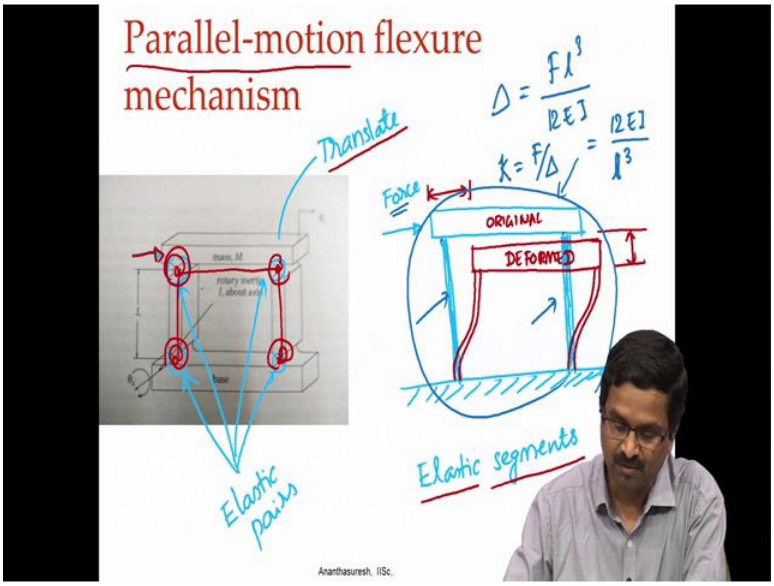
Compliant Mechanisms: Principles and Design
Prof. G.K. Ananthasuresh
Department of Mechanical Engineering
Indian Institute of Science, Bangalore

Lecture - 09
Linear finite element analysis of compliant mechanisms with beam elements

Hello in the last 2 lectures we discussed elastic pairs that is narrow sections that connect to relative rigid segments in a complaint mechanism and well that they can be modeled using empirical formula, that many people have derived starting from 1965 when parallels invoice bar did it to even now people are trying to improve them and trying to find the ranges for which they are accurate and so forth, but instead of doing all that today 1 can use finite element analysis and try to get the simulated response.

But as we discussed finite element is not so useful if you are trying to design that is you do not have a mechanism now you would like to have 1 based on some specification right. So, let us understand the difference between what finite element analysis can do and we what it cannot do. So, easily today in this lecture we are going to discuss actually finite element analysis for modeling not elastic pairs although we can do that also to have the small length flexure provide or the circular not hinge all of those can be modeled using finite element analysis, but we focus on elastic segment such as long beams or slender beams.

(Refer Slide Time: 01:58)



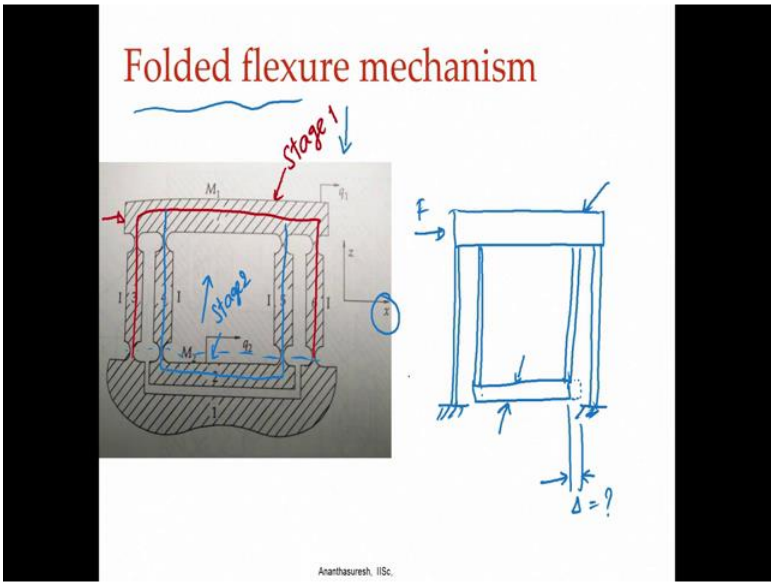
So, we will have mat lab code also run. So, we can see how we can analyze complaint mechanisms that have elastic segments. So, let us look at this slide and see that we have this parallel motion flexure mechanism that we had discussed with the last couple of lectures. So, once again we have this flexures over here let us get the thing we have this flexures here four joints right. So, here if I apply a force in that direction this block at the top is going to only translate. So, this 1 will translate if I apply force in that in this direction it is going to translate in that direction, but there will also be a little bit of motion in the perpendicular direction, but no rotation.

So, this 1 we can also do like this it is a 2 d 1. So, I will just plot the 2 d mechanism. So, we have the base this is the base. So, instead of using this what we called elastic pairs we have 4 of them in this what we called elastic pairs, instead of using them now will use elastic segments or beams slender beams just slender because their length is much longer then there cross section dimensions, which we can just show like a lines I can show this thin lines they both can be identical. So, let me just fill it they can be strips of metal or plastic or whatever and it could be (Refer Time: 03:49) silicon as well, when you have such a thing when we apply a force here this is going to deform let us try to plot that how does it deform, I said that its going to only translate there is no rotation let us assume that this has come down its kind of any exaggeration it has come down and moved.

So, there is some displacement in this direction we also there is some downward proof and that is what it will do it will come down as it translates to the right because these beams are going to deform like this of course, whatever I am drawing is exaggeration will do finite element analysis to see how exactly it is going to look or if you have a real device of course, you can see this is parallel motion because you have the original 1 and deformed here are parallel to each other and this is used in many precision motion systems as we discussed in the last 2 lectures.

Now, we want to analyze them we can do that analytically. So, elastic pairs when you do it everywhere you will have this kinematic joints and then we got equivalent torsional spring constant over here for all four of them and then we have the rigid bars connecting them and we can do kinematics and force equilibrium to see for a given force how it deforms, that is 1 way to do it will be do in that later 1, but today will try to do when we have this elastic segments rather than elastic pairs. So, how do we analyze this?

(Refer Slide Time: 05:53)



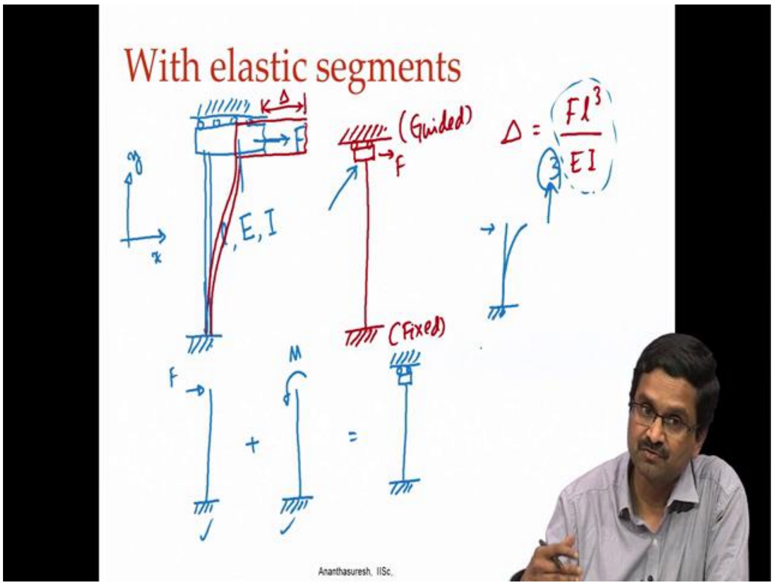
We can also look at this folded flexure mechanism again if you see we had discussed this when you have the top 1 that is this one we just saw that this block not only moves to the right when apply a force over here, but also comes down little bit right.

But if you do not want that you want the stage if I call this the moving stage let us call its stage 1 we do not want that stage to move in the vertical direction. So, what has been done here is to at the 2nd 1 we have added the 2nd 1 here attach to the 1st 1. So, this becomes our stage 2, if you imagine here how stage 2 would deform? The stage 2 would deform in it will translate to the right, but the same time it will move up just as this is stage 1 is going to be move down, stage 2 is going to move up and those 2 will be compensated. So, stage 2 we have pure translation in the x direction that is the folded flexure mechanism how do we analysis this.

One of the things we should notice here is the ingenuity of this design. So, you are taking simply 4 per 8 flexures now the way we did here with 2 beams if I add 2 more beams. So, I have the block here and I have beams over here it is fixed and fixed now this stage what we can call stage 1 is going to translate in both x and y directions. So, we add another 1 here another set of beams pair of beams now attach this will block now, this stage 2 if you dimension the beams beam lengths and the positioning of the 2nd block related to the 1st 1 if you do it right as it is done here. So, they should all be aligned here which have not done in what I have drawn.

Ah is if you do that this 1 is going to purely translate let us say I want to find that you also find the stiffness, if I apply a force here how much does if move how much does this move if it moves purely in x direction what will be this movement is let us call it delta we want to know what this delta is if you know the geometry of the beams and there material properties then is say geometry the length as well as the cross section right.

(Refer Slide Time: 08:56)



Now, we are taking uniform cross section schemes how do we do that, that becomes our strength of materials as we incorrectly call or mechanics of materials is what we have.

