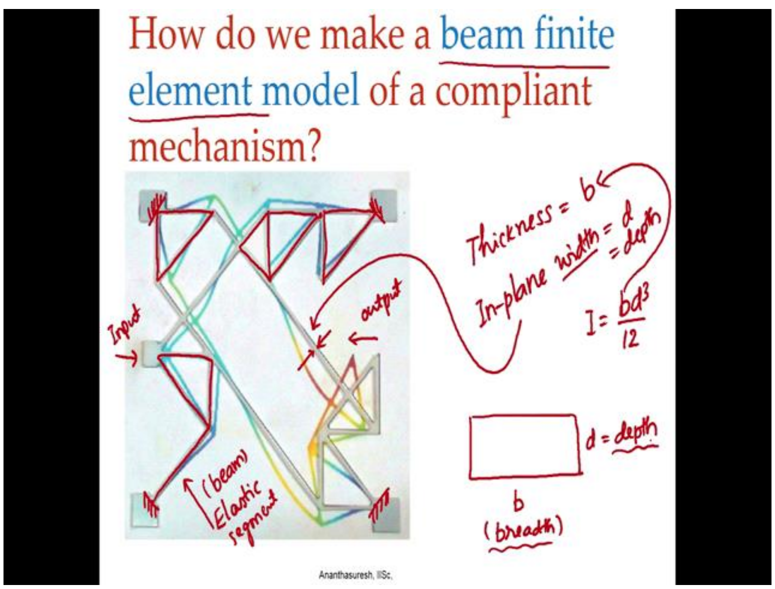
Compliant Mechanisms: Principles and design
Prof. G. K. Ananthasuresh
Department of Mechanical Engineering
Indian Institute of Science, Bangalore

Lecture - 10
A compliant mechanism kit

Hello, today we are going to look at a special case of finite element analysis for compliant mechanisms which actually leads to what can be called a compliant mechanism kit, a kit meaning like a Lego kit with which you can assemble using intuition compliant mechanisms. It is actually a way of designing compliant mechanisms let us look at the basis or premise for that compliant mechanism kit by starting with finite element analysis of compliant mechanisms, let us look at the slides to see how this idea for the compliant mechanism kit came about by looking at finite element analysis of compliant mechanisms.

(Refer Slide Time: 01:04)



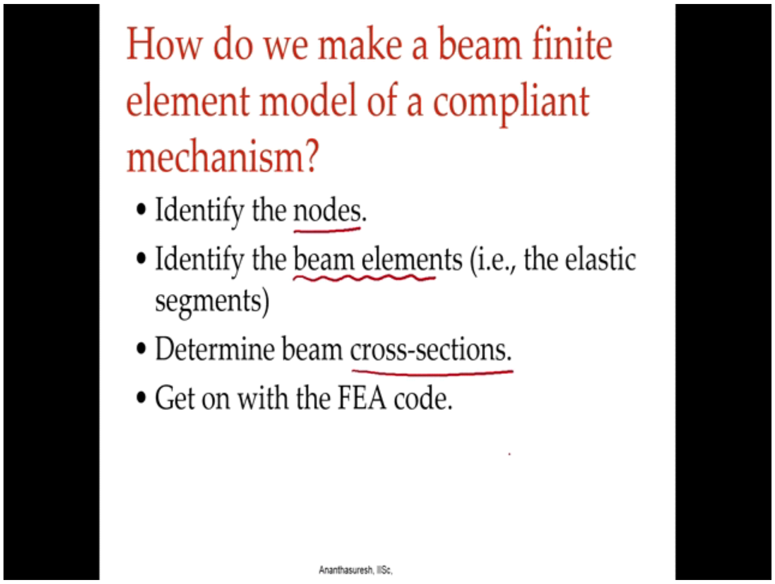
If you look at the mechanism that is shown in the slide, if you look at this everything is like a continuum in fact, what we have done here is we have taken a sheet of polypropylene and using CNC machining have cut out a lot of holes that you see this is this triangular thing is a hole, this rectangular thing is a hole, this is the another triangle, another triangle, another triangle and of course outside if you cut out all that material you get from a continuum sheet of polypropylene; you get a compliant mechanism and that is been pasted on to a paper on which the finite element analysis result was printed.

So, the one that you see in the color that color here represents displacement so, points that have moved a lot are in red point that are not moved at all are in blue. So, it do not see that moving, it is all blue color little bit of displacement is in blue. So, it is color coding for the displacement. So, if I apply force over here from there if I move it to here so this point moves over there this is going to move from here to here. So, this is input this is output.

So, if we look at a mechanism such as this here this is the input and this is the output now, I want to model it one way to do this is to model using continuum elements itself meaning the 2D elements which could be triangles that in is constraint strain triangles or quadratic interpolated kit triangles or quadrilateral elements and so forth. That is a continuum base basically you model the entire amount using entire amount of material using continuum elements, but then we also want to do that with beam finite elements which we have seen in the last lecture that our own math lab code or your own math lab code or your own code in any language, we can simulate and try to look at the behavior and later on we will also look at how to synthesize compliant mechanisms using either beam elements or continuum elements now, I want to make a beam finite element model out of a continuum compliant mechanism such as the one shown on the slide. So, I have let us say there is a beam segment here.

So, that is what we call an Elastic segment are for us it is beam. So, beam segment is there. So, I can take a node here and a node there and say that that is my beam element and likewise, I see that there is an another beam element from here to here, there is another beam element thing this part is input that is where you apply the force and here is another beam everywhere there are beam elements, we can identify the nodes and make this and this one because that is fixed in fact, in this one these four corners are fixed in this particular mechanism this four corners are fixed; we can make beam elements like this. We identify all the nodes and connect them to make our finite element model just like what we had discussed in the last lecture with math lab code. So, that is how we do it.

