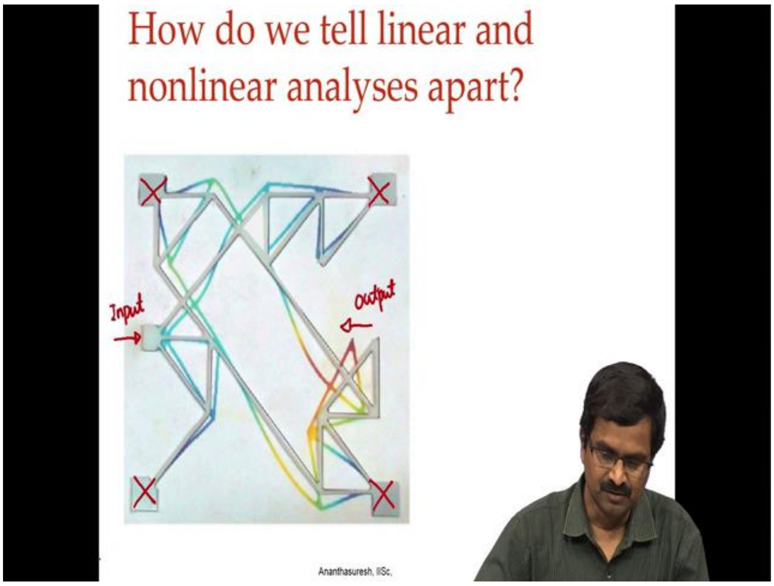
Compliant Mechanisms: Principles and Design

Prof. G. K. Ananthasuresh
Department of Mechanical Engineering
Indian Institute of Science, Bangalore

Lecture - 12
Subtleties in finite element analysis: geometric nonlinearity and contact

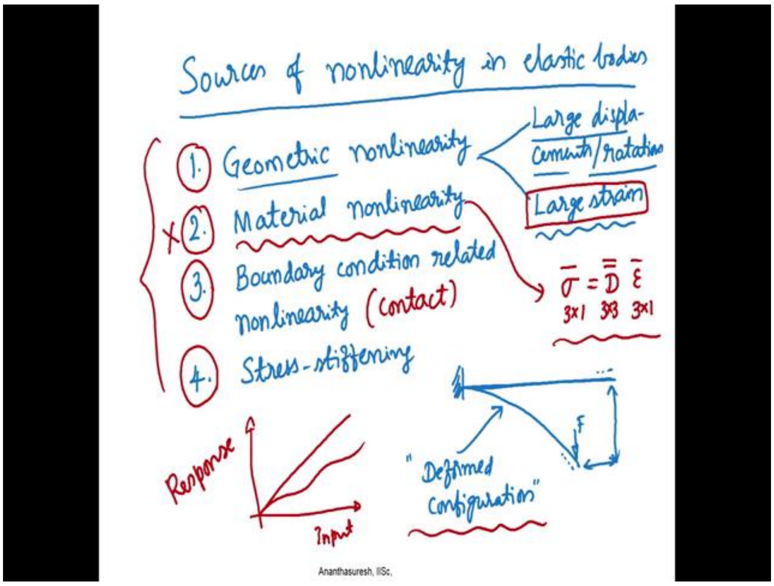
Hello, continuing the lecture number eleven where we discussed finite element analysis and how it could be used to analyze compliant mechanisms. Today we continue with that spirit especially with regard to linear and non-linear finite element analysis as it applies to compliant mechanisms. So, let us look at the Subtleties that we need to be aware of when we are doing Finite Element Analysis of Compliant Mechanisms.

(Refer Slide Time: 00:49).



We had started the last lecture with this question that when I take a real compliant mechanism and analyze it using finite element analysis be it your own code or any commercial finite element analysis software. We want to see how we can tell apart linear and nonlinear analysis because, it is really important because compliant mechanisms do under go large displacement, large rotations and lot of nonlinear behaviors they undergo when we use them in practice.

(Refer Slide Time: 01:28)

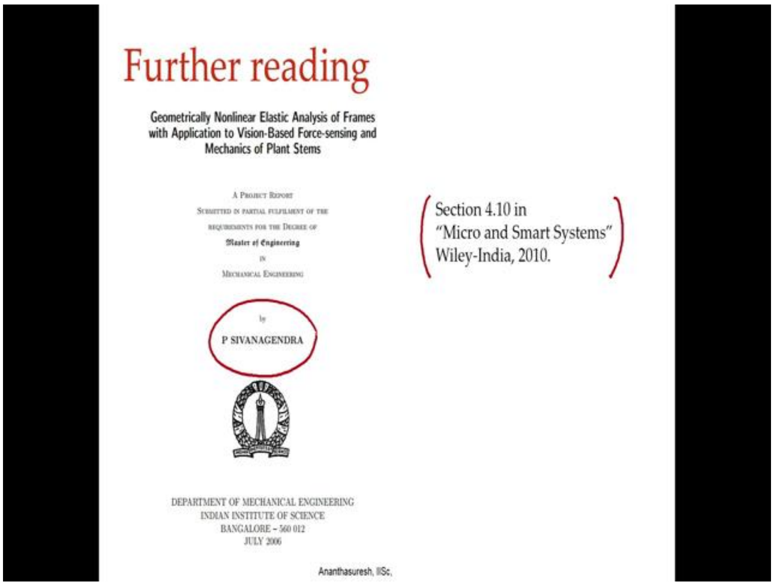


And that is very important for us to notice. Just to recall what we discussed in the last lecture we identified these as the sources of nonlinearity in compliant mechanisms or in general elastic bodies geometric nonlinearity that is large displacement, large rotations also large strain which is also important, and in the context I did not mention last time about buckling. Buckling is a consequence of geometric nonlinearity that may not be obvious, but we look at the analysis whether it is done using finite element analysis software or analytically it is basically a geometric nonlinearity that manifest as buckling, which is important for compliant mechanisms as we will see with couple of examples.

Material nonlinearity which we do not consider it is basically; the stress strain relationship that will not be considered in compliant mechanism normally because, we want repetitive motions. Boundary condition nonlinearity which is contact which we will show couple of examples today; stress-stiffening happens in beams and that has to be paid attention also; if you look at this emphasis and deformed configuration basically, the geometry nonlinearity arises when we use our equilibrium equations in the deformed configuration rather than the original configuration.

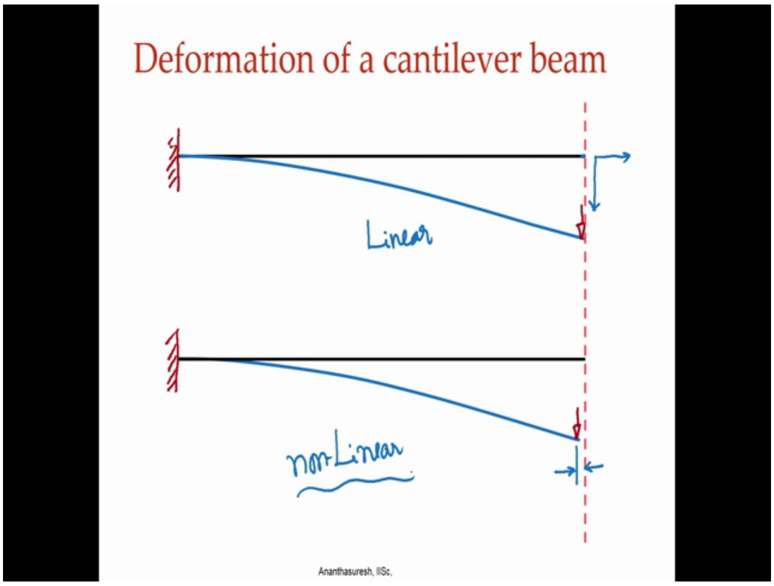
But in general nonlinearity really means that whatever response that you are interested let us say, there is some input usually a force or displacement and some response of the compliant mechanism when that is not linear and nonlinear in some fashion we call it nonlinear respond. That can be due to any of these sources, but only material nonlinearity we discount in the case of compliant mechanisms.

(Refer Slide Time: 03:20)



If you want to know more about it there are lot of sources for nonlinear finite element analysis all of those apply, but just in a concise form there is a section 4.10 in this book Micro and Smart Systems Wiley-India where you can look up some of these things and also my master students Sivanagendra had looked at geometric nonlinearity using beam elements, that has complete formulation in that masters report which you can find it on my website that is for your further reading to understand geometric nonlinearity.

(Refer Slide Time: 03:59)

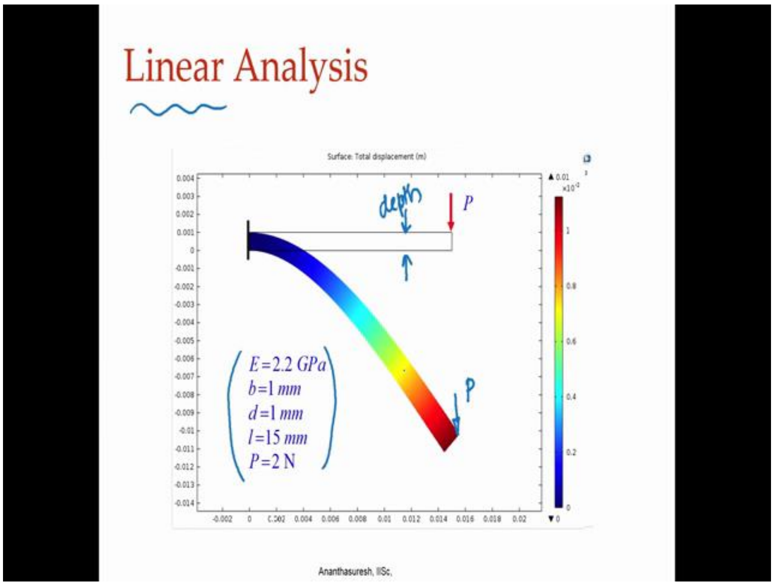


Now, let us go back to our question if we show this particular case where there is a cantilever beam with a tip force like I said last time we should not show force there we actually show force here because only when there is force there is deformation. Here the force is zero in those configurations. Now, let me just erase those so that there is no force there I am just saying that there is no force in the original configuration.