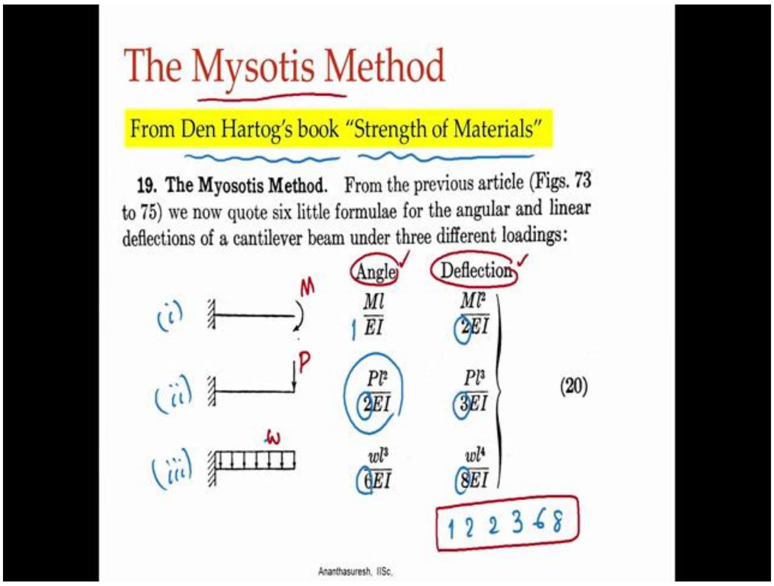
So, what we have is if I say I have a beam which is fixed here other side it has a block I am looking at only 1 beam. So, what are we saying this block is not going to rotate and we are saying that, if we do it right we can also prevent its motion in the y direction meaning again let me say that this is y direction this is x direction; that means, that I can imagine that there are some rollers here. So, this block can move in this direction, but cannot move in a y direction also then how do I analyze this beam let us say this has the length l and its modulus E and 2nd moment of area I, if I have that when I apply a force F here how much does it deflect.

And the deflection as we have already seen its going to be like this. So, there the block which was there is going to move there. So, we want to know this delta here. So, it was over here it was move there now we want to know what this delta is right. So, now, if I just do the abstraction if I have a cantilever beam that is guided here. So, 1 and here it is fixed here and the other end here is guided transversely it can move this way when apply a force here I want to know how much it moves. Some of you may remember this formulas right away as how much it will move that delta you may remember something, if it is a cantilever beam everybody knows or most people would remember it will be F l cube by 3 E I.

So, that 3 is for a cantilever beam, but that is going to be different for fixed guided beam cantilever is fixed and free; that means, that I will have this if I have a beam like this in apply a force it is going to deflect like this there will be a slope here whereas, here the slope is 0 it is tangent should be the vertical line so guided. So, what is this number they all are going to have this form and there will be number that is different, that you can do by using what we call the summation linear summation method super position method where we take a cantilever beam plus which has this force F acting on it another cantilever beam where there is moment acting because when we are not allowing this to rotate there could be moment that everybody knows.

So, this problem that we have here we are splitting into 2 of what we call static determinate cases this statically indeterminate we get this and this can be done very quickly we do not have to look up anything because this will be useful for analyzing complaint mechanisms.

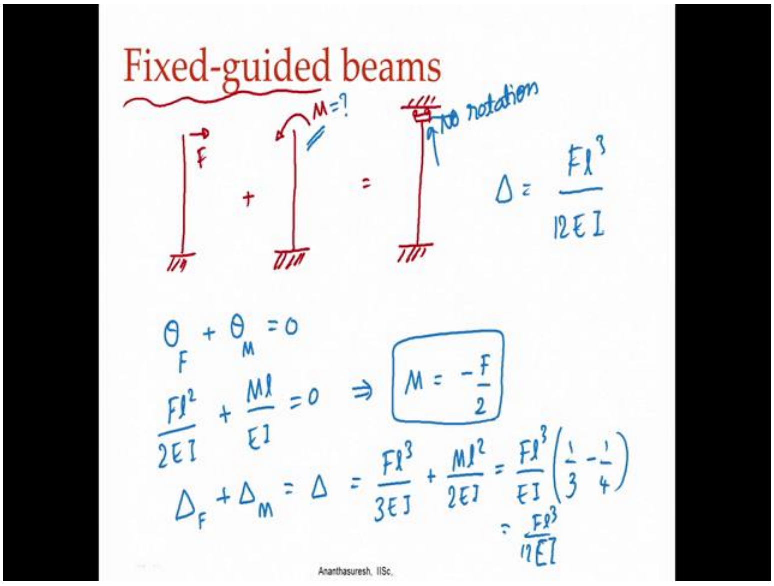
(Refer Slide Time: 12:41)



So, let us look at what is known as this Mysotis method which I have taken it from den Hartogs book title strength of materials or mechanics of materials, you need to remember only 3 numbers in this method. So, this is 1 and then 2 and another 2 and 3, 6 and 8. So, you have to remember 1, 2, 2, 3, 6, 8 and there are 3 cases here case 2 cantilever with n moment case 2 cantilever with a transfers force and case 3 which has a uniform loading this case I think uses the letter W and. So, let us use that though there is W and there is P there is mid M.

So, what is the end angle and end deflation? So, all of them you can see the form M l by E I in there is 1 M I square by 2 E I, P l square by 2 E I, P l p by 3 E I, W l Q by 6 E I, W l to the 4 by 8 E I. So, these numbers if you remember their form is easy to remember by just looking at the dimension of the quantity, this has no dimension these an angle deflection should have length units or length dimension. So, if you remember 1, 2, 2, 3, 6, 8 then you can solve any big problem can be boundary condition.

(Refer Slide Time: 14:22)



So, that is the Mysotis method we can very quickly do this before we go to finite element analysis for this fixed guided beams. So, what we are saying is that we have the cantilever beam with a force F here and then we have the moment here and moment M, that is going to be equivalent to something that is going to only move like that it cannot rotate is what we want fixed guided beam, that is what we have fixed guided beam. So, let us get that condition. So, its statically intuitive minutes. So, we do not know what this

M is, but we can find the m based on the fact that if you do linear is proposition let us say the angle here that is the rotation for this 1 and this 1, theta at the free end due to F plus theta due to moment m should be equal to 0 because there is no rotation here because we said that block does not rotate.

So, what is theta F if you go back and look at Mysotis formulates. So, the rotation is this is P here do you have F, F l square by 2 E I that is the theta here. So, F l square by 2 E I and then due to the moment that is M l by E I that is 1 that 1 here right M l by E I that is the angle. So, we have M l by E I that is equal to 0 which means that M is equal to minus F by 2 moment will be the opposite to the direction that we assume there. So, now, I want to know the total delta over here. So, that is over there how much does it move. So, delta F plus delta M will give me the delta that I want what is delta if you go back to Mysotis and delta did you force is this familiar F L Q by 3 I. So, F l cube by 3 E I and then we have due to moment that is M l square by 2 E I.

So, plus M l square by 2 E I again 1, 2, 2, 3, 6, 8 formula noting that M is equal to minus F by 2 I can take F l cube by E I that will be there for any beam by the way and that will give me 1 by 3 here then M is minus half times F, F we have taken out. So, that will become half and then 2 here that will become minus 1 by four that will give me F l cube by 12 E I. So, in this particular case the delta that we need comes out to be F l cube by 12 E I now, if you go back and look at our parallel motion flexure we have 1 beam there that we analyzed another beam these 2 beams are in parallel arrangement.

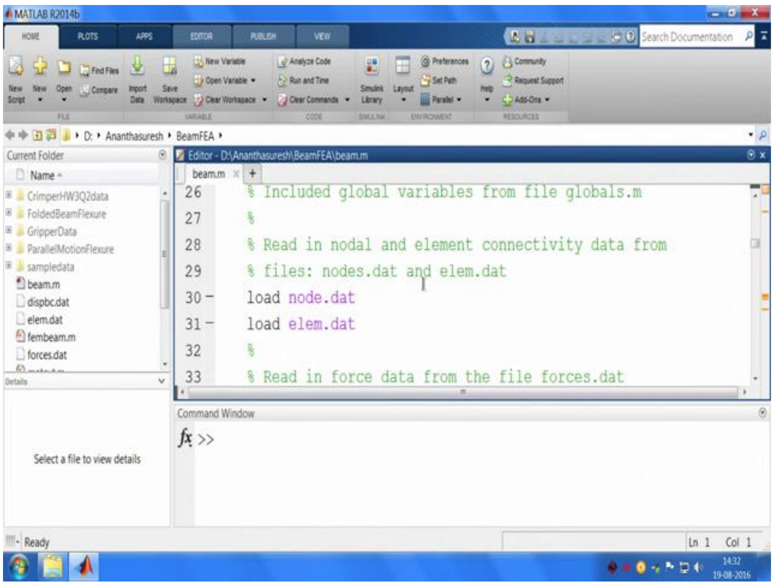
So, the force that we have here will be shared equally by these 2 half half force goes has half in other words when you have these 2 the stiffness will be added because then parallel. So, the delta that we have here will be we got F l cube by 12 E I. So, we have to add another of 12 if I 1st get the stiffness constant F l cube by 12 E I. So, the spring constant for this will be F divided by delta that will give me 12 E I by l cube that will be the K for each beam. So, we have 2 beams in parallel. So, you have to add this spring constant that becomes 12 E I by l cube plus another 12 E I will cube will get 25 E I by l cube.