(Refer Slide Time: 04:49)

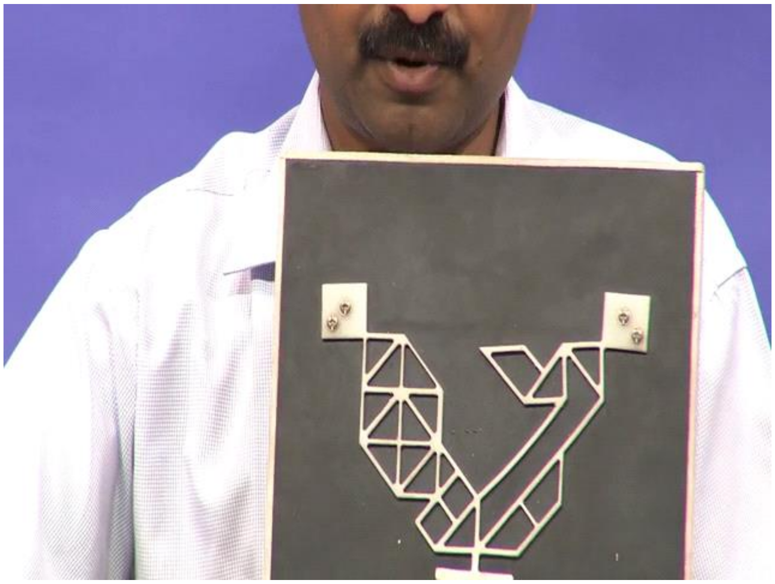


So, the first step is to identify all the nodes that are there. These are the vertices where different beams segments meet, once you identify the nodes you join the relevant nodes using beam elements or elastic segments each of them will be a beam finite element, and then you also have to specify the cross sections of all these beam segments if you go back to the figure that was shown in the previous slide all of them here happen to have more or less a same width and there is some thickness because, we took it 2D sheet of polypropylene and cut it out. So, there is a thickness that in this case is uniform and here the In-plane width which is if I have beam element how much is that that is the in-plane width, that I can measure that need not be the same for all the elements whatever it is you note it down and give as a cross section in this case we have rectangular cross section what we normally indicate this as b and d.

So, the second moment of area is going to be b d cube by 12, there is always some confusion as to which is b which is d. So, I would like to say that let us use the notion of breadth of the element which is like a width this in-plane width instead of using width I call it a breadth, because in this particular case in-plane width is actually d this is the depth of the beam, depth is a cross section dimension which is in the plane of bending for example, this mechanism is moving in this plane that is shown that is plane of the screen and the dimension here is actually d the depth of the beam, that is why whether the thickness in this case thickness happens to be the breadth.

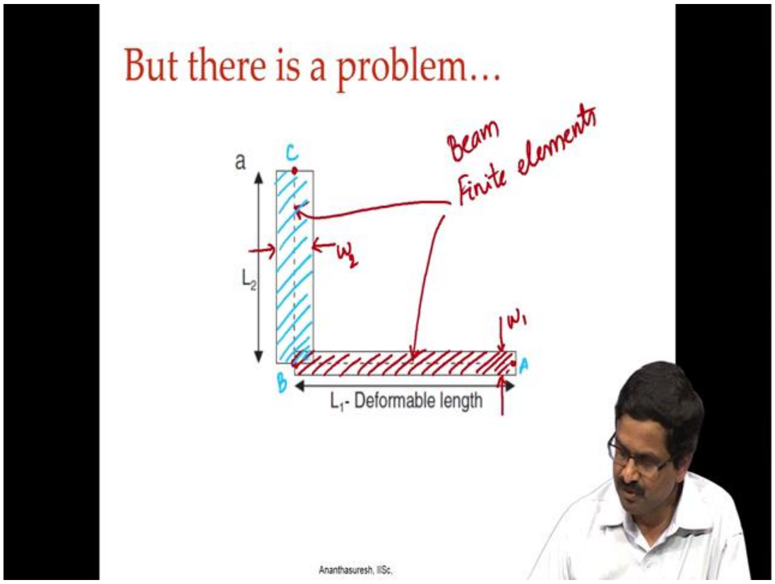
So, b d cube by 12 in this particular case b is actually the thickness and d is in-plane width. So, it is better to stick to the terminology breadth and depth. So, depth is a dimension in the plane of bending and the breadth is a dimension perpendicular to the plane of bending that is thickness. So, you have to identify this for the compliant mechanism and you input a cross sections and of course, material properties, forces and fixed boundary conditions and so forth gnd get on with the finite element code that is a procedure it is fairly straight forward given a compliant mechanism, let us say I have one here that I have already shown couple of time earlier it is a compliant mechanism. So, when apply a force it is deflecting.

(Refer Slide Time: 07:47)



So, I can look at something like this identify all the nodes and all the elastic segments and know the cross sections with depth and breadth known then I can put it in the finite element code then I can analyze as we have shown.