Now, one of them is linear, other is non-linear how do we tell it apart? To the eye when you just take a look they both look similar; this is a result of finite element analysis over which I have traced this curves. So, one is linear other is non-linear if you want to really know which one is which we should actually imagine this line. So, if I look at the first one this point over here moves only in that direction, but not in this direction at all, whereas; if I see here there is a little bit of movement in the axial direction also.

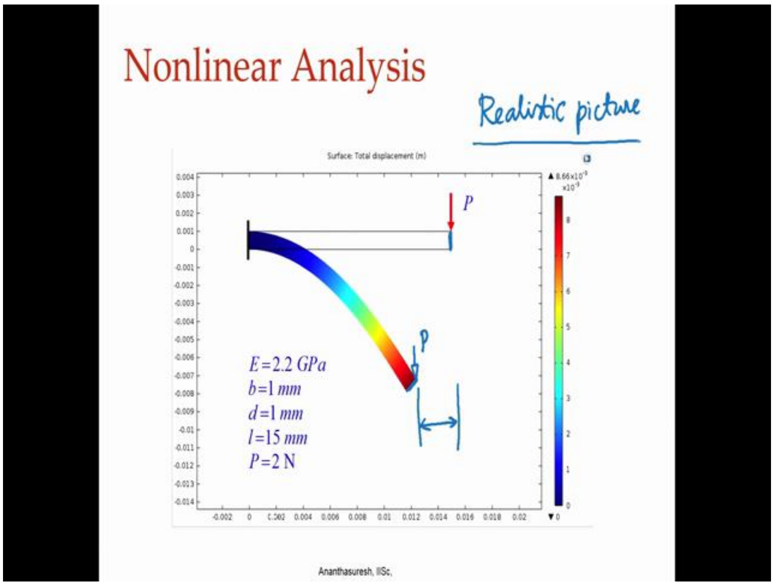
So, that makes this actually non-linear and this is linear. Linear analysis does not actually give beam analysis if you do, does not give any displacement in the axial direction if there is only a transverse force whereas, non-linear would give you that; that some way that you can actually recognize which is linear analysis, which is non-linear analysis. Let us look at this with continuum elements now.

(Refer Slide Time: 05:45)



If I take the same cantilever beam because nothing is actually aligned in reality, only that will model some compliant mechanisms have very slender beams as beam elements, but in reality everything is going to have some width as it is shown here; there is a actually is planar deformation which we call the depth dimension. There is some finite size to it if I model it as it is with some values that are shown here, if I do linear analysis it is going to look like this looks very artificially you do not imagine that a beam such as the one that is shown in the black lines when I applied force P over there that is actually P should be here.

(Refer Slide Time: 06:53)

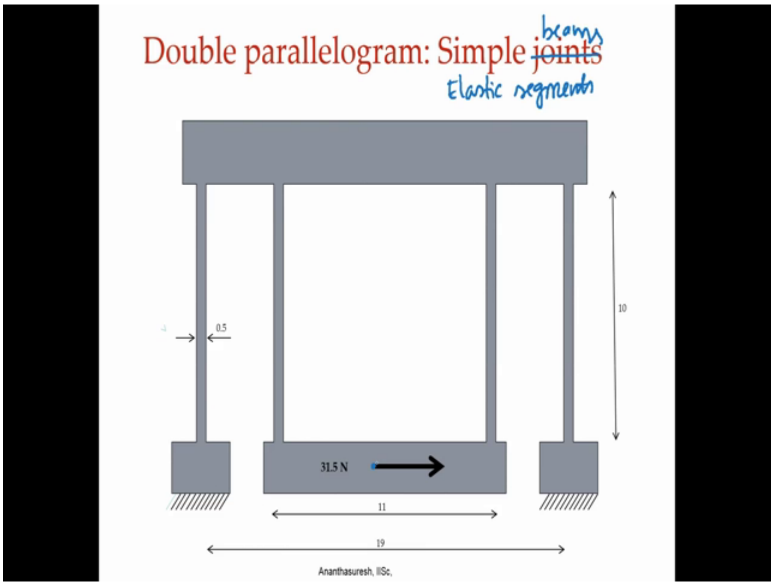


So, in this case P is not there, when P is here you see that it is actually seems to be expanding ladder as if it is heated that is the unrealistic thing that linear analysis gives us when the force is large. The same problem if you were to do with nonlinear analysis usually all finite element analysis (Refer Time: 06:56) we will have an option where you have to click I want to do geometric nonlinear analysis.

So, then that beam here when this force P is there it is actually going to deform that looks realistic when you compare the one that we had before and now the same data all that has been done is to check that box that says consider geometric nonlinearity then you get a realistic pictures. So, nonlinear analysis as we had also said last time gives you a Realistic picture of what happens to the real compliant mechanism.

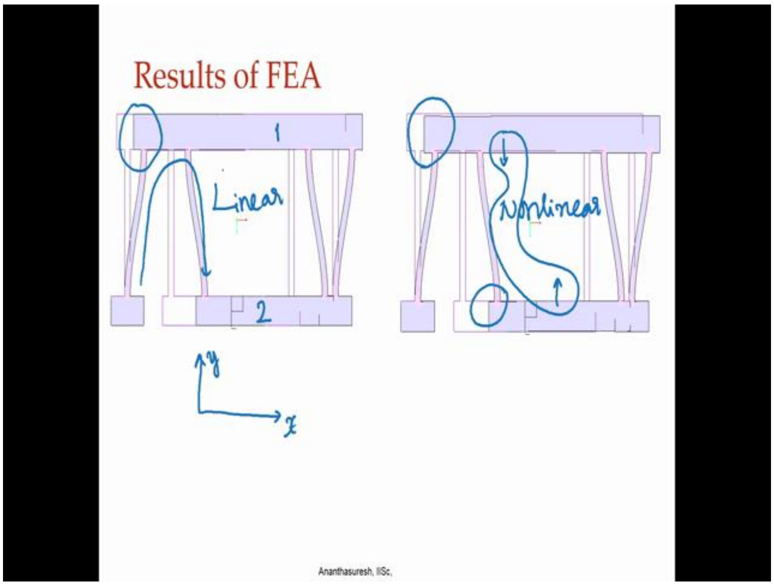
Now, we can understand what we said with beams you can see there is substantial displacement in the axial direction of the beam and you can also see the rotations; what was straight there now it is rotating like this. Which was also apparent a little bit over here, but when you see that it is expanding and unrealistically elongated with in fact, if you see compared to this it seems to be moving in the wrong direction, whereas; here you see it is moving towards the support that is what you actually imagine you hold piece of wire or a strip of metal when you do that the free and when you apply the force it comes towards the fixed end which is the realistic picture that we get the nonlinear analysis.

(Refer Slide Time: 08:31)



So, let us look at this double parallelogram mechanism or what we call folded beams suspension where with this flexure beams when you say simple joints it is actually we are using elastic segments they are actually not joints really they are segments. So, simple beams are slender beams that we have. So, when I have these when I apply a force on this one somewhere a point you can choose and force is apply some dimensions are shown all are in millimeters for our discussion today we do not need to worry about actual values so much because, we are trying to see qualitatively what will be difference between linear and nonlinear analysis.

(Refer Slide Time: 09:23).

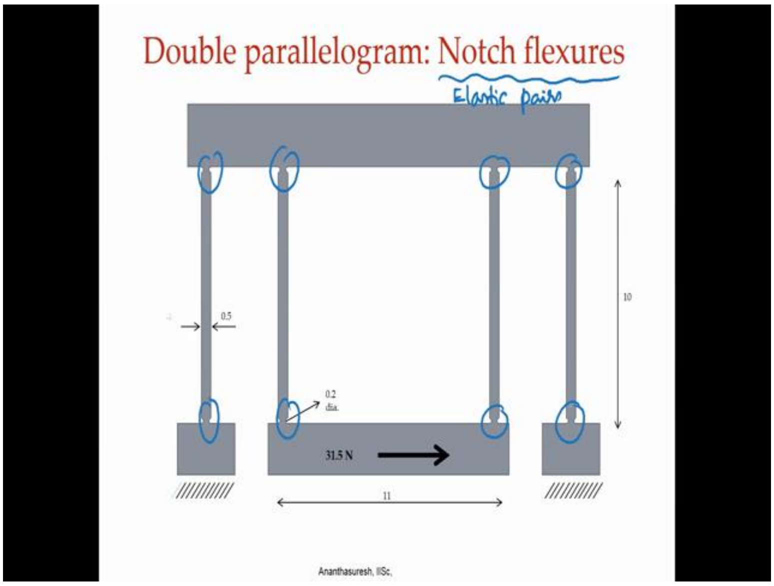


If we take this; so again one of them is linear other is nonlinear how do we tell them apart. If you look at here if you focus your attention there in this case the upper platform is moving purely in the horizontal direction that is this is what I am calling a let us call it horizontal vertical instead of that x and y. It is purely moving in the x direction whereas, if you pay attention here it is moving both in x and y (Refer Time: 09:54) so little for the force that is applied for this platform here, but the qualitative difference is very important.