So, once you have that for given force you have these spring conscious for the overall beam you can get delta, this is how we can do by hand for small displacements now let us go to finite element analysis and try to analysis these things because as things get complicated for example, what we have done if you go to this 1 we have to have again four beams each of which is fixed guided fixed and guided. So, fixed guided relative to this 1 this is fixed and this is guided you need to keep on adding now these two will be in series and we can go like that arrange, I identify the arrangement whether it is series are parallel you looking get for any configuration of complaint mechanism.

We have seen quite take complicated set of flexure mechanisms in the last 2 lectures, for doing that we have to go to finite element analysis, towards that we will look at mat lab codes and tried to analyze this parallel motion flexure. So, let us look at the folder which has all the files to run finite element beam finite element codes in mat lab it has 4 data files this could be see the d a t l m dot d a t forces that d a t node that d a t we look at what kind of data these 4 data files have you need to have the beam dot m that is m file mat lab script file, you need that and then you also need f e m beam and then you need 3 other files call mat cut plot beam l e m and then veccut they are all map labs scripts and there is also a read me that t x t file that you can read about the format that these data files need to follow.

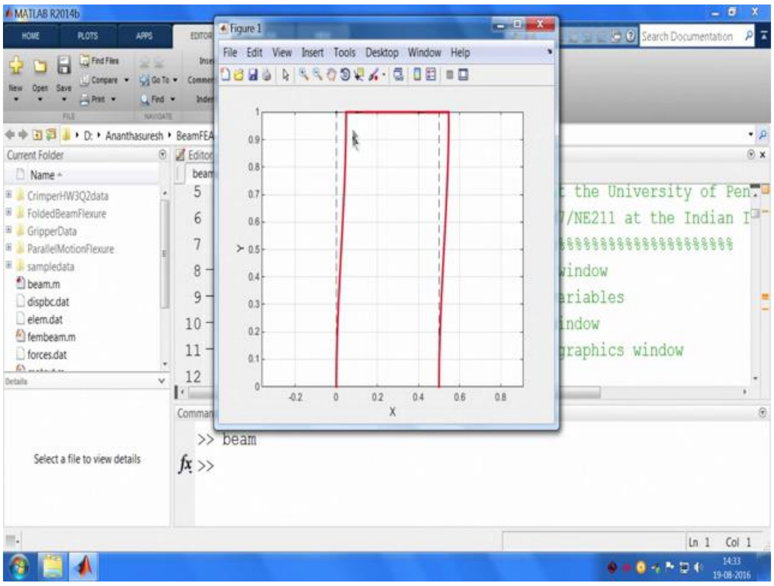
That we will see when we get to mat lab code, we click on this beam that is a main file that opens mat lab, sometimes version compared to be it is may be there you have to take care of few comments.

(Refer Slide Time: 21:02)



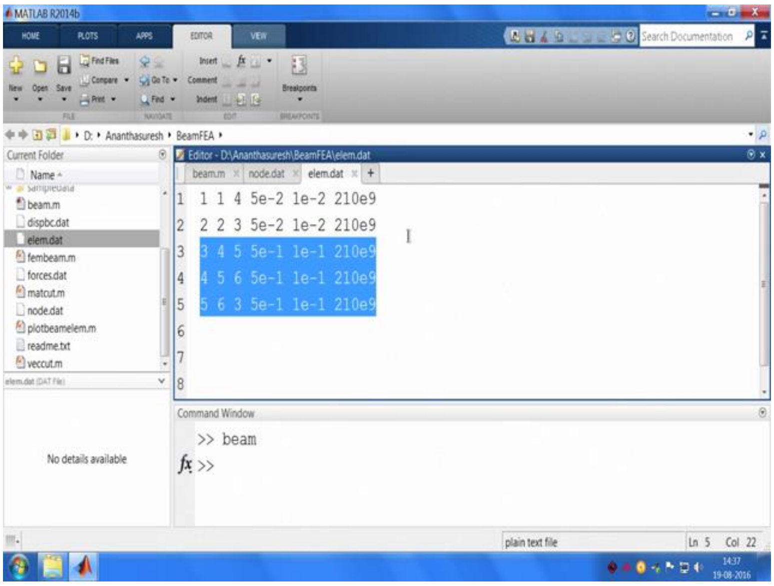
But this is there will be the latest version mat lab 20. 14. So, here you have the code which was written a long time ago, but it works and you can use it to run any type of 2 dimensional compliant mechanism this font is little too large. So, let us try to you make it a little bit smaller. So, let us go to preferences you need fonts lets reduce it to let us say 18. So, that you can see more of the do not do preferences 18 this is there I can reduce to 18 supply its say is still does not where I need bigger font is good. So, you can say. So, this 1 already have loaded the input files. So, when I run it when I run this here run this click this it direct and it gave me the output file, whether dash line here shows the parallel motion flexure before applying the force and then after applying a force over here it has deflected like this.

(Refer Slide Time: 22:21)



Here we have 1 this is node number 1 node number 2 node number 3 node number 4 and there are elements connecting 1 and 4 and then 4 and 3 and then 3 and 2 the 3 elements and we have taken care to make this 1 relatively rigid compared to these 2 segments, we have elastic segments to vertical beams corrected by horizontal block that also in this code we have to modalize a beam element, but will make it rigid how do we make it rigid for that lets look at the input files to see how we do it, in this particular case when apply a force it is deflecting the way it is shown.

(Refer Slide Time: 23:37)



Let us look at the files here data files 1^{st} 1 is node that d a t which has nodal coordinates. So, you have 6 nodes here I talked about 3 in that plot let me tell you what this additional ones are this is 1 2 3 and 4 those are enough for this mechanism later we are going to add the folded beams I have created 2 extra nodes this is 1 2 3 4 and then there is 5 here and then 6 there. So, there are 2 extra elements. So, that from there I can hand 2 other beams to make the folded beam later on.

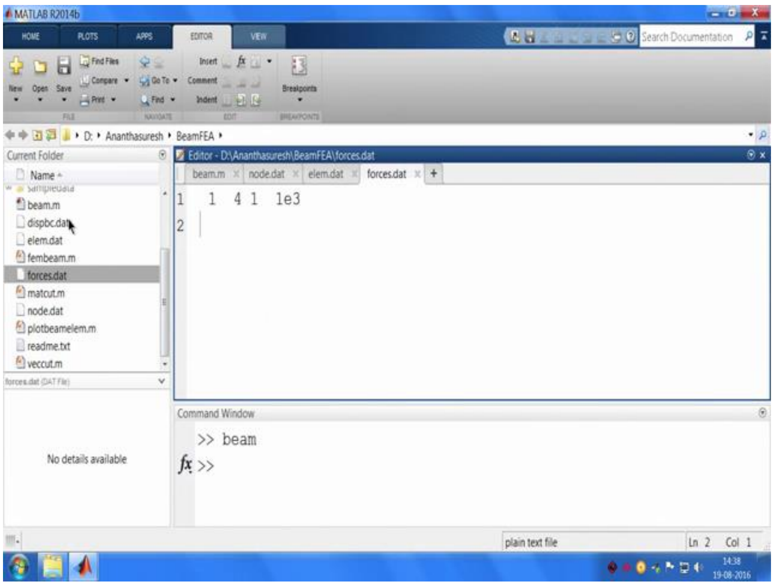
So, we have 6 nodes here those 6 nodal coordinates are shown here the 1st 1 shows the node number 2nd column shows the x coordinate 3rd columns shows the y coordinate the 1st 1 is that the horizon 000.5 and 0. So, the distance between the 2 vertical beams is 0.5 and the height of the beams or length of the beams is 1 the y coordinate and then we have this intermediate ones 5 and 6 at 0.1 and then 0.4, next thing that we need to look at how to define in the nodal coordinates you can add as many nodes as you wish or as you need in a complaint mechanism.

Now, let us look at the element that is e l n e l e m dot d a t. So, here we have 5 elements again why are there 5 elements let us look at this code. So, this is the 1st element flexible beam 2nd principle we will 2nd element and since we introduced 2 x l nodes here I defined this as a element 3 element 4 element 5. So, element 1 and element 2 are the elastic segments 3 4 5 are rigid segments and will see how we make it the rigid. So, the 1st 1 that connects 1 and 4 2nd 1 connects 2 and 3. So, this is the element number 1st column 2nd column and 3rd column indicate the node numbers that make up that element. So, node 1 is made with node element 1 is made with node 1 and node 4.

It has a rectangular cross section where the breadth of the beam is 5 centimeters that is 5 10 power minus 2 and this is 1 centimeter breadth this is b this is depth d and young's model is we take an steel to an in tang gigapascal all in s I units meter units, 2nd flexible segment is made up of nodes 2 and 3 2nd element 2 3 and get 5 centimeters 1 centimeter and then same is not in us the other 3 elements that is these 3 are supposed to be rigids. So, if you notice here the node numbers are 4 5, 5 6, 6 3 and then they have 0.5 meters. So, this is 50 centimeters this is 10 centimeters and same thing these how you can make rigid blocks in finite element analysis using beam elements you break there cross sections much larger.