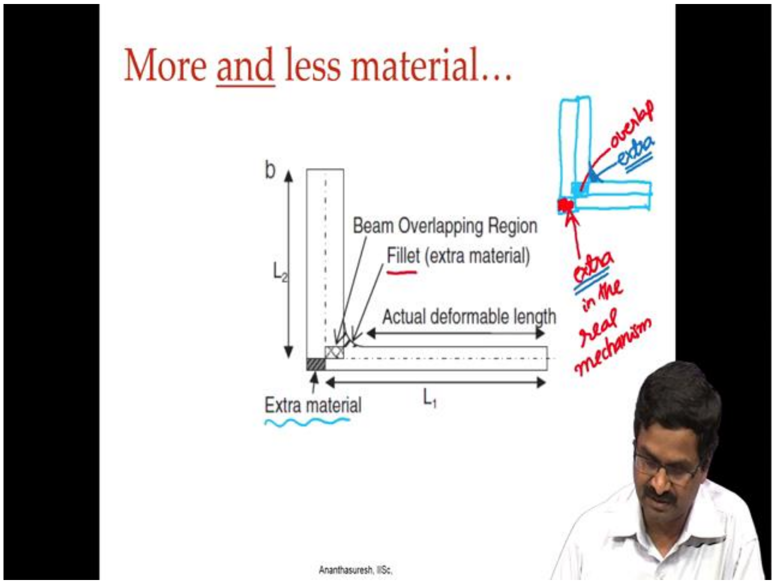
(Refer Slide Time: 08:08)



But there is a problem and let us look at what the problem is? The problem is here as shown on this screen what we have shown in dotted lines there is a node there, there is a node here, there is node there, basically two elements I am showing; three nodes and two elements, if these what I have given to finite element that is the dash line. Dash lines are my finite element beam elements in this one and that one or finite I have should say qualify them as beam finite elements. When I have this beam finite elements and there is a width here in-plane width let us call it w 1 which by the way is depth for this beam and likewise for this one here there is another w I can call it w 1 this is w 2.

So, this is what I am modeling as beam elements the dash lines with three nodes, but physically if you think about there is this entire beam that we are imagining when I give w 1 as my in-plane width for this I am actually modeling all of that material likewise, if I choose another color this particular one over here because the width is w 2 which is shown to be little larger than this I have model that entire rectangular beam when I put this dash line from if I call this node let us say this is A this is B this is C, B C node B C are the nodes the line connecting them is the beam finite element and modeling is all this blue colored (Refer Time: 10:11) region now, you see that there is a problem interpreting the continuum here.

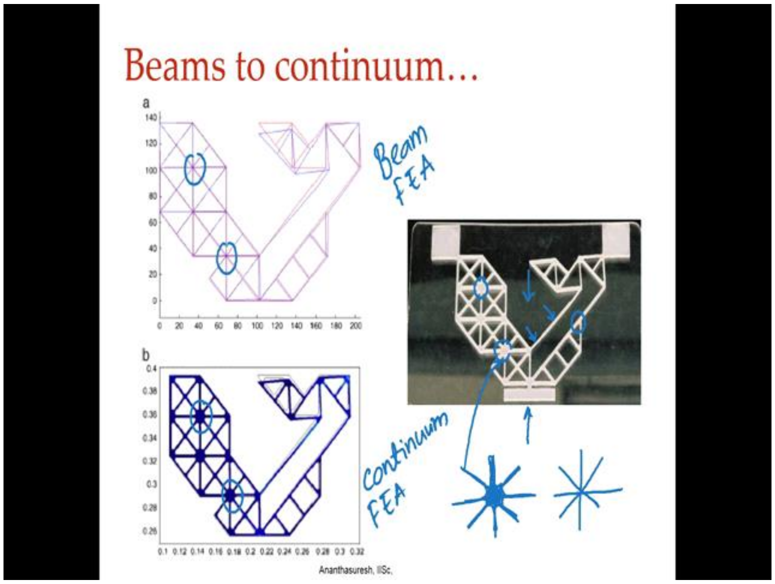
(Refer Slide Time: 10:18)



So, let us look at that if I have something like that I have some Extra material that is coming here that extra material which is not really what was intended for us because we just had one element, one node there and then another one and another one here I only had this the three nodes and two elements I am really worried about what is here. So, when I join these two when this one and that one are overlapping I get some extra material, but I also have this overlapping thing here. So, when I say extra material if I take this width here and then take width for this one little smaller let us say I take this, there is a overlapping region I am modeling it twice and the other hand when I make the real structure are not going to make it like this I am going to fill this up. So, this portion let me use a different color for it say red color at this particular is going to be extra in the real device extra in the real mechanism that I am going to make that is extra whereas, this one is overlap accounted twice this overlap region is counted twice.

So, there is a problem in terms of what material we are actually modeling; there is also another issue which is with the Fillets. So, whenever you use a milling cutter or any other way of making a let us say milling is what I mention, when I do that the milling cutter radius will come like this and this material will also be extra in the real device this will be extra. So, there is a portion of material that is counted twice there is a overlap between the two rectangles which are created from the beam elements that we have specified these beam elements are line elements and we also have some extra material coming because of the outer thing over here and also due to fillets over here.