Again this is linear this analysis is linear and this is nonlinear. This is very important in this particular example because, the reason we go for the folded beam that is this one is folded back here like that the reason is that this platform if I call platform 1 that is going to undergo in reality both motions in the x direction little bit in the y direction; to compensate for that we introduce another one folding this beam back here so that when this platform comes down as you see in the nonlinear analysis we have the other one that compensates now here you do not see.

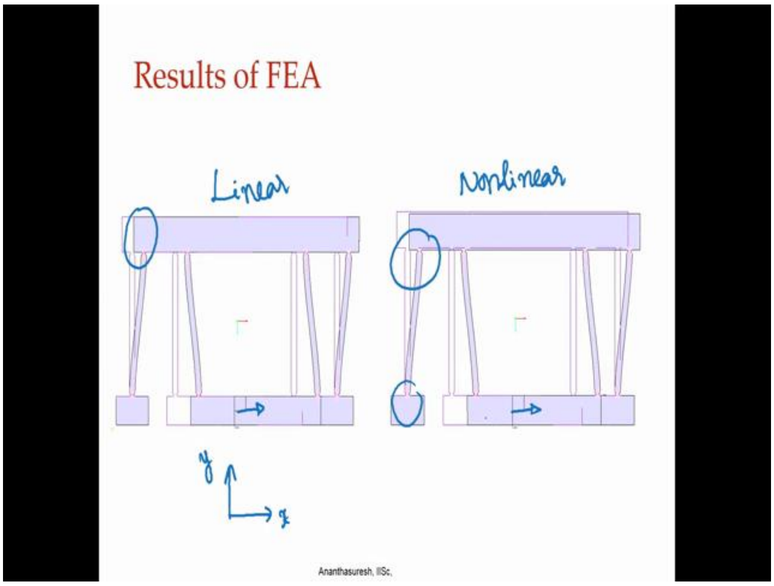
So, this one comes down related to that one this actually goes up; these two things cancel each other and there is no motion. So, (Refer Time: 10:57) linear analysis if you do we are actually being misled that this platform is not actually moving in y direction at all. So, it is important to notice the effects of nonlinearity.

(Refer Slide Time: 11:11)



Now, instead of having those elastic segments if you put notch flexures or elastic pairs as we had called them in earlier lectures elastic pairs. So, we have eight of them and this one and the dimensions are almost the same some circular notch flexures are there.

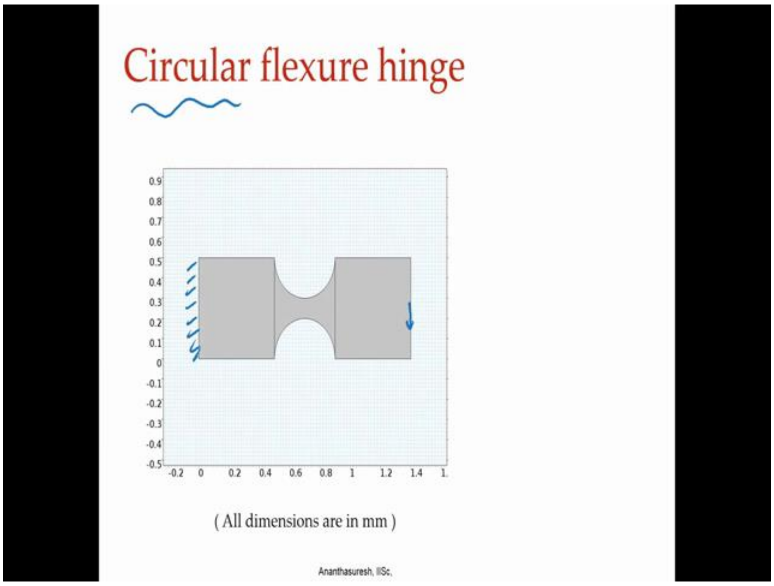
(Refer Slide Time: 11:34)



And this behavior is still the same; there is no noticeable motion in the y direction while there is movement in the x direction when there is force purely in the x direction that is applied on the bottom platform.

In the case of nonlinear there is a little bit of the displacement again; this is linear and this is nonlinear. No need to put a hyphen for nonlinear because it is used so often a word non and linear you can just write as one word.

(Refer Slide Time: 12:14)

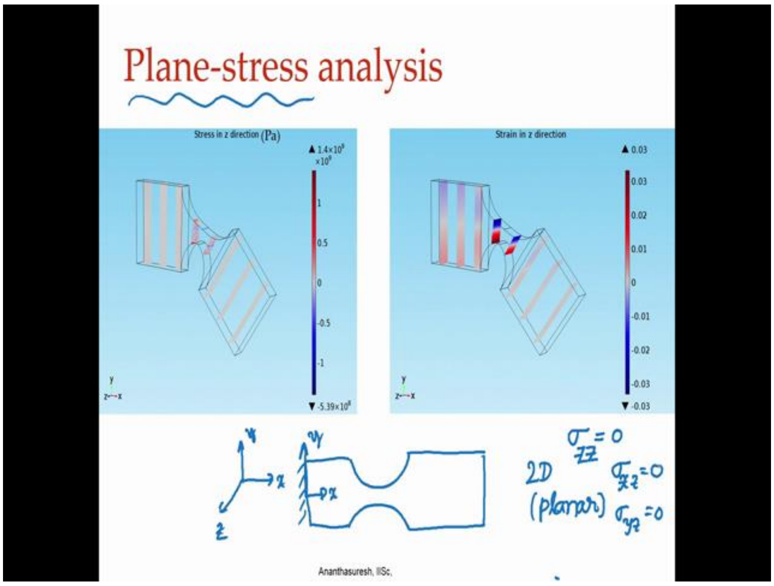


Now, let us look at the circular flexure because that is what we saw in the previous one when you have circular flexure what is the difference how do you model it? Of course, you have geometry when you do in continuum analysis you can mesh it and see, but there is a little subtleties that we should be aware of here if I take circular flexure we had said that need not be circle; it can be elliptical, parabolic, hyperbolic lots of different things that have been done in the literature.

But whatever it is when you have a thing like this if I fix at one end and apply some force on this one how does it deflect and what do you do? When you do finite element analysis you have to choose a type of element; choose a type of element merely means that you are defining how you interpolate the displacements or other quantities within the element because every element will have certain number of nodes and only at the nodes we compute the displacement. Insight that element how do you know we use the shape functions; those shape functions selection is what you do when you use different when you choose different finite elements that are there in any finite element analysis software there will be elements with some name, some code name for it and behind that are the shape functions.

Lately finite element analysis software actually gives you explicitly whether this is a linear interpolation, quadratic, cubic and so forth. (Refer Time: 13:41) so finite element software being one of them. It is a new way of looking at it so that you are mathematically more connected rather than blindly going by the element name.

(Refer Slide Time: 13:53)

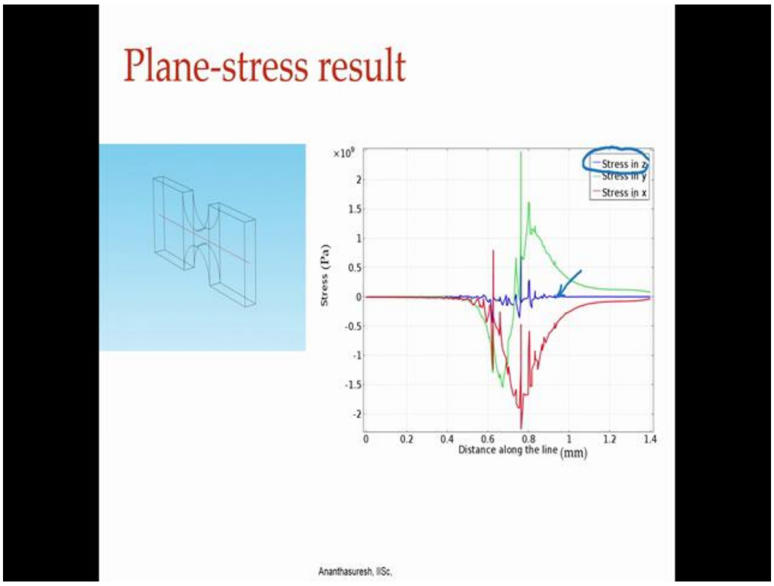


So, which elements do you take? So, here this is shown using what is called Plane-stress analysis. When you have a three dimensional structures such as, circular flexure if you were to take the 3D model and do the deformation analysis you would get the correct result; if you were to use fine enough mesh that something to remember in finite element analysis that you have to use the mesh that is fine enough to give accurate answer. If it is too few elements you may not get the accurate answer. So, you have to mesh it in progressively finer fashion until result does not change significantly.