If you want you can make young's modulus very large whichever what matters is e times i young's modulus times 2nd moment of area. So, you can make either of them or even both of them very large compare to flexible segments. So, you can create a rigid block with that we are able to get the deformation like this a where would you apply the force we applied at this node that is node number 4.

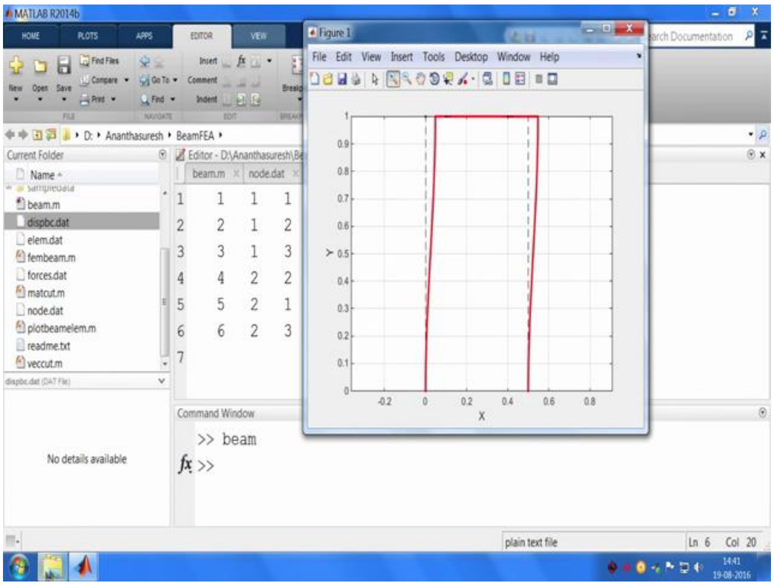
(Refer Slide Time: 27:24)



So, we can look at the force find there again this the serial number the force boundary condition node number and then I put 1 here, the reason is there is 1 is that I am applying force in the x direction in a beam finite element code each node in it is a pain or beam each node we have 3 degrees of freedom, the 1st 1 refers to x displacement, 2nd 1 refers to y displacement, 3rd 1 refers to rotation or the slope of the beam of that point.

For beam finite elements the slope degree of freedom and transfers and axial or x y displacement translations they are all independent. So, we have 1, 2, 3, 1 refers to x direction these the force some 1000 Newton's have applied 1 into 10 power 3. So, that is a force file we also need to say where we have fixed that is contained in this file called dis b c dot d a t.

(Refer Slide Time: 28:23)



Lets open it and see, we have serial number 1 2 6. So, we have six boundary conditions what are those we are these the 2nd column is the node number, 3rd column is the degree of freedom number, 4th column is the value of the displacement that you have specified there. So, the 1st 1 refers to the serial number 1 boundary condition of the displacement the 1st 1 is at node 1 and it is x degree of freedom that is fixed to 0 if you want to you can also put non 0 values.

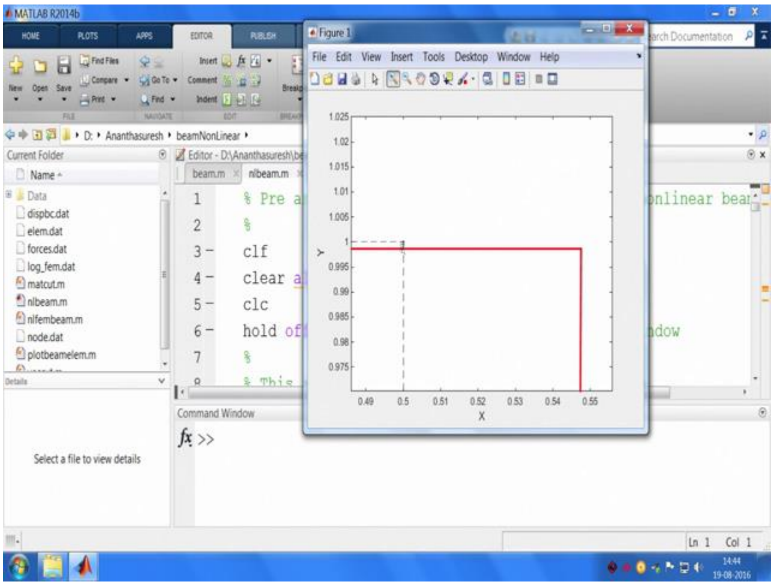
And there 2nd 1 node 1 y displacement 3rd 1 node 1 the rotation because if you look at our structure this is node 1 this is node 2. So, these 2 are completely fixed what we call cantilever condition it cannot move in the x and y directions, it also cannot have any rotation that is why an inner have any rotation that is why it is tangential to this line and same thing here and that is why we have node number 1 and then node number 2 all 3 degrees of freedom are fixed with that variable to get the deformation that we got, but if you see if i zoom in here how much ever i zoom in you would find that there is no y displacement of this rigid block it was here it is still there.

It has moved from here to here in x direction, but not in the y direction there's in nature of the linear finite element analysis. We have to run this with non-linear finite element code in order to see the more realistic 1 because this structure in reality is going to have this block not only in x direction, but also move a little in the y direction which is not shown by the linear analysis, 1 more thing before we go to non-linear before that you need to notice is that from here to here i have a single beam element. So, in finite element analysis the unknowns are only the degrees of freedom at the nodes. So, i know the x y dis x y displacement and rotation at this point i know x y displacement rotation at that point.

So, if I joint this point that point after displace in them occur in to their values that I compute using finite element code I would have a straight line from here to there, where as here we have a curved line like this that is because if you go back to our file there is something called plot beam l e m script, that takes into account these slopes at this 1st node and 2nd node and the displacement and interpolates using a cubic function because for this beam element in finite element analysis we have used cubic shape functions and that is what is used to interpolate in between that is why we will get in the nice curve 1 that you see in the real device also and this applies to any complex beam complaint mechanism that you make up it will show you the interpolate it beam also.

So, you have degrees of freedom only at the ends, but you would know the deformation at agreed point, normally in any finite element software this will not be little show as a straight line, if you wants to the curved 1 then you have to put several nodes in between. So, that you can see the deflected profile where as if you use a shape functions to interpolate throughout the element you get a realistic picture that is what you have done. Now what will do is we will go to non-linear finite element analysis I can open that here in the same folder that we have going to have we are going to give you both linear and non-linear analysis files there is n l beam, it has a few more files n l f e m beam and then plot beam l e m whatever we used in linear analysis and then we have mat cut data that you need. So, I will open this n l beam and try to run this, when I run this it will say change folder because that is happens to be in a different folder.

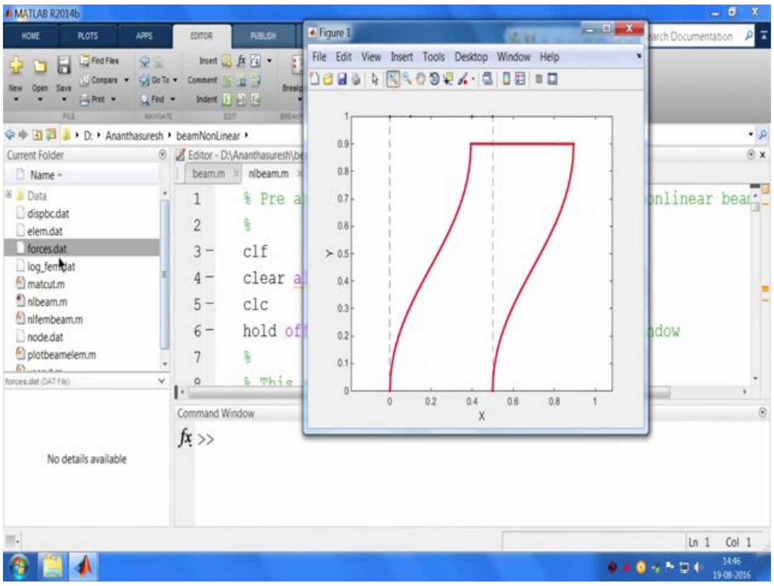
(Refer Slide Time: 32:47)



Now, you see you got a different 1 because I have different set of input files here where I have applied some force i have not only put this original 1, but also added the folded beam suspension that is we have 2 vertical beams hanging remember we had 5 and 6 nodes now 7 8 and there is a 9 also here, I will tell you why we have an extra node 9 here that is what I have done, but then if I want to run the old once 0 this what i can do is lets go back to the folder where we had these input files which i will copy 4 input files that i have let me copy these and then put them into the non-linear folder. So, i am just copying it ask copy and replace is say s for all of 4 of them because we want a over right what are they now let me close these old once because they pertain to the linear 1.

Now, learning the non-linear beam for how to use it. So, now, if I go there and run this now you see it took the files that we use for linear analysis. So, with only 6 nodes that we had 1, 2, 3 or 3, 4, 5, 6, now you can tell whether it has move down or not unless you zoom in. So, let us actually zoom in over here to see there is any deflection in the downward direction and you would see if I do move or zoom in there is indeed downward deflection also here because a no linear code, if I increase the force value which we can lets go to this force open as text how is only e 3 node number 4 is what you are applied.