(Refer Slide Time: 12:51)



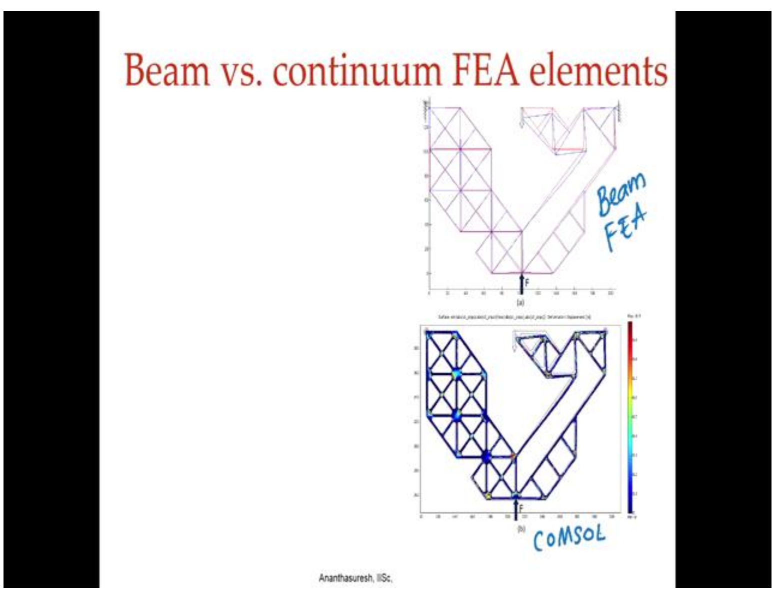
So, actual mechanism and the one that we are thinking we are modeling with the beam elements are actually different, it is not just about the amount of material it is about the effective length of this and that we can understand when we look at a real mechanism the one that I just showed you whereas, said we apply force it is going to move like this.

So, if I look at any of these things let focus on that. So, if you zoom in and look at it that actually has a lot of material whatever I talked about extra material any of this places it is not like let us say I look at this one it is not like I have so many elements from all this directions meeting at that point. It is not like that actually this is also there all eight are there at that point, but at the center there is something like this, that comes naturally when you make it out of milling machine or (Refer Time: 13:43) whatever now all these are beam elements. So, these are all beams of different widths even though you imagie in finite element it is more like this, that is how you would model, but in the real structure is going to be this big continuum there.

So, when you go from beam to continuum you have a problem meaning that we have extra material coming in the real device where as that is do not there. So, if I look at any one length in fact, in this particular case you see this beam element and this beam element have different widths they can have different in plane widths, but no matter what the those are whenever two beam elements meet at an angle 2 or 3 or any number maximum 8 as it is shown we have this extra material and that leads to some error.

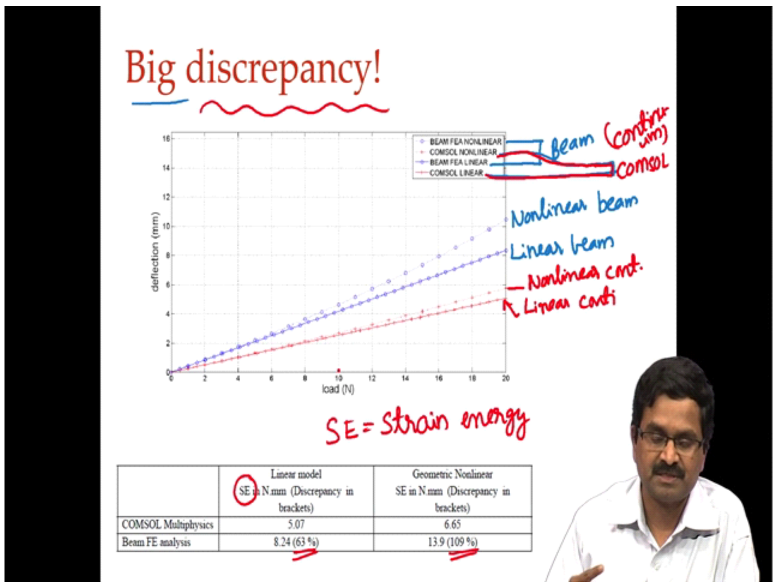
So, I take this mechanism the beam element model it is going to look like this is the Beam F E A this is Continuum F E A; continuum finite element analysis that is F E A finite element analysis. You see all these things that we have in the real structure or over there. So, whatever I have circled in this picture I have circled here also. So, there is a lot of material, but just absent if I look at the corresponding things they are like that; obviously, when there is this mismatch between the model and the real device the results are not going to agree that is what we see and surprisingly the error sometimes could be very very high. So, in this particular case this is with the Beam F E A that we had discussed in the last lecture.

(Refer Slide Time: 15:23)



This particular one was done using COMSOL Multiphysics software you can do with that with any finite element analysis software (Refer Time: 15:42), COMSOL, (Refer Time: 15:44) many other things that are there (Refer Time: 15:45) anything that you do if you do 2D analysis with continuum elements it is going to look like this.

(Refer Slide Time: 15:58)



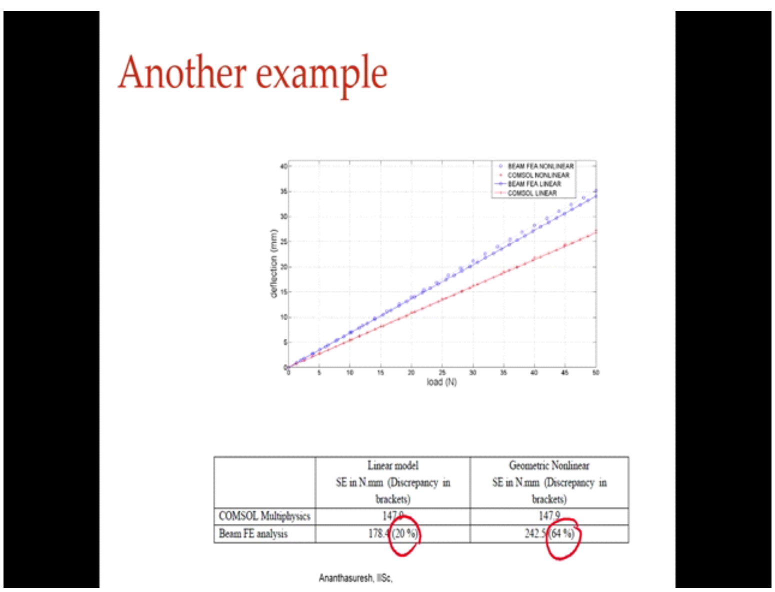
	Linear model SE in N.mm (Discrepancy in brackets)	Geometric Nonlinear SE in N.mm (Discrepancy in brackets)
COMSOL Multiphysics	5.07	6.65
Beam FE analysis	8.24 (63 %)	13.9 (109 %)