But if you take 3D thing you have to have a lot of elements and lot of degrees of freedom computation takes more time, but then this structure is essentially planar it is an extruded section, nothing changes is that direction. So, you can use 2D analysis there is something called plane-stress and if you were to do that what; that means, plane-stress condition is that if I were to indicate the axis they say x and y and z is out of plane; let us say my structure here that is the circular flexure that we have other way around it is horizontally drawn.

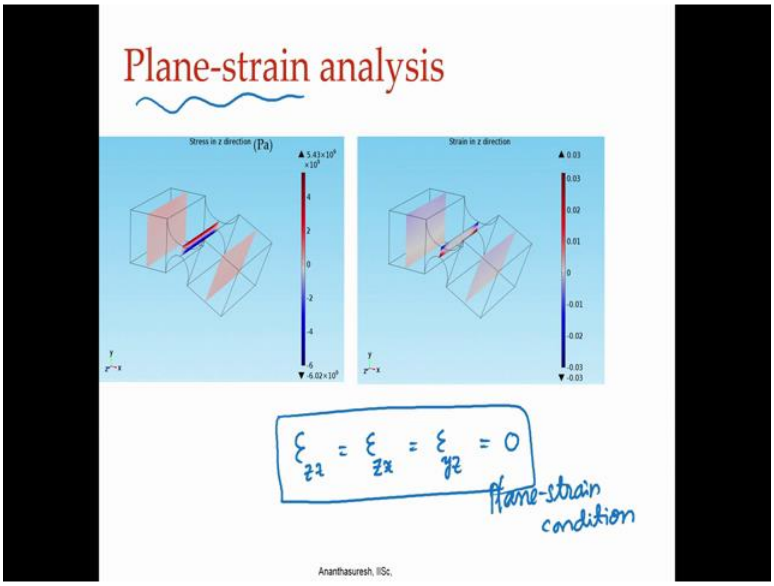
Let us draw it, so that is still in the eraser mode go to a pen. What we look at is circular flexure like this some part like this. So, we are fixing it here and applying force. So, there is thickness, but I want to do planar at 2D analysis because that way I can get away with much less computation as compared to 3D analysis then I choose an element that is say plane-stress what it means is that if this is my x direction and y direction z is out of plane then plane-stress condition means that sigma z equal to (Refer Time: 16:05) in fact, (Refer Time: 16:06) sigma z z normal stress as well as sigma x z and sigma y z shear stresses these are all zero that is what you call plane-stress condition; some structures are of that kind; whenever a structure is a thin then we use this plane-stress condition.

(Refer Slide Time: 16:33)



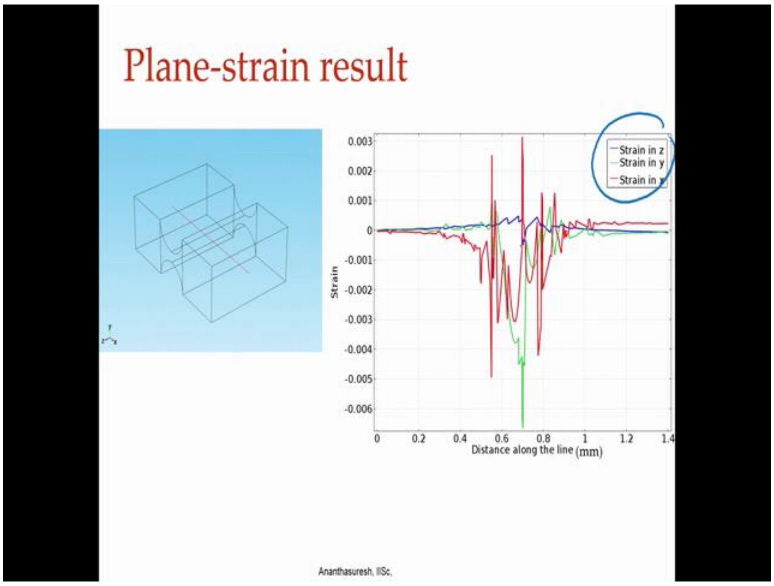
How accurate that is in order to find out let us compare the stresses that exist in this device where you are applied the force here, the stress in the z direction is shown with the blue curve and the stress in y and x directions are shown with green and red. So, now, we see the blue one is much less compared to the green and red; that means, that in this particular one if you were to do plane-stress analysis in the previous slide I showed 2D analysis you would be ok because the 3D analysis when you do and compute stress in x and y these are all only normal stress are plotted here, but if you look at shear stresses also anything that involves that is anything meaning one normal stress and two shear stresses that involve the thickness direction those will be much smaller than the other stresses that involve x and y directions.

(Refer Slide Time: 17:36)



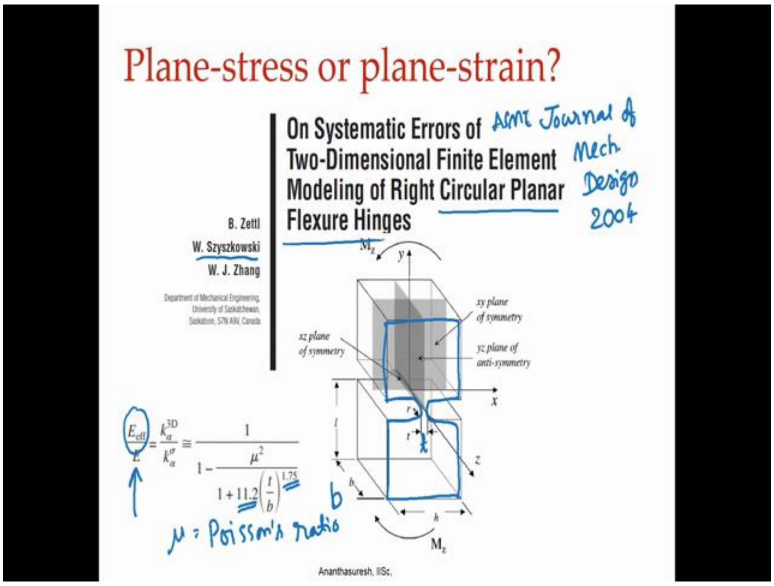
So, you can go; you can get away with a 2D analysis rather than a 3D analysis that is a plane-stress result. Similarly when you have a 2D structure sometimes we do what is called a Plane-strain analysis here, strain in the z direction anything normal or shear those component anyway z x, x z, y z, z y they are the same they are equal. So, this is the plane-strain condition. Plane-strain condition it is an approximation of a three dimensional thing with a 2D problem. So, sometimes you are tempted to use the 2D analysis here if you were to do plane-strain analysis if you have a compliant mechanism let say that is very long like a usually plane-strain elements are used for pipes where along the axial direction of a pipe you do not have much change in the strain because it is so long because this condition is valid.

(Refer Slide Time: 18:39)



So, here if you were to look at the strains in the z direction that is what are plotted here again the blue one is that direction that is much smaller compared to red and green because they are the strains in the y and x directions. So, if you were to analyze this using plane-strain analysis you compared with 3D we see that plane-strain 2D analysis would have been just fine.

(Refer Slide Time: 19:09)



Which one you do, which one you use for a compliant mechanism usually it is plane-stress, but then for circular flexure hinge there was a detailed analysis done by a group in university of (Refer Time: 19:13) there was a Szyszkowski had done that some time ago this is from journal of mechanical design year was this is I have to write that; this is ASME Journal of Mechanical Design year is 2004. You can look that up with the names and you can find this paper where they had done it detailed analysis if we take the circular flexure with various you know there is a b here there is a b and there is a t which decides that space between the two circular features there (Refer Time: 19:57) circular cuts and some other parameters and they had found that you could be getting as much as 12 percent error.

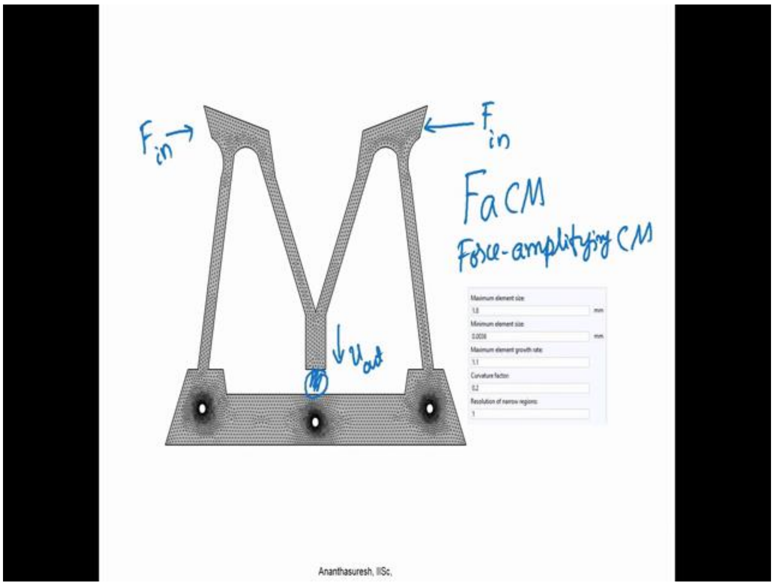
If you were to use plane-stress or plane-strain elements if you take a two dimensional version of this two dimensional version would involve only taking this and not doing the three dimension analysis either way whether you use plane stress or plane-strain (Refer Time: 20:24) error and in fact, they had compared for various values of this b and t how this error changes sometimes you could be accurate very close sometimes you could be very wrong. So, they had done this analysis and you quite interesting and quite revealing that if you blindly assume that all planar compliant mechanism can be just (Refer Time: 20:47) plane-stress elements sometimes you will get as much as 12 percent error or sometimes even more, but they also went a little further to say that if you were to make this change you can still use plane-stress elements because that is always 2D analysis better than 3D analysis.