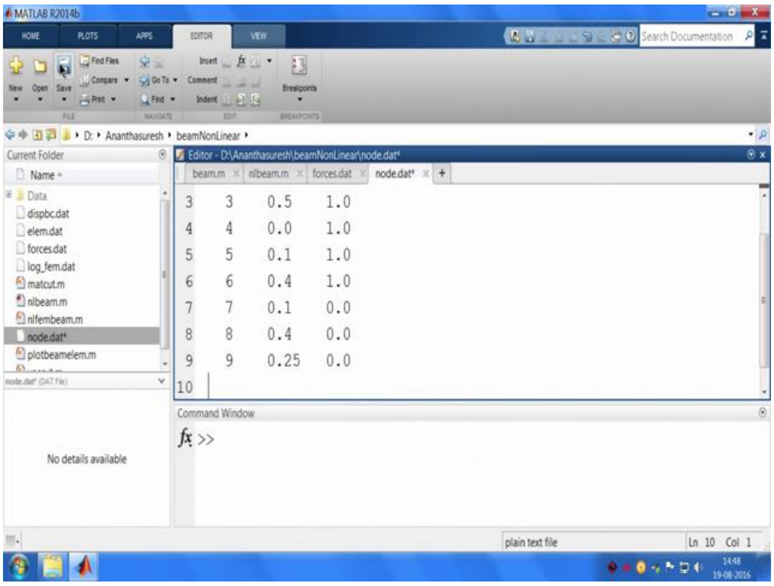
(Refer Slide Time: 35:00)



If I increase it lets say that much that is ten thousand Newton's let save it lets go to that non-linear beam code now even it, now we can see clearly that it has not only moved in the x direction the node that was here has moved all the way down here there is movement in this direction and also in this direction that is what the these are the non-linear code later on we learned more about non-linear finite element analysis what it actually means.

Now, we have this. So, if I have this 1 as the stage that is supposed to be purely in x direction and not getting it here when I do non-linear analysis which is actually realistic is moving down we have to compensate for that we add 2 extra beams which is what we had which is opened the file. So, let me go back and put those files and then see what happens, again instead of having this I have 2 extra nodes there is 1 5 and 6 i am going to have a vertical beam vertical beam and this. So, in order to get little bit of experience for that lets actually try to modify it right here.

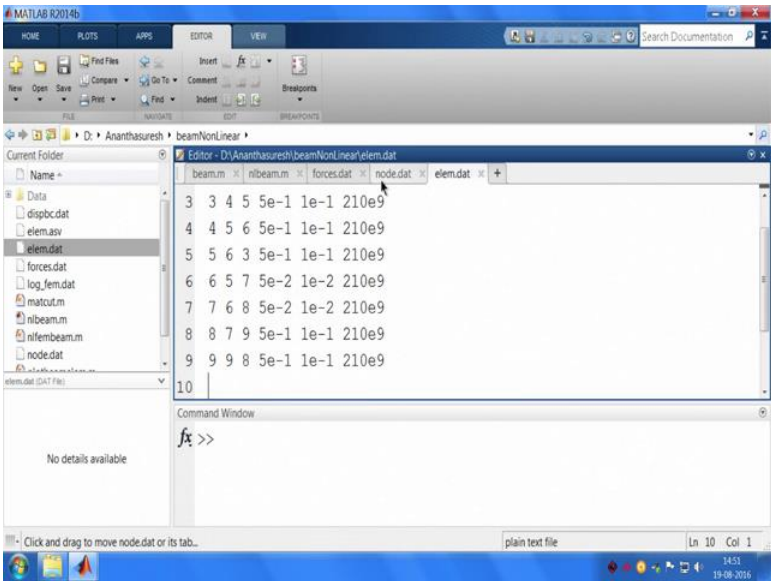
(Refer Slide Time: 36:25)



So, let me open this you get some experience of doing that node that that has 6 nodes now I have to add a few more nodes. So, I am going to add 7 8 9 at the bottom that is let us look at the figure. So, I have this original 1 I am going to have three more nodes here 1 here 1 there 1 there. So, that I can have this vertical beam this vertical beam and 2 rigid blocks. So, the center I will have that becomes a stage now that becomes our moving stage which is supposed to have purely x motion and no y motion because we see that this top stage cannot have purely x motion it has y motion.

So, if you do this just as this is moving down relative to this line the next stage that we have that is going to move up closure to its relative frame. So, this down ward motion this upward motion both are supposed to cancel. So, get this stage to move purely in x direction that was intend of the folded beam suspension. So, I am going to add 3 nodes here. So, let us put that over here. So, that is 7 and that I will put at 0.1 because that was node number 5 just below that, but I at take it to 0-th level that is it comes down that is 7, then I will put 8 in a node number 8 node number 8 I will put at point 4 because it should be write below node 6. So, point 4 that also comes to 0 there and then introduce 1 more which is node number 9 and this I am going to put it the midpoint between the 2. So, that becomes a center of gravity of the 2nd stage that is hanging from the 1st stage that is also at 0-th level.

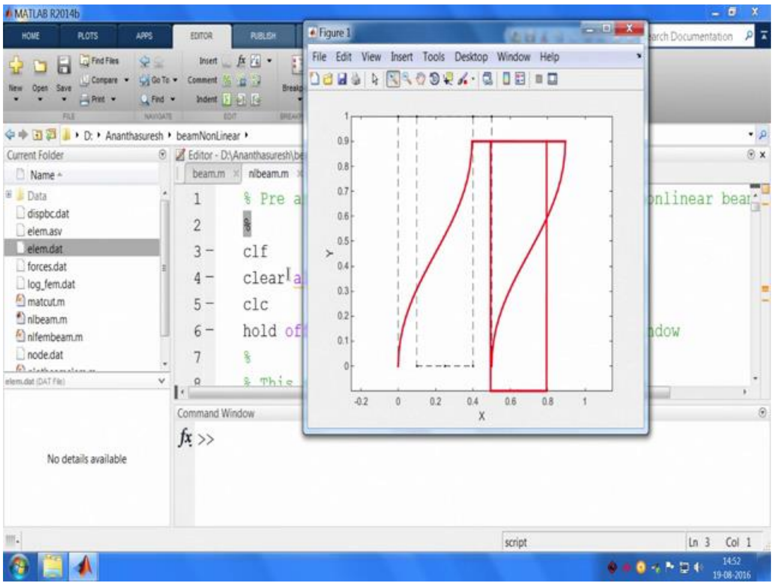
(Refer Slide Time: 38:44)



So, I have 3 more nodes now let us save it and then let us look at the element. So, here already have 5 elements now I want to add 2 elastic segments and 2 rigid segments. So, 2 elastic segments again if we go back to the figure I have to hang from that node to this node that I have created and then that node to this node I have created in between I have 2 more that is from here to center of gravity center of gravity to the other 1 from the element. So, the 6th element will make it like a flexible beam element that is going to be from node number 5 to 7 that we just created. So, 5 to 7 and that is going to be flexible. So, that is 5e minus 2 5 centimeter and then 1 centimeter and young's module this is 1 10, e9 and then 7-th 1 also flexible that is going to be from 6 and 8 6 and 8 as we remember of nodal data for their 1 below the other again that is 5e minus 2 and then 1 e minus 2 and then 210e9 and the I also want to have this rigid elements that we have.

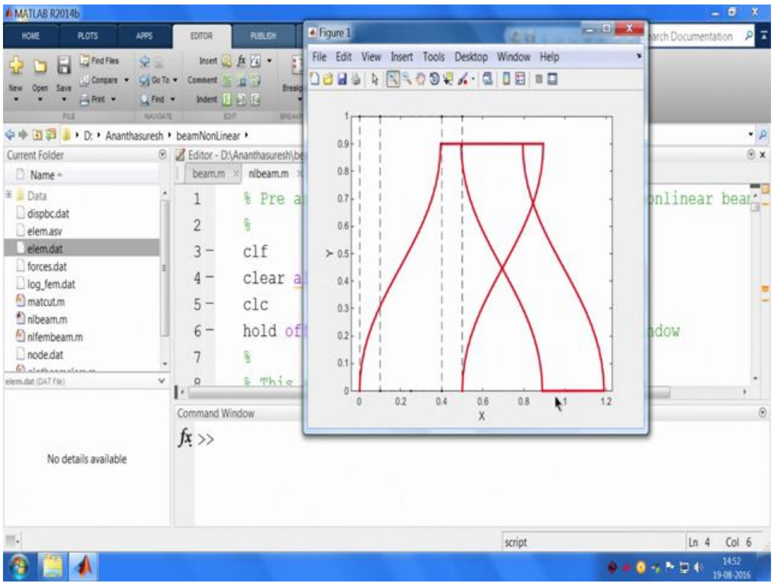
So, already have 7 elements and need to add few more now that is 2 rigid elements that we have element 8, but I would do the new nodes that we created are here. So, 1, 2, 3, 4, 5, 6, 7, 8, 9. So, 7 and 9, 9 and 8 that is what we need to have rigid elements. So, 7 and 9 and this is I would like to put 50 centimeters half a meter and this is also 1 e minus 1 will keep young's modulus the same as before and then 1 more element that is going to be from 9 to 8 or 9 to 8 then this also a rigid 1. So, 5 e minus 1, 1 e minus 1 and then 210e9. So, let us say this and as far as the displacement boundary conditions go did not change till whatever where fixed they are fixed, but forces we need to change and force here instead of node number 4 now we need to apply force for the 2nd stage if I apply 4 lets run it lets run the way it is now to see what happens.