Now, let us compare what is the difference between these two and that is what I am showing and there is Big discrepancy. So, we are comparing here both linear as well as nonlinear finite element analysis. So, this blue ones are with the beam finite elements, these two are the beam finite elements, there was a dotted line is nonlinear, dashed line is linear. So, this is Linear beam elements linear analysis using beam elements this is nonlinear analysis with beam elements and the red ones here let me actually change to the red color so we can see it better.

So, this is COMSOL and did not change the color let me change to red. So, the red one there and red one here are COMSOL or any continuum I should say it is nothing to do with the specific to continuum F E A. So, those are here again this is nonlinear continuum and this is linear continuum. You can see that there is difference if you apply some load 10 there is difference initially there is smaller error as we go to higher loads there is a big difference, if I look at some quantity like strain energy this S E here is Strain energy stored in the structure that is one global measure for the entire compliant mechanisms, if I look at the strain energy in COMSOL Multiphysics and beam finite analysis linear and nonlinear are shown in this table we can see there is as much as 63 percent difference and here 109 percent if you go to geometric nonlinear that is your error in strain energy is almost 100 percent more than 100 percent, 109 percent in nonlinear 63 percent.

We cannot tolerate such large errors in analysis between experiment and analysis theory we can say be 5 percent is tolerable or 2 percent is tolerable, 10 percent tolerable, but when you have 2 different analysis they differing as much as 60 or 100 percent is just not acceptable you know the reason the reason for this discrepancy here is that our modeling is not, when you model a compliant mechanism with beam elements you have to be really careful as to how we are modeling because when more than 2 beam elements or even 2 beam elements meet at a node there is going to be a rigid mass at the center, and at the intersection of the of the 2 beam elements unless you model that correctly you are not going to get the correct result, which may not be apparent when you look at the analysis figures they both look the same they are deforming the same and that is misleading.

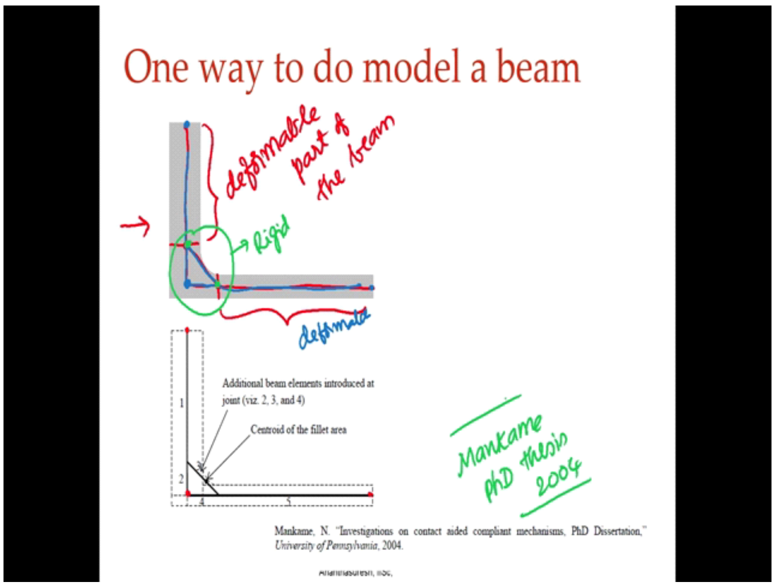
(Refer Slide Time: 19:35)



	Linear model SE in N.mm (Discrepancy in brackets)	Geometric Nonlinear SE in N.mm (Discrepancy in brackets)
COMSOL Multiphysics	147.9	147.9
Beam FE analysis	178.4 (20 %)	242.5 (64 %)

The quantification when you do it that is when these errors will be apparent and here is another example that is based on a different mechanism that I do not have which mechanism it is, but here the error is 20 percent and 64 percent. Sometimes the errors could be low, but even 20 percent we cannot tolerate, sometime maybe only 5 percent but the problem exist the way you model a mechanism is causing this error and that has to be fixed.

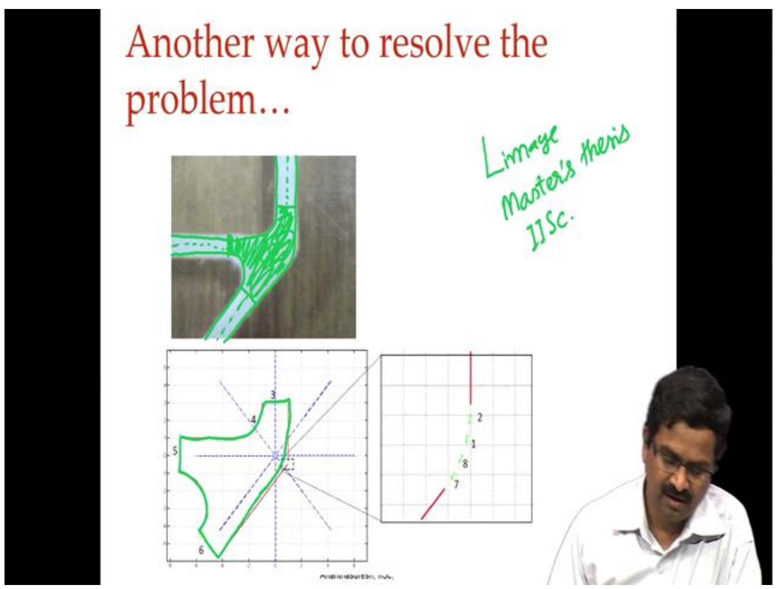
(Refer Slide Time: 20:05)



So, one way to deal with this situation is that when I had a realistic structure that looks like this. So, I have a compliant mechanism that has segments like this now one way to model this is to have a node there and a node here and a node there, that is I put a node and a node and a node there, but the fact that there is so much more material at the center and the effective length of the beam.