If you were to change your effective young's modulus using this empirical formula that they arrive that through very detailed analysis of various values of this b and t and other dimensions where this mu here his poisson's ratio in this formula for because 2D analysis that does matter.

So, this is poisson's ratio and this t by b is there raise to 1.75, 11.2 all these are empirical but they had done really hard work to arrive at this formula so that if you were to do this something involves circulars hinges you want to do plane-stress analysis that is, 2D analysis for a compliant mechanism you would reduce the errors if you were to use an effective young's modulus based on your t by b value and the poisson's ratio; that way you will instead of incurring 10 to 12 percent or larger or like that we can reduce your errors and this is only for circular flexure hinges, this elliptical, hyperbola, hyperbolic and parabolic whatever you have to re do it; that means, that you cannot blindly use plane-stress elements.

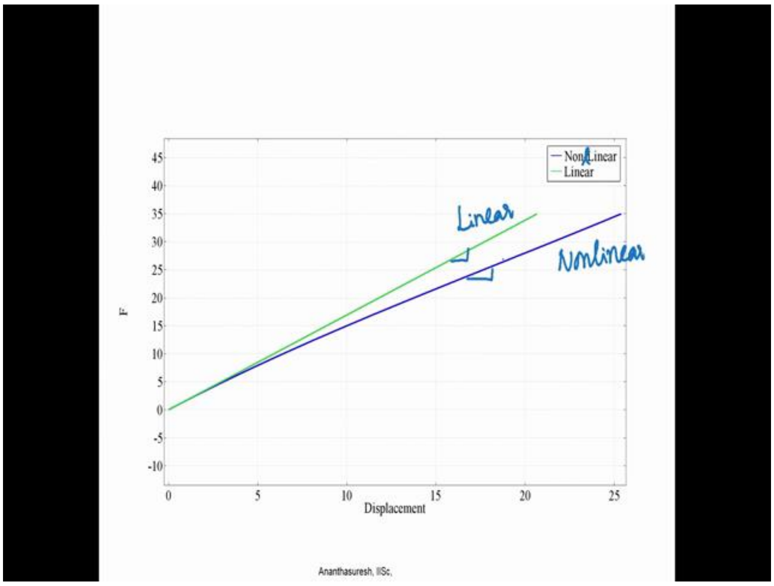
For just because a compliant mechanism is a flat and thin compared to its other dimensions that just a warning or a caveat for us when we analyze compliant mechanisms using this 2D elements plane-stress.

(Refer Slide Time: 22:43)



Now, let us look at some compliant mechanism where there is a very fine mesh that is taken for this where the force is there; this is the input force I can call it F in. It is like a crimping device you have an object you will crush that or apply force on it. So, displacement is here this is the U out. When I take something like this I can do linear and nonlinear analysis how do I know which is which if I look at the deformed configurations of this mechanism actually it be quite difficult to say it because this particular compliant mechanism is what we call FaCM that is Force Amplifying Compliant Mechanism.

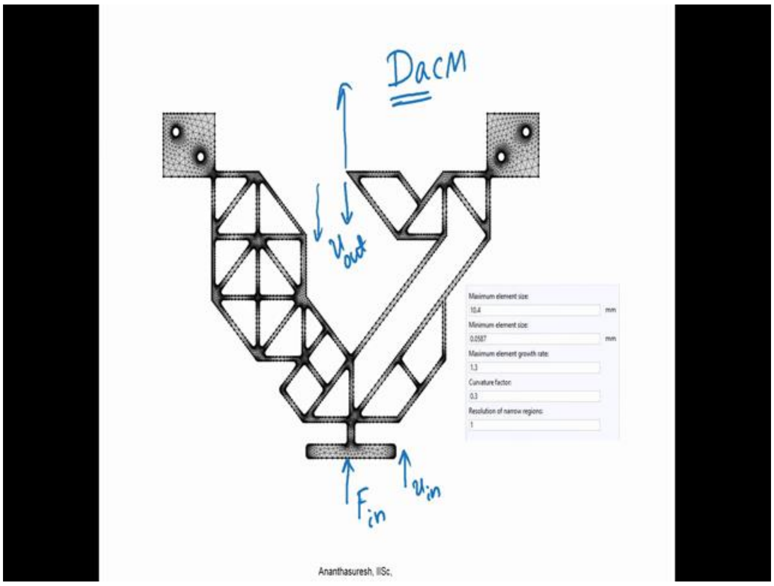
(Refer Slide Time: 23:44).



In general it is a stiff when it is stiff linear and nonlinear analysis will not be very different, but if you look at a quantitative data for this then things are clear this is nonlinear again there is no need for hyphen and this is the linear.

In general for most compliant mechanisms for the same force linear and nonlinear analysis if you what to look at the slope if you see this one and this one it looks stiff for linear and not so stiff for nonlinear, but this means that this particular one in nonlinear case it is (Refer Time: 24:22) meaning it is more flexible, but in general that is not true. The nonlinear analysis will give you stiffer behavior compared to a linear that is a FaCM

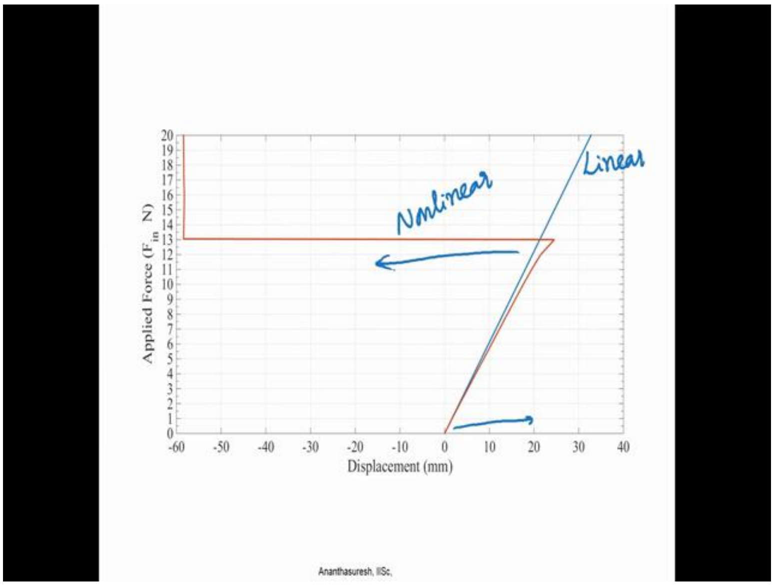
(Refer Slide Time: 24:34)



Let us take a DaCM meaning Displacement Amplifying Compliant Mechanism if I take this one some data is given so that you can repeat it find out and there is a mesh that is shown here.

So, this behavior force versus displacement; so where is force here? Let us see that here is where force is and displacement of this is supposed to be in that direction. Now when I push up like this at this point U in will also be in this direction and we want output to be downward direction; now this force and displacement when you look at here this is Linear and red is Nonlinear.

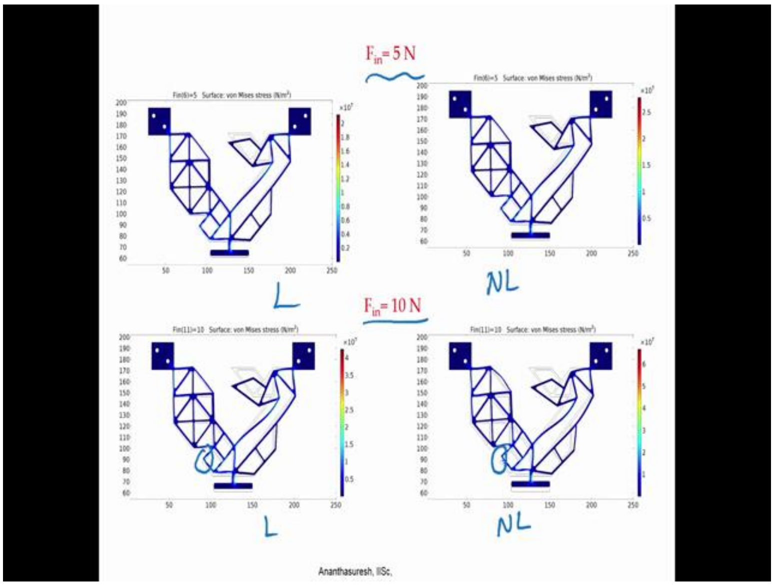
(Refer Slide Time: 25:19)



This is quite dramatic in terms of nonlinear response to begin with itself it is different nonlinear and then suddenly something drastic happens to the extent that the direction itself no it is going displacement is going this way it is increasing to this direction and it reverses what is happening meaning that this particular as I am pushing it comes down in the direction that is intended that is this direction suddenly it changes in goes that way.

