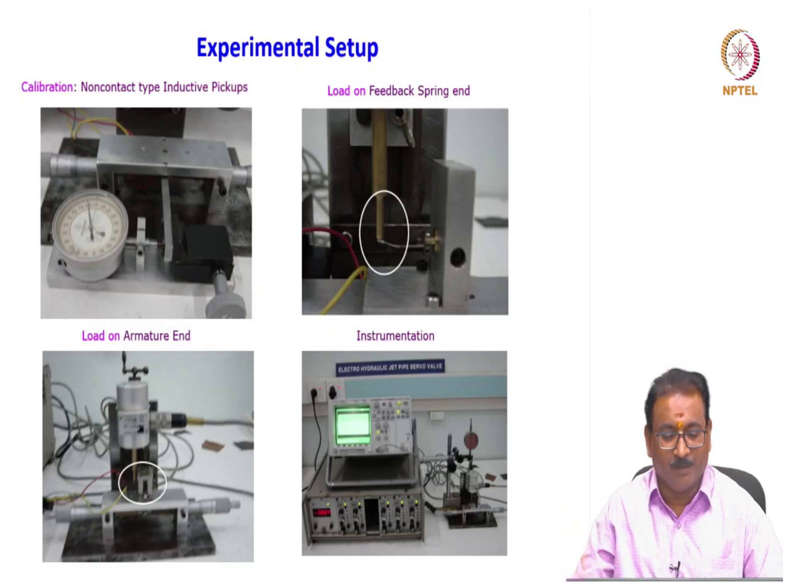


Now, we will see what we did the torque motor at the feedback spring assembly one to one relations all friends.

(Refer Slide Time: 24:53)

Then what for we are using this? We have to use for finding the stiffness of the component. Later; what we did is we have to create at the test rig to apply the force and proper boundary condition on the realistic model.

(Refer Slide Time: 25:10)



Now, we will see here friends this is a calibration of non contact type inductive pickups from the here jet pipe will enter. Jet pipe is deflected in the air you should not touch to model that we have used the non contact inductive pickups down. Then applying the force on the feedback spring you will see we used the German load cell of 0.5 kg.

What we did here friend is, we are goes on applying the load on the feedback spring we captured all these thing in the CFA carrier frequency amplifier, then what we did? Then we are goes on releasing in the same to find out the linearity of the spring on the both direction.

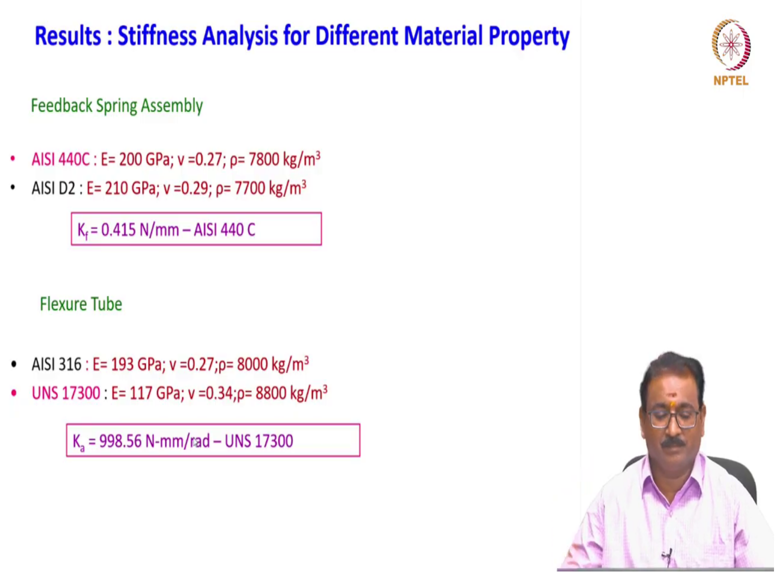
Similarly, you will see here this is the feedback spring here jet pipe middle you will see jet pipe. Then same way we are applying the load on one side to create the torque.

Then we are studying the deflection of the jet pipe this is whole objective is to find out the rotational stiffness of the flexural tube after assembly. These are the some of the instrumentation part correct to give the a slow scope it is to give the sinusoidal input all the things we meet.

(Refer Slide Time: 26:33)

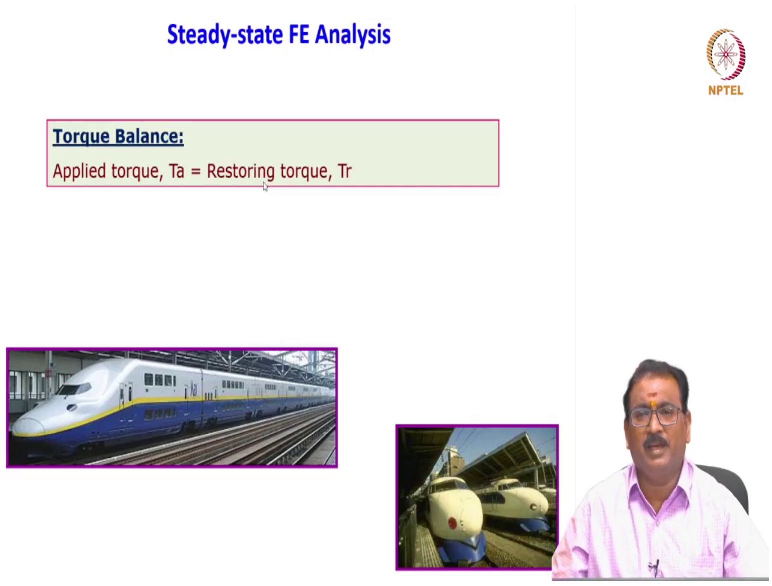
Now, we will see the this is the brass knob to apply the load on the armature, you will see the different types of modification in the mandrel to hold the feedback spring.

(Refer Slide Time: 26:45)



These are the results you will see for the AISI 440C, what are the stiffness AISI D2 all are the spring steels, what are the values we are getting. Similarly, for the flexural tube AISI 316 and UNS 17300, all are having the rotational stiffness. These are very very essential to predict using finite element model very quickly.

(Refer Slide Time: 27:16)



Now, I will quickly I will show you friends steady state finite element analysis. What is the steady state? I told you in the last class also apply the torque on the armature is equal to restoring torque from the spool movement to bring back the jet pipe to the null position. This is a very very important for any current input jet pipe will deflect due to the spool movement again it will come back to the null position. This is a steady state it should always establish in the servo valve.