(Refer Slide Time: 41:23)



So, now you see it has deflected this portion has simply moved like a rigid body, because if you see now I have added these thing this and this are rigid a applied force still at the node 4 in the x direction. So, this has deformed where as the other 1 is like a rigid block to that 1 we attach a 2nd stage which 2 elastic beams that moves a rigid body because there is no force here. So, we have to apply really the force at this node are will at the apply at the center of the right there is a node number 9. So, we will go back go to the forces this is the node number 2nd column that I will make it 9 let us save it.

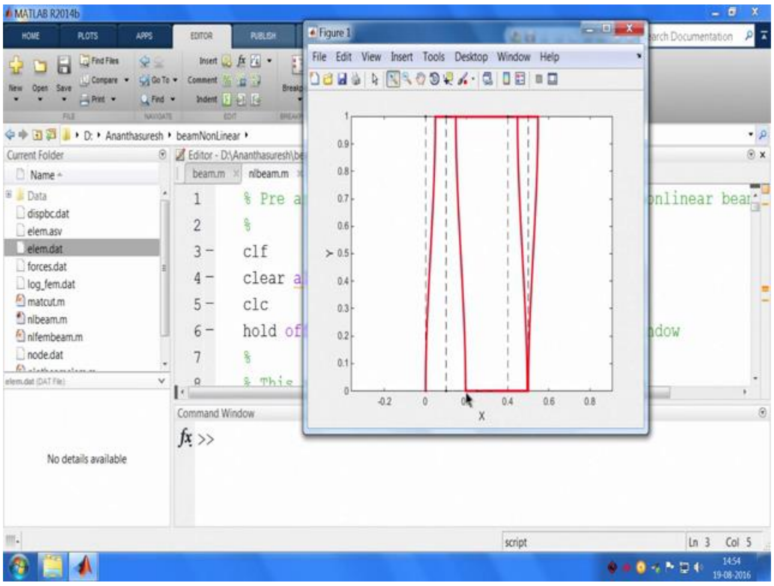
(Refer Slide Time: 42:21).



And now let us run it you see that now we are applying force it was here it has moved a lot of course, let us reduce magnitude re run it, but what you notice here is that this is stage was over there it has come down whereas, this 1 is at the same level we can actually zoom in and find out if there is any deformation at all if you see its exactly at 0 0 how much ever I do, if you want to see the line we can come and type grid command in mat lab that will create grid in the thing we will see that its right on the little bit difference is there see.

Now, i am really zoomed in a lot this little deformation this the difference is coming, because it is non-linear code 1st of all and other is that these beam elements and or elastic segments whereas, this is rigid segments I just made cross section ten times larger, but they may also be deforming a little bit and the other reason is that when you look at the spacing between the 1st stage elastic segment that is from here to here that is 50 centimeters whereas, this is from here to here there is less than that its much less its goes to 0.1 here 0.1 there about 30 centimeters right that also matters for how much it would move in this direction that is if I our to take this beams put further apart how much it comes down actually is effected by that spacing, we have to do non-linear analysis to find out how much that is and that is why if i really zoom in a lot. then I start seeing a change if you want to make a another modification which we will discuss after reducing the force. So, it will be let us not apply. So, much force let me apply 1 10th of that only 10 Newton’s and then go back here and run it now its little bit better you know what was the stage here has moved over there again.

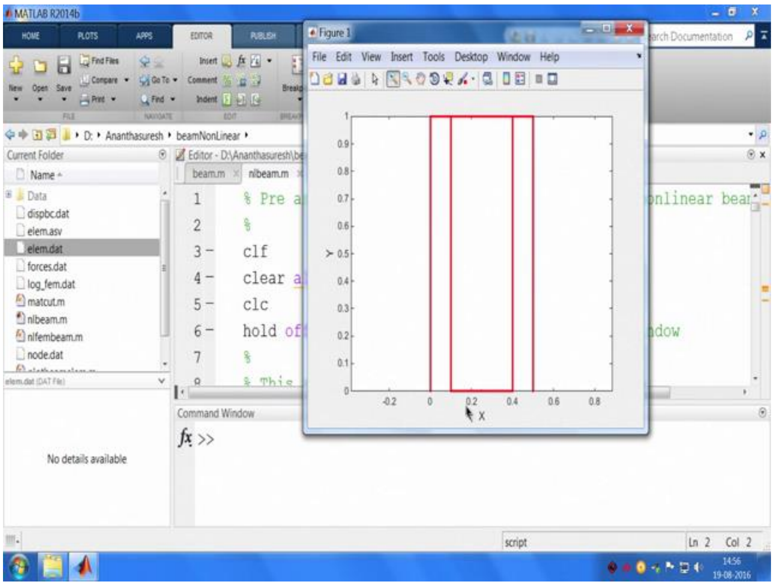
(Refer Slide Time: 44:26)



If i zoom in I might find that it has deflected to little bit I am really zooming in a lot then I see that there is a little change or here several levels I had to zoom in to see that.

So, otherwise there is not that much of this, if I really want to avoid this also then as you remembered from the earlier part of the lecture where the 1st stage and se 2nd stage were connected with a lever, again with couple of flexures we can do with beams here the reason that lever is needed is for this spacing the original spacing from here to here they need a little bit of downward deflection, from here to here for the 2nd stage there will be different amount of deflection. So, if you move this deflected of time you couple them using the corresponding lever ratio there were let as a and b if we arrange them correctly we can make sure that even under large displacement this particular stage will not run a vertical motion. So, now, we have a stage we are applying force it moves like this. So, once you have the code you can play with it lets do that now, instead of applying force in the x direction let us see what happens if I applying the y direction let us save it.

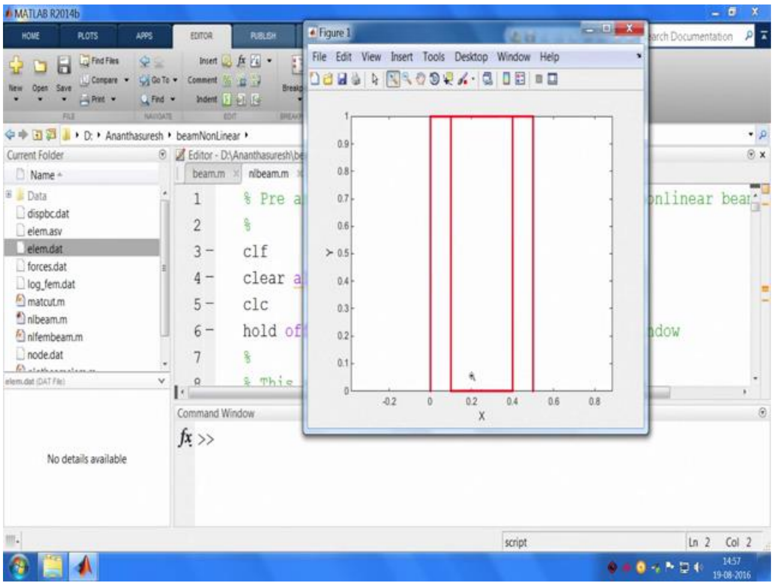
(Refer Slide Time: 45:53)



And now let me run it you see hardly anything because, when I apply a force in vertical direction this would be very stiff that you can imagine that if you have a beam transverse lead to be flexible, but axial it will be very stiff. So, for the force that we have applied which was thousand Newton’s it is hardly moving and we can see the values also in mat lab when you type w h o we can see all the variables that are there.

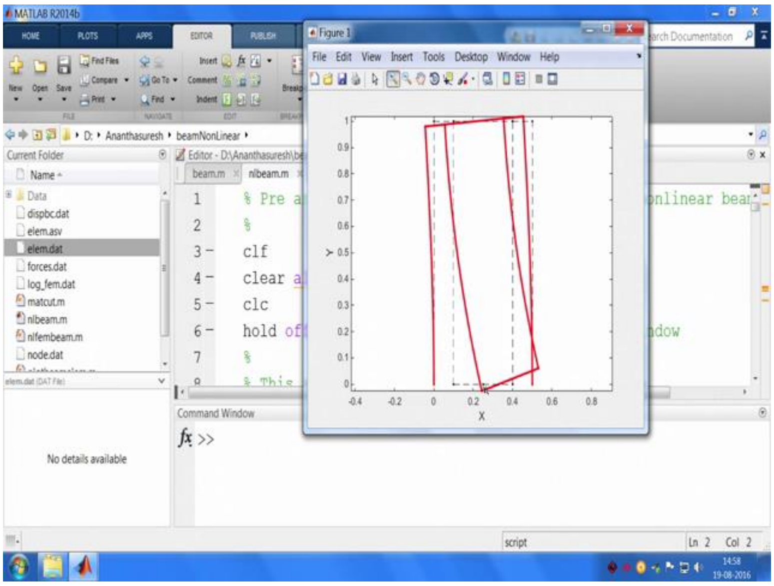
That’s u is the displacements if you type u it will show you all the values of course, there are lot of values it just shows that, what we need is the displacement let say at this node 9 which was in the middle how much is it moving in the y direction. So, the node 9 the degrees of freedom of that are going to be coming from back 27 because each node has a degrees of freedom, if you have 9 nodes there will be 27 degrees of freedom those 3 degrees that pertaining to node 9 are going to be 25, 26, 27, 25 in the x direction 26 in the y direction 27 thus rotation. So, if I want to look at what is the y displacement I have to look at degree of freedom 26 u 26, if i do will you show this is about 10 micron 9.69 microns for a beam that is 1 meter long that we have taken property geometry you have you can go back and check it moves very little very stiff. Now for fun lets actually apply a moment load there and make it 3 degree of freedom let us save it let us look at it now still nothing not much.