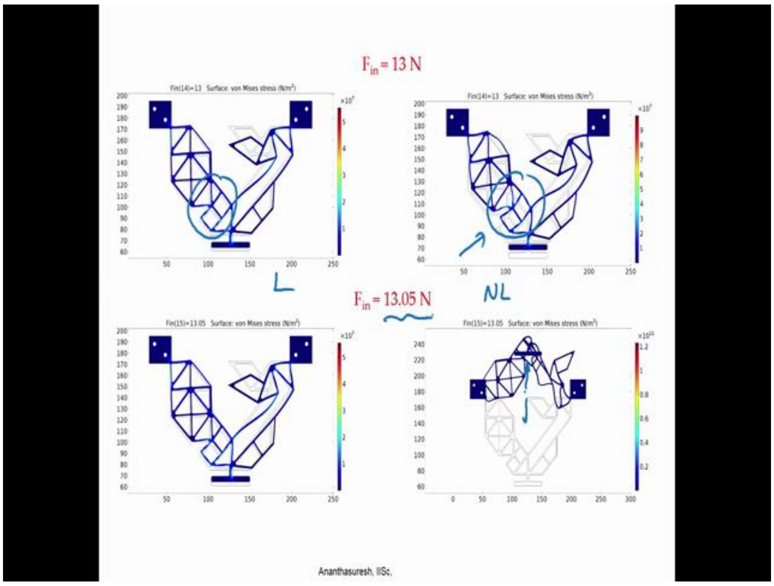
So, what is happening? So, let us look at this analysis at various values of the input force.

(Refer Slide Time: 26:07)



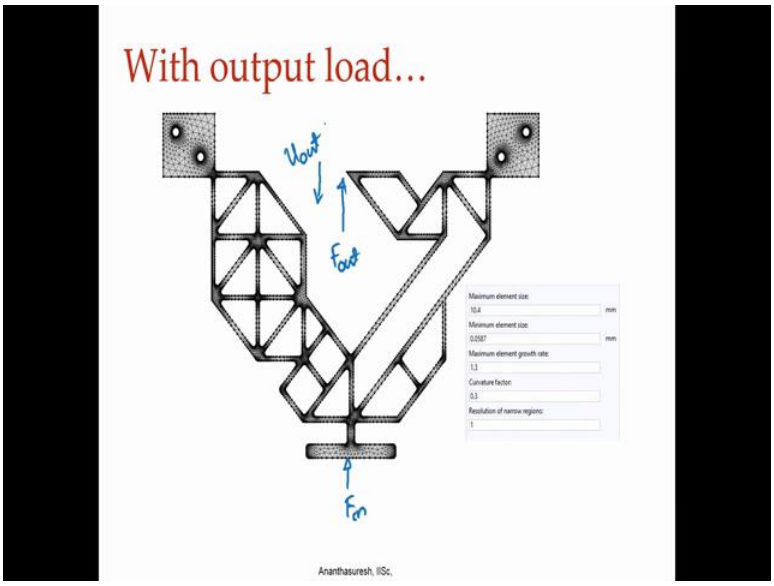
If it is only 5 Newtons are there between linear and nonlinear there is not that much difference that we can actually discerned both of them look almost the same. If I increase the force to 10 Newtons then we can start seeing if I look at this is again linear this is nonlinear there are some differences, but not so apparent even at this point; there are some differences you can we can look at especially this region this has a lot more displacement compared to the linear case. Other places also you can see that there are some differences.

(Refer Slide Time: 26:55)



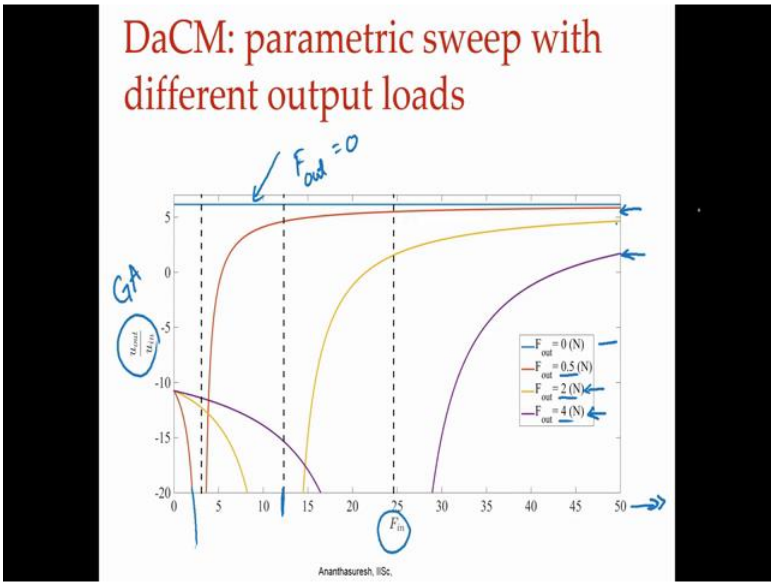
But if we increase the force let say 13 Newtons then we can see especially if you look at this region there is a lot more changes between linear and nonlinear. And this particular mechanism if I go beyond 13 Newtons that is where this dramatic thing happens that linear does not show any catastrophe whereas, here you see this one instead of coming down it has started going up and it is completely meaningless, but this exactly what will happen in reality when I keep applying there is some buckling happening we can see the signs of it are 13 Newtons just a little bit anything that happens suddenly is a catastrophe phenomenon, that is of buckling occurs that happens and that is what we saw in the deformation behavior over here as we increase the force at some point displacement changes direction goes to a very large value at shown there and that we captured only in nonlinear analysis.

(Refer Slide Time: 27:59)



Now, let us say if there is an output load let us say that is output load direction will be opposite to this; let me get the pen back. So, let us say that it is supposed to move down let me put F out here while I put F in here (Refer Time: 28:21) vary the output forces what happens? The displacement as we saw already when I apply force in this direction displacement we want in that direction; it is an inverting thing we are pushing up that comes down, but now there is a output force the stresses then what happens.

(Refer Slide Time: 28:40)

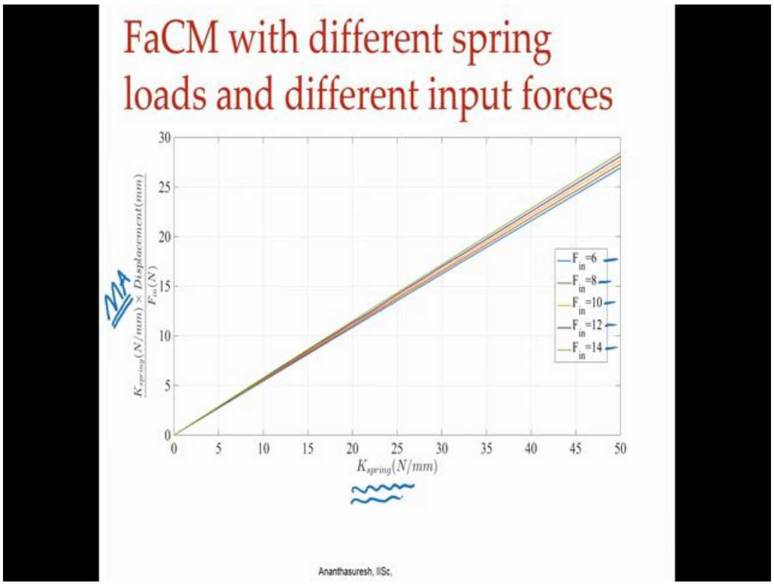


We can include that and do a parametric sweep that is what finite element analysis software enables us to do today for various output forces the blue curve no output force and this one is 0.5 Newtons, 2 Newtons, 4 Newtons and various values of input force up to 50, you see what is happening? If I follow the no output force at all this is the curve that the one that we have that is F output is 0; there is no load then it has geometric advantage U out by U in (Refer Time: 29:17) more than 5.

But when I have 0.5 Newtons initially U out will be in the wrong direction because, if you go back and look at this when applying this since there is output load also when F in is low it is actually going to move up meaning U out is going to be negative that is why this is actually comes down at some point U in is going to be 0; because F in is increasing that is why goes to infinity at some point becomes infinity comes back and eventually catches up and becomes close to what we have. Whereas I increased the output load 2 Newtons then beyond this value again it these where it goes to infinity it is all in the wrong direction U out is in the upward direction which is the negative direction for us and then eventually it (Refer Time: 30:04) you need to go much larger than 50 Newtons to catch up; same thing with output force 4 Newtons we can say that it takes much more input force to eventually go to that geometric ratio.

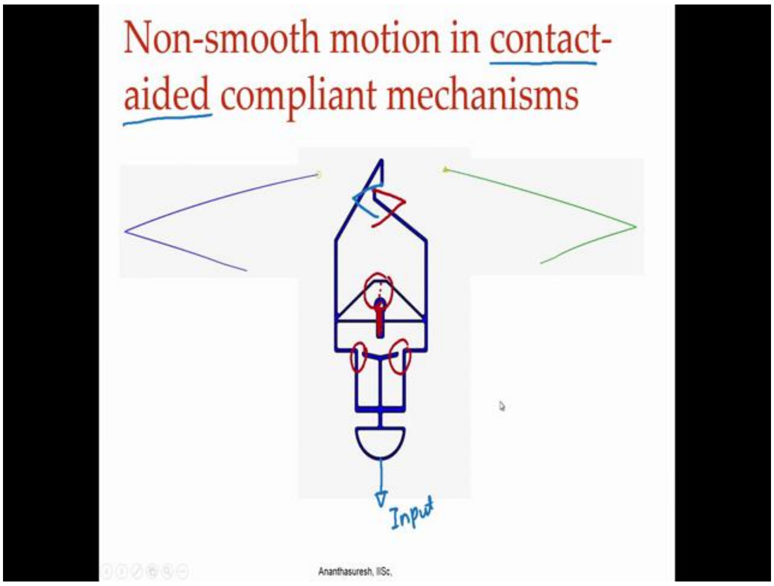
We can see how geometric ratio of varies with various output forces. This is something we can do very easily using finite elements analysis, but we have to remember that we have to use nonlinear finite element analysis for that and if you go back to FaCM; Ican actually put a spring that finite elements analysis software provide this we can call output spring to simulate an elastic object and vary the stiffness elastic object.