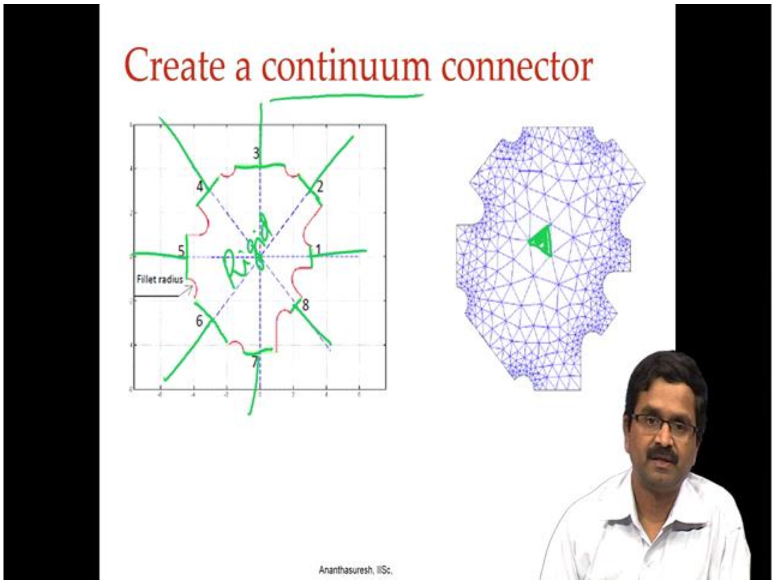
So, in fact, what will happen is if you make a beam element you take this entire length and say that the deforms and you said this entire length is going to deform, but that is not the case there is so much of rigid thing here effectively only this part of the mechanism is going to be able to deform and same thing over here only this is the deformable part of the beam here even it is long. But the fact that there will be so much more material it would not. So, one idea is to connect these two, create an extra node and extra node here. So, let me indicate that extra nodes in different color. So, I put a node here, node there and node there and I put extra node an extra node and connect this. So, I make element like this. So, now, I am making that only this is the deformable part whereas, this portion here this thing let me choose a different color this portion is more or less rigid and that is what we find in any compliant mechanism we model the beam element that is the rigid portion where things intercept, but then where do you take this node where do you take this node this is all ambiguous, but this anyway this was done by Nilesh Mankame in his PdD thesis at University of Pennsylvania back in 2004, see he had done this and kind of improve the model using beam elements.

(Refer Slide Time: 20:26)



Little later we figured out another way this was done by Padmanabh Limaye in his Master's thesis at in IISE, you can find this Master's thesis report. So, what he did was if I have a compliant mechanism such as this which has this type of thing that results when you cut it out of CNC machining he decided to model that as it is; basically a portion that you think it is going to deform. So, you have this portion this entire portion is kind of rigid. So, he modeled it as it is and only here you have beam elements with dotted line that I am doing the beam element.

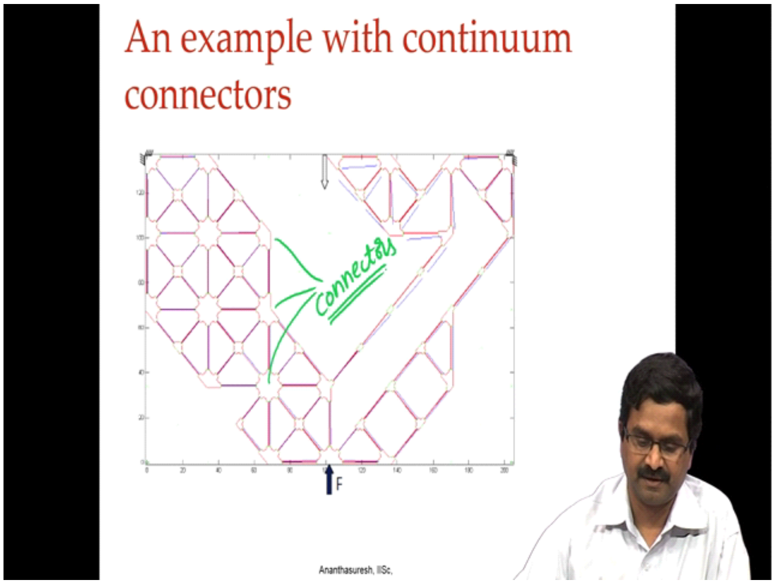
(Refer Slide Time: 23:29)



So, this connector that you see is rigid one with all the fillet radiate taken and that is modeled as rigid element along with beam elements. So, if there were to be eight things meeting at a node such as this they all have different widths. So, this width is more than this width and that one is more than this and everything has different widths here is where the beam elements go. So, a beam element goes there, goes there, goes there, goes there and so forth.