(Refer Slide Time: 47:38)



So, if I have to do this I have to apply very large value I put 6000 times more that is 1 omega Newton meter.

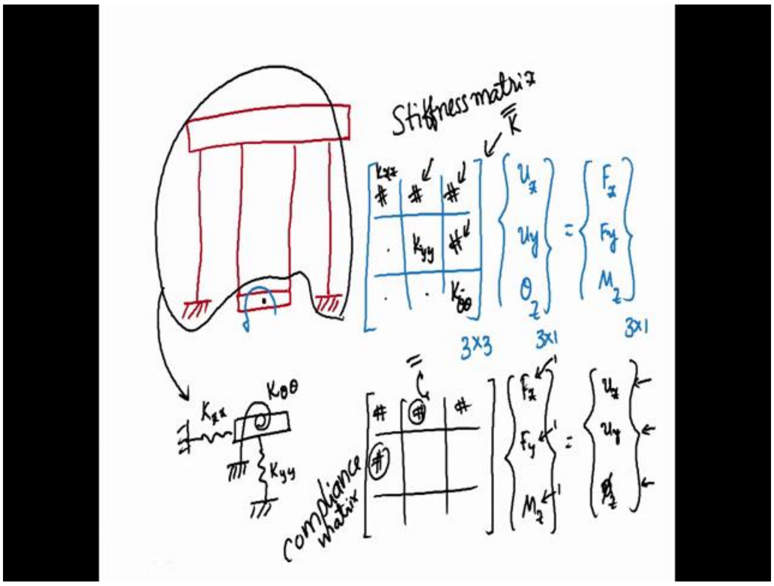
(Refer Slide Time: 47:56)



Let us see what happens, if I do now you see it is rotating that is I have to taken this and rotate it, when I rotate it you would have expected that this stage is simply rotate, but it does not it actually moves from there to here that is your rotation of the stage the 2nd stage we are talking about this stage now applied a moment at this point about z axis out of plane. When I do that it not only rotated, but also has moved that is called the cross axis influence that is your asking it to move in a particular force there, but the stages that you have this compliance stages will not do that they also will try to move in the other directions that is a difference between rigid body mechanisms and compliant mechanisms. Compliant mechanisms since they can be other degrees of freedom they do. So, they have this rotation also now if you remember in the 1st lecture and an a earlier lecture we talked about a stiffness matrix that we can construct for this block or for that matter any part of compliant mechanism at any point, now let us say the 2nd stage block I would like to know its stiffness in the x direction y direction and then theta direction as well as the cross axis meaning, when I applied rotation about z axis over here it has laid to axial this x displacement and y displacement.

So, for that I need to construct a 3 by 3 matrix let me just 1st go to power point actually show you what I mean. So, let me just draw that.

(Refer Slide Time: 49:52)



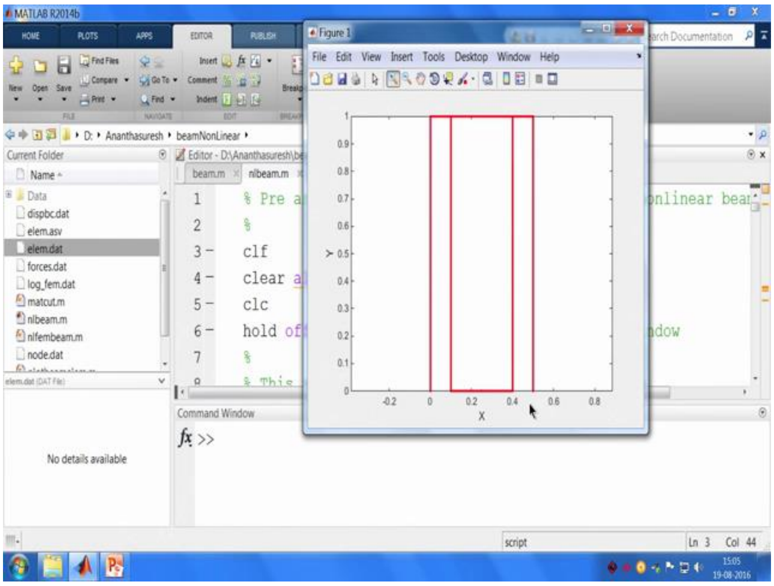
So, I have at that block. So, now, we have a that beam in this top block is there in another beam and this is fixed here that is the parallel motion stage, now we have 2 more things added and we have this block here and now what I am saying is that what we just ran is applying a moment load on it, when we did that instead of purely rotating it also moved in x and y why does it do because the 3 degree of freedom of this 1 if I call this u x, u y, x displacement y displacement and theta z and these will have corresponding forces F x, F y and moment M z that we just applied off course these 2 are not equal there is a stiffness matrix this is 3 by 1 this is 3 by 1 this is going to be 3 by 3.

So, we need to get these numbers here. it is going to be symmetric matrix we need to get those the way we want to do this is what we call stiffness matrix, which basically models all of this effect of all of this on this block. So, that we can have this block as if it is connected to 2 springs like this that is entire mechanism we are modeling with these springs and there is a torsion spring also here right, but this is not complete because we do not talk about this cross element right that is not here. So, having this K x axis if i call it K y y that is this going to be K x x this going to be K y y and this will be k theta theta, this torsion spring constant that is not enough there are this 3 more things we need to get we can get that using quantum analysis code which is what we will do now, the way we will do it is we construct what we call a compliance matrix which is inverse of this meaning what this will do is it will try to relate the inverse of the stiffness matrix.

Now, this is stiffness matrix this 1 is called compliance matrix. So, this will relate F x here F y here and M z and then these will be u x, u y and theta z and this also a 3 by 3 matrix this we denote by c we denote this by k this is the k matrix this is the c matrix. In order to get this C matrix what we will do is we will apply unit force 1 at a time we will apply F x equal to one when I do that at that point the finite linear model I will have x displacement y displacement and theta z right. So, if apply unit force in the x direction I will get displacement in the x direction u x I will put that there I will put the displacement I get when I applied force only in the F direction over there in the y direction displacement, I put there whatever slope I get I put here when I do that you see when I apply force in x direction when i multiplied this row with this I get u x because F y and M z then or 0 i get this value over there for that I will get this value, I am the when I apply this what I need here when I want to get u y I will take the 2nd row and multiply this when I do that this is symmetric matrix that is this value will be same as this value this multiply by that will give me u x.

So, I need to do 3 finite element analysis runs in a 2 d case applied force as 1 Newton there and other things are 0 then another analysis we are apply this 1, other to 0 and apply 1 Newton meter here other to 0, 3 finite element analysis which is what we will try to do now going back to mat lab.

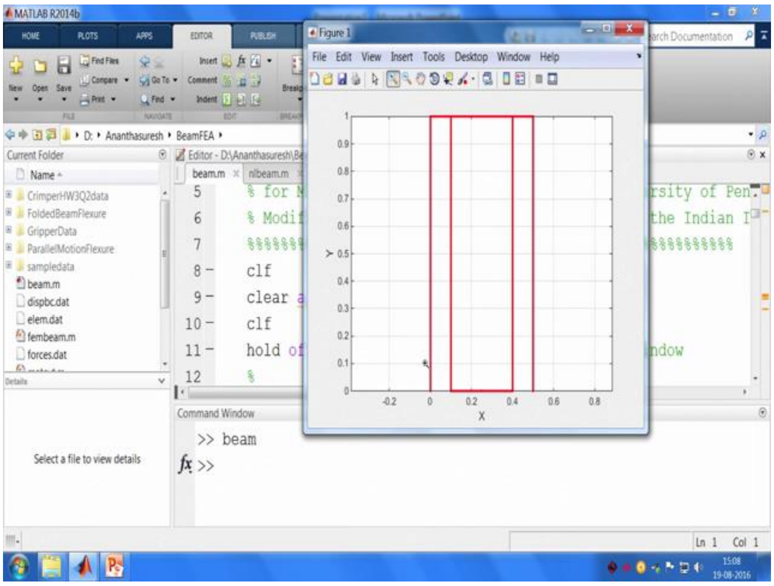
(Refer Slide Time: 55:07)



So, we can go back here go to forces other node 1 I will apply in the x direction. So, 1 Newton saves that and then beam and run it you hardly see that 1 is 2 little for it, but then we have all this displacement that we know about. So, if I look at u 25 that will give me displacement whatever 9.5313 e 10 to the power minus 5 if I put this as a 1st entry 1 1 entry of the C matrix I get that and then I look at the u 26 which is the y displacement of node 9 this 5.2712 10 to the power minus 12 I put that as 2nd 1 and 3rd 1 for the u 27 if I take that will give me 1.411 e 10 to the power 7.