(Refer Slide Time: 30:48)



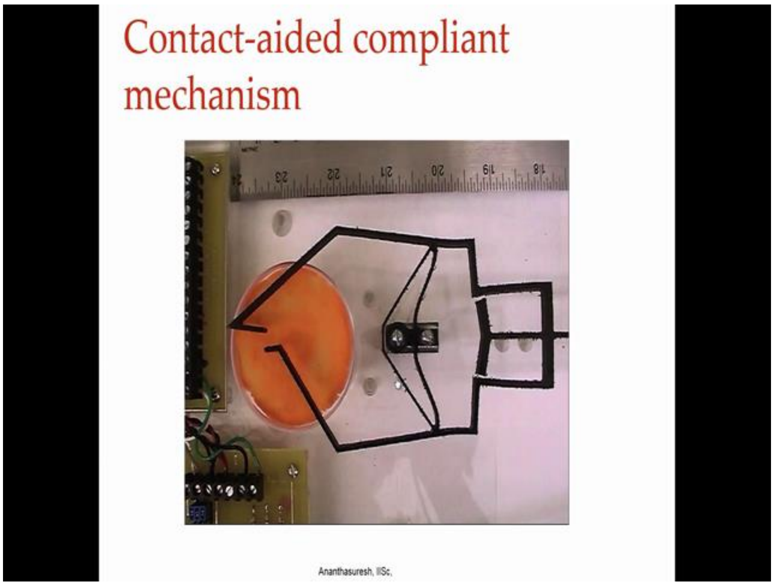
And then see how the mechanical advantage this is the output force that is K spring times displacement divided by F in that is what is plotted here mechanical advantage against stiffness of the spring that is elastic object here for various input forces that is what a color says 6 Newtons 8, 10, 12, 14 here all of them must still linear, but sometimes you could get the like I said this device is quite linear because it is quite stiff. So, it does not going to geometric nonlinear region at all and, but you can see the change in slope as F in increases here as against when you plot the K spring.

(Refer Slide Time: 31:34)



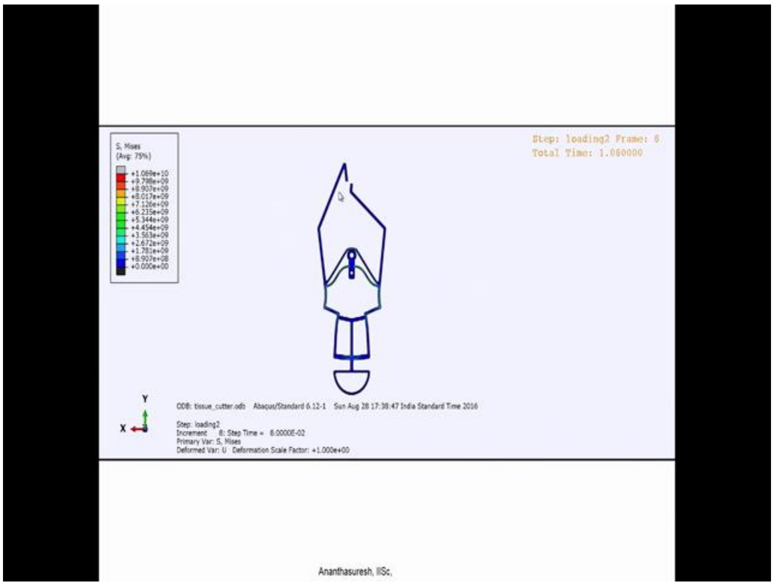
Now, let us go to another kind of nonlinearity which is found in what we call contact aided compliant mechanisms; this particular one the input force is simply here it is smooth input that is you attached to your linear actuator and keep pulling at a constant rate in the same direction, what will happen to outputs here is something strange. So, this one is of course, magnified this point actually goes like this suddenly it is changes it is mind comes back like this similarly this point let me change color for that this point would start there and change come back do this. We will see a simulation under real device here why that happens that is because this particular involves contact this one is fixed here and applied force there this contact as well as contact here and here, this is the contact with a rigid surface, this is contact to with itself.

(Refer Slide Time: 32:52)



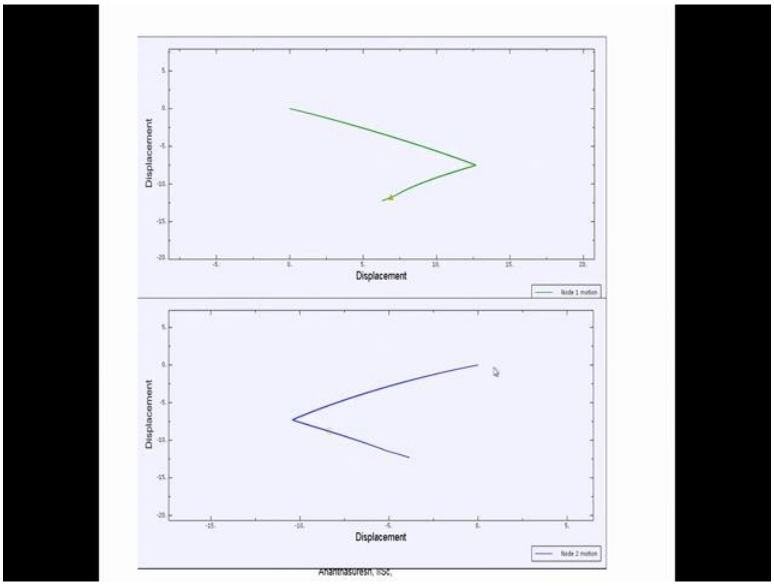
So, when those contacts are there you can still simulate using finite element analysis and see what happens. Let us look at this mechanism in action real device. So, if we look at this device what you just saw the same geometry there is input force attached to a stepper motor and you can see the motion of this. When contact happens here as well as here this output one change their path that is the non smooth path that is suddenly the slope of the locus of this points changes and they undergo a complex motion here even though your input is just pulling something and these are the kinds of thing that you can do with compliant mechanisms any complex motions where your input is smooth output is non smooth. By the way rigid body linkage is have this feature anyway it is very easy to get non smooth motion rigid body linkages here we can show that even with compliant mechanisms it is possible. Now let us look at the deformation (Refer Time: 33:47) the finite element analysis whatever we saw it is contacting here and there and you can see the motion of this; it will play again.

(Refer Slide Time: 33:45)



So, let us play that again. You are just pulling here it goes suddenly changes mind that sharped thing that we see we get that.

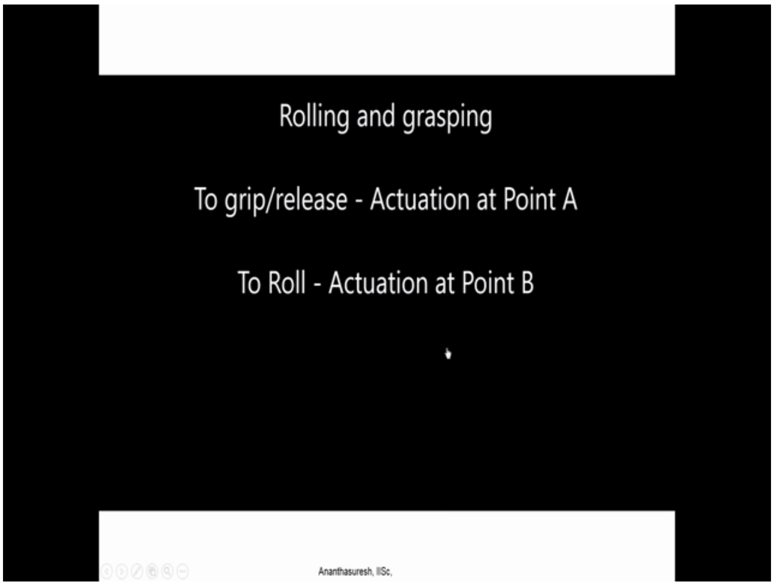
(Refer Slide Time: 34:04)



That is close up motion of those left side and right side; it goes there changes mind comes here this goes in changes of course, these are not really synchronized, but they happen at this point. So, you can simulate these contacts quite well with finite element analysis software today.