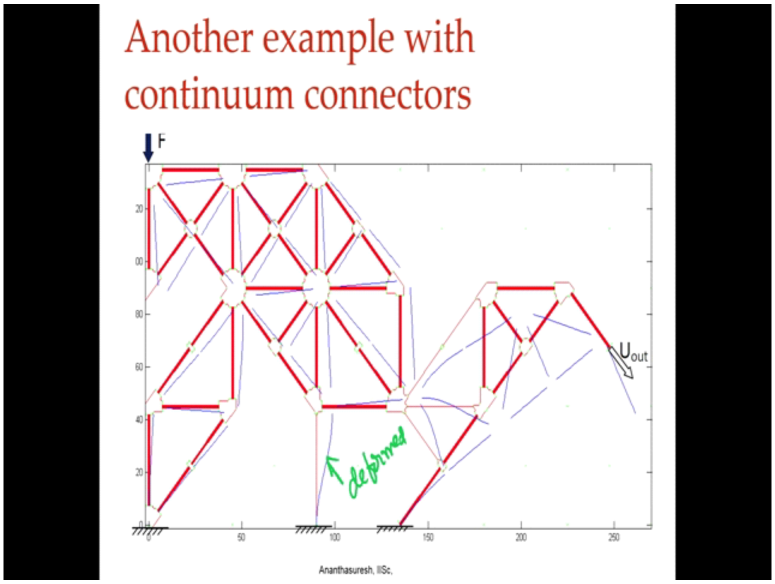
They all can be modeled as beam elements because they deform in between this rigid structure is modeled using continuum elements, what I mean by continuum elements is that it is made of let us say triangles there are lots of this little triangles here you have to model using continuum elements this thing we can call connector for the beam elements, If you do that the same mechanism I showed can now be modeled using this connectors.

(Refer Slide Time: 24:15)



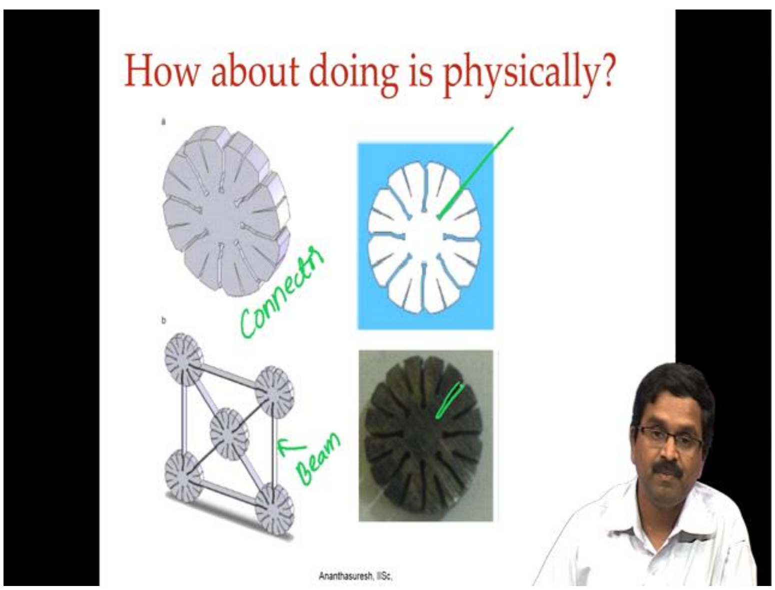
So, these are all the connectors we see lot of them wherever beam segments meet we can make this connectors now, this turned out to be more accurate and closer to the real device which is analyzed using continuum elements, another with beam elements is simpler and it is preferred, but continuum elements are always more accurate and here the beam elements are fine, but the way we model them is different. So, we decided that we need to have these connectors there.

(Refer Slide Time: 24:57)



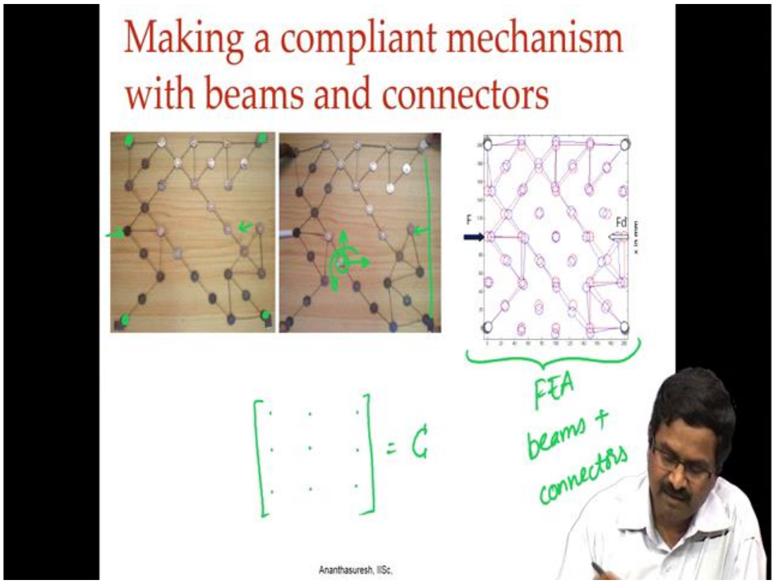
So, here is another mechanism that also has lot of this connectors were the blu e lines are the deformed ones this is deformed and the red ones is undeformed and again it compared well with finite element analysis of the continuum elements.