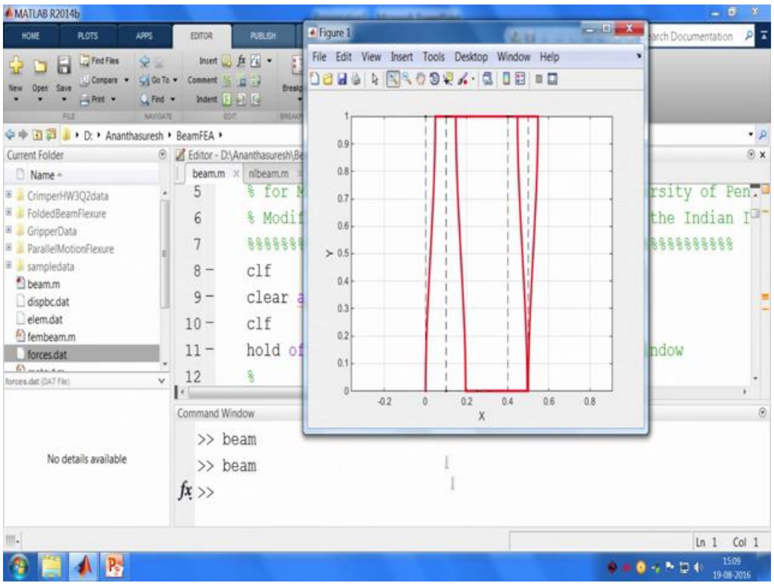
So, I put that I get the 1st row then I go back here and change it to 2 save it and come back and run another finite element analysis run, then I get again changed the displacements I look at degrees of freedom correspond to this node 9 that I have interested I get the 2nd row, then apply moment here change in this 2 3 I get that before I do all that I will get something that we have this C matrix if you take the inverse of that then you will get the stiffness matrix. That stiffness matrix will give you as we saw this stiffness matrix that let us actually go here stiffness matrix 1st you can say C matrix by putting these things we take inverse of that you get this matrix, which you can actually verify we come to the formula the way we did earlier in the lecture which would give you whatever that value is to compare and we will see the difference between linear and non-linear f e a. So, I do the linear and non-linear f e a there will be a difference for example, with the same data files that we now have we have a non-linear files this we have changed. So, if I put the linear 1 we will see the difference.

(Refer Slide Time: 57:31)



So, we have that and this and copy this put them into the linear infinite element code over right on 4 now go back to mat lab and close these files go to beam now run it change folder . So, now, if you see we get something, but let us actually look at the force data where we are applying force by putting a large force.

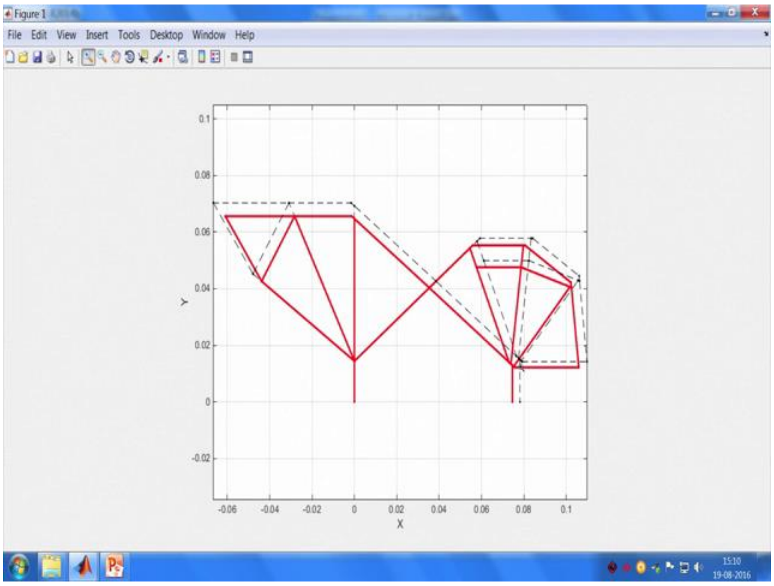
(Refer Slide Time: 58:12)



Here these in node 9 Ii will apply in the direction let us put this e 4 that we had earlier or e 3, that is 1000 Newton's come here and run now you see that this 1 if you go here and zoom in there won't be any vertical displacements how much ever you zoom in we do not find anything because linear 1 cannot do that. So, here also 1st stage also is not that we had already seen that is what you get, but what we learnt now is if we have finite element analysis code with u we can generate the lump stiffness matrix for any rigid segments that you have or even a point at that point how does it feel in x y and z directions.

So, you get the cross axis stiffness's also as part of this or you have to do 3 finite element analysis in 2 d and 6 because in three d because you have 3 translations and 3 rotations all of that you can get.

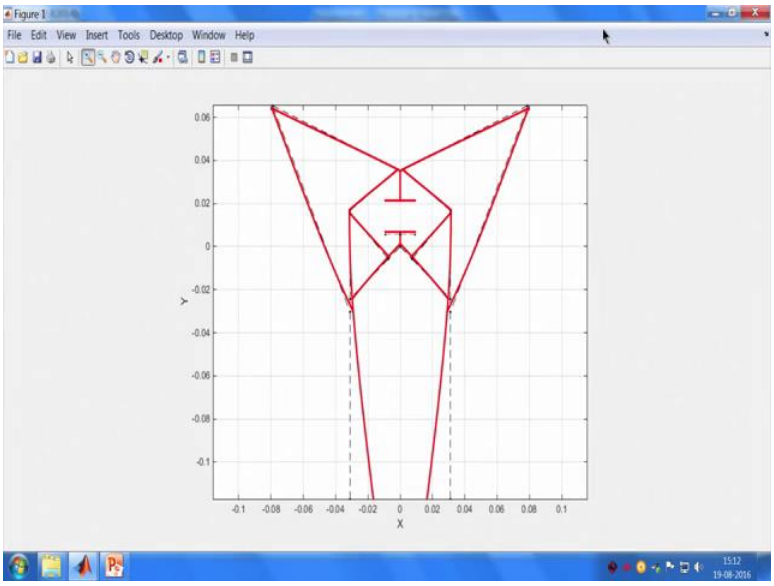
(Refer Slide Time: 59:29)



Now, 1 more thing I would like to show before we leave this is I also have over here lot of other data of file the 4 data of files some crimping mechanism and a gripper and other things, let us go to this gripper and we have this 4 input files which are which have many elements and they are already done. So, now, let me put those files over here into this folder copy and replace now, if I go to mat lab and try to run it we got a gripper 1 done. So, let me a logic a little bit and that mechanism I have with me. So, you can look at this. So, this is the mechanism that we have simulated 3 this 1 when apply force over here like this is closing in this is like a gripper.

So, this is symmetric mechanism whenever there is symmetry we do not have a model completely will model only the top half. So, if i cover this whatever is there is what we have the code in the mat lab that we just ran. So, there we have to apply force down here and then if I take both if I take half if I fix it if I do this will close in as it is seen in the mat lab simulation see this symmetric mechanism. So, we are taking only the top half for analysis that something to remember when you have a symmetry in a device you take 1 half and that is good enough because you save time computationally and with this 1 here we have again lot of beam segments 1 here 2nd 3rd 4th we are applying a force there and it is moving down. So, we can just putting the data files and get this.

(Refer Slide Time: 61:30)



There is another 1 over here. So, there is something called as crimper that have also 4 input files. So, for all of these mechanisms that we have a collection. So, for all of them we have this files which you can use and if you have new mechanism you can put them all here, now I go back to mat lab and run this that shows a different mechanism. So, that also I have here. So, which has thing like this when apply a force you can see the mat lab formation that we had when apply a force it moves this portion is moving up. So, you can ever so little.

It is a force amplitude put your finger here and then do this you will feel the amplified force, that is why displacing a lot at the input and moving a little bit at the output. So, this 1 even though its symmetric we have model the whole mechanism. So, over here we have taken entire mechanism applied forces here and here how it moves it has moved a little bit as you can see it has moved from here to there. So, let us say for this block if I want to get the 3 by 3 stiffness matrix then I have to run this finite element analysis 3 times for this by applying force in the x direction y direction and the slope and get the equivalent stiffness of that block by hiding the details of the rest of them, then I do not apply input force here I have to apply on the block itself suppose to be a rigid block on that block I need to apply x direction force y direction force and rotation. And the same set of files that I have I can also copy them into our non-linear code it is a different folder that is put them there copy and replace and then now run the non-linear code change folder.

Now, it shows, but here you can’t tell the difference much because this moving a lot, but this is non-linear code and was a linear code, but this will be more realistic and how much value this would have been verses what was there for the linear code there will be differences and will discuss that in a different lecture as to what is the essential difference between linear and non-linear course. So, now, we have spent lot of time to familiar with a mat lab codes both linear and non-linear the theory for this you have to take a finite element analysis class for that, otherwise we are going to put some supplementary files for you to look at where there is a theory of finite element analysis for this codes is explained in a few p d f files that you can download and read. So, what we are learnt now is that any compliant mechanism how much ever complex it is as long as it is 2 d and has beam segments we can analyze it we can also find the equivalent stiffness for a rigid block that may be there on it.

Thank you.