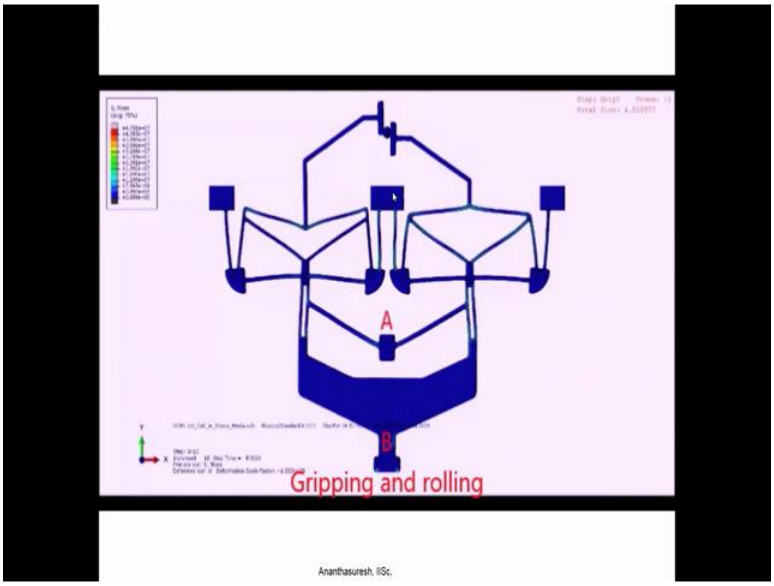
(Refer Slide Time: 34:22)

(Refer Slide Time: 34:25)

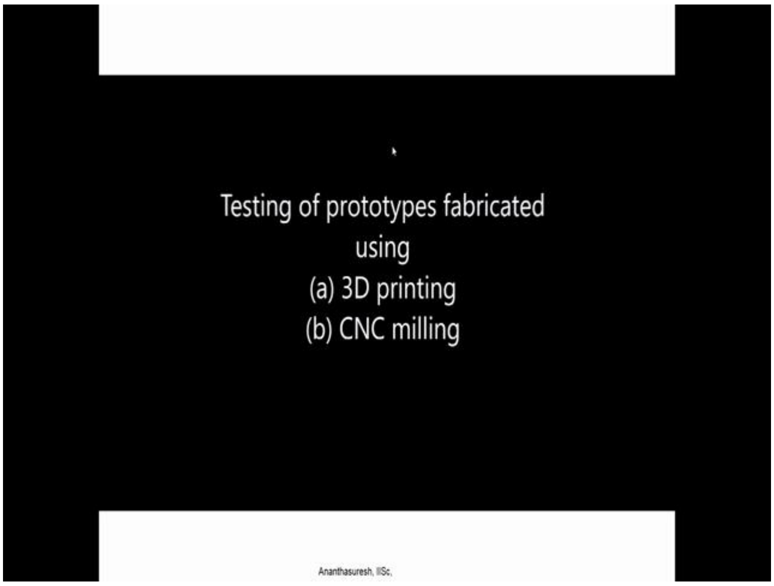


And here is another compliant mechanism which also involves contact and this is a single device with single input are actually two inputs, it is fixed at some point and A and B are the two inputs that you can give it is fixed over here as you can see the outputs here they actually come together come closer.

(Refer Slide Time: 34:28)

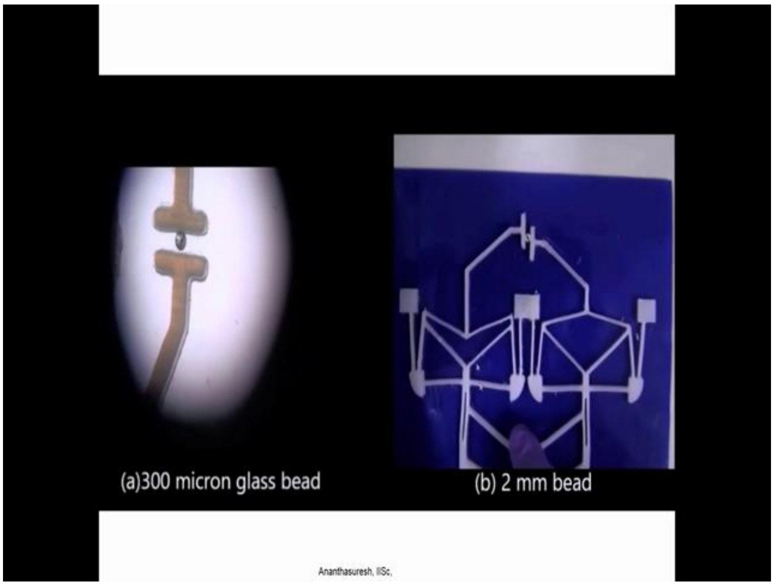


(Refer Slide Time: 34:44)



And then can grasp something and then rotate these are real device that you see your only pulling here it can grasp something.

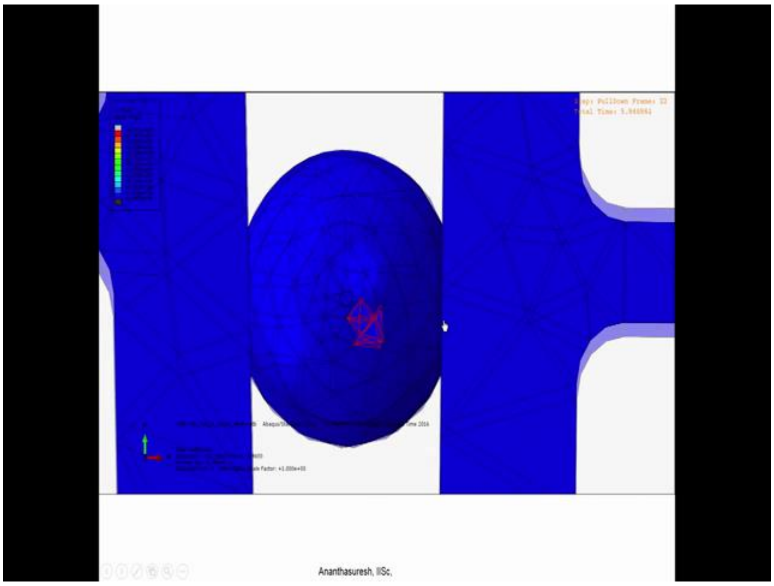
(Refer Slide Time: 34:47)



And role this is 2 millimeter bead this is 300 micron glass bead even at small scales let us play this again.

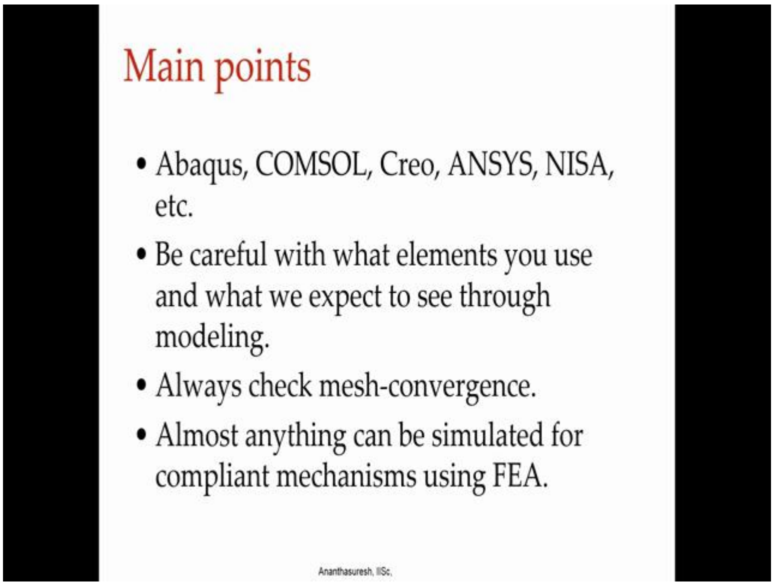
So, you can see a compliant mechanism that can grasp as well as role a device, this was done recently it my group by Harishankar and Krishna Chaithnya with help from all others in our lab. So, you can actually grasp and go, but the motions are only A and B. The way you apply A and B related to one another then you can either get grasping motion or rolling motion of the rigid object it could even be flexible object you can actually see that. Now this is the view of that again perhaps to see how it can role when you apply force on these things.

(Refer Slide Time: 35:43)



And there is a close up view this is the way the contact modeling you can actually see how an object that is freely suspended can be rolled up with this and (Refer Time: 35:53) we are looking under close up view. So we can do just us about anything with finite element analysis software today.

(Refer Slide Time: 35:59)



So, let us just summarize what we have discussed we can use any finite element analysis software. It does not matter which one you use whatever you are comfortable with you can use that the results that I have shown over done with Abaqus, COMSOL, Creo and some with ANSYS and NISA. So, these are the ones that we use in our group but just about anything is fine, but you have to pay attention what kind of elements that you use and make sure that you check that option of nonlinear geometric nonlinearity contact is there you have to do define contact elements and do kinds of things that we saw in two examples today. And we have to be very careful with the element type as well as the mesh convergence, you have to use sufficient number of elements to make sure that your answers are close to reality and let us note that almost anything can be simulated using finite element analysis within the context of compliant mechanisms and if we were to do 2D analysis of a 3D compliant mechanism; well compliant mechanism is always 3D because real ones will never be 2D may be in modeling we can do with 2D. When we use plane-stress elements we have to be careful that particular paper that I sited where the reported that errors can be as much as 12 percent.

So, there is a correction that you can do by using a an effective young's modulus based on t by v ratio in that thickness a circular flexure hinge and the breadth of the element or the breadth of the flexure and making sure that they have given a range actually if it is less than 2 or greater than 12.5 what will happen and that has to be paid attention to.

Thank you.