(Refer Slide Time: 25:17)



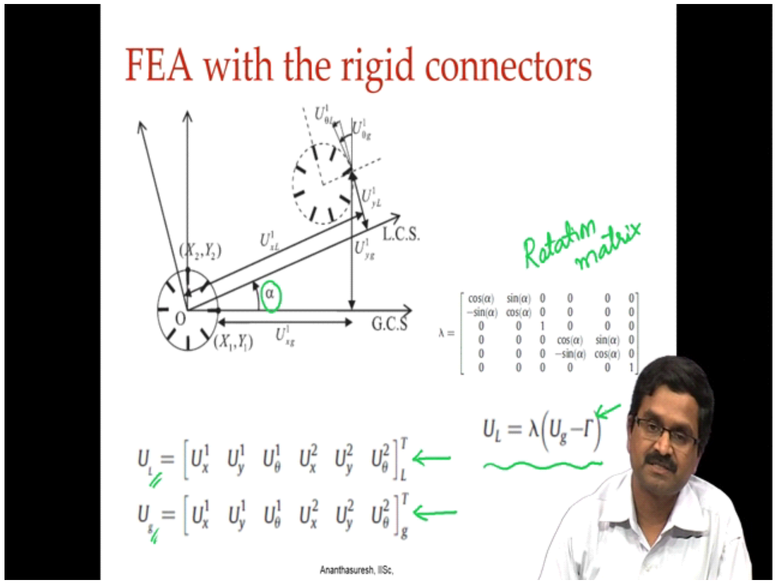
Now, having done this it give us an idea at the time that why do not we do it also physically just like I have this connectors. So, this is like a rigid connector and there are this segments these are little beams. So, just like we are molding a continuum compliant mechanism we thought we can take this connectors and make them something like a snap fit you know if I take a beam segment such as this one I just want to put it inside and snap it. So, it was done actually with a spring steel piece where we have cut out this using (Refer Time: 25:57). So, we put a connector so that it snaps in place we can build compliant mechanisms.

(Refer Slide Time: 26:04)



Such as this; so now, in this particular case these things are fixed let the mechanisms I showed at the beginning of this lecture, when apply an input force here this is suppose to come like this which is what you see we are pushing here it is actually from where it was it has moved there. So, you can put together this compliant mechanisms using these rigid connectors and flexible beam segments made of spring steel and you can actually develop the finite element analysis for this now it is not just the beam elements, but also you include the rigid connectors along with the beam elements to simulate. So, that whatever result you get with this F E A with beams plus connectors where connectors are modeled as continuum elements then the result that you get here will be close to the reality.

(Refer Slide Time: 27:11)

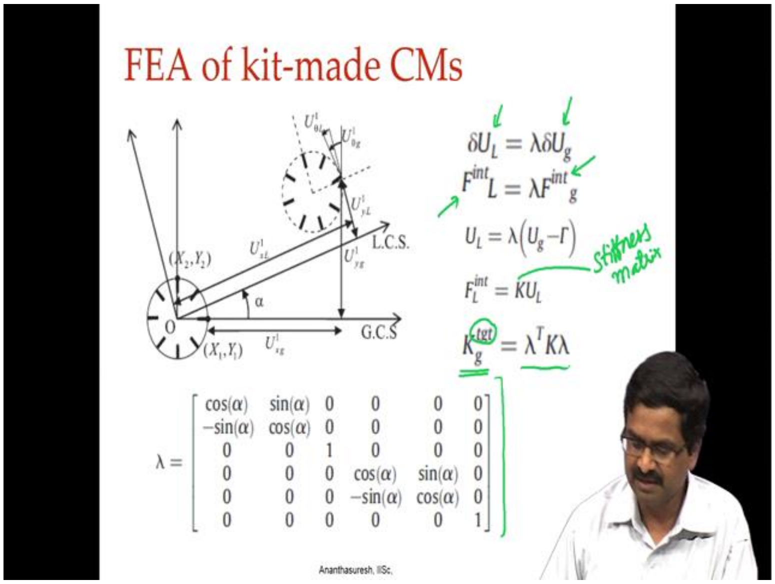


Now, that is a lot of work to have connector meshed with lot of continuum elements it increase the problem size. So, what was done was to use this Influence coefficients method which we had discussed in all the earlier lectures the idea of this is that in a real device if I take let say I want to model the stiffness of that I do not want to model it using continuum elements with a fine mesh. So, what I would do is I apply unit force in the x direction I get that displacements and rotation in the x direction, y direction and the rotation about z direction apply unit force in a y direction get this x displacement, y displacements and rotation and apply a unit moment there and get x displacement, y displacement and rotation.

Then I get this 3 by 3 compliance matrix which we had discussed and inverse of that give me a stiffness matrix that I can put this along with the beam elements and for that you need to do some rigid body rotations because a connector that is here has rotated by an amount alpha has moved by a displacement in x direction, y direction here, two things are shown one is global with a g subscript and local with L subscript. So, we can measure in the local coordinate system or global coordinate system we will come to that little later, but when you have this rigid connectors they can translate in two directions they can also rotate about the z axis, which have to be taken in to account as extra degrees of freedom at the nodes and then beam elements will have their own thing starting from this point. So, we have beam elements going form there and we will have rigid connector as its own degrees of freedom in the case of a plane three degrees of freedom.

So, that can be done there is a local coordinates and global coordinates for this displacements x y and theta for the left node x y and theta for the right node 1 and 2, first and second nodes local and global these two are related in the manner that is shown here where lambda here is the Rotation matrix, that depends on the alpha which is this rotation of that rigid body rotation of this it not only translate it also rotates above the z axis with that you can consider rotation matrix and that is how these two are related this gamma is also something depends on alpha which is not shown here I give a reference were you can find out how this gamma looks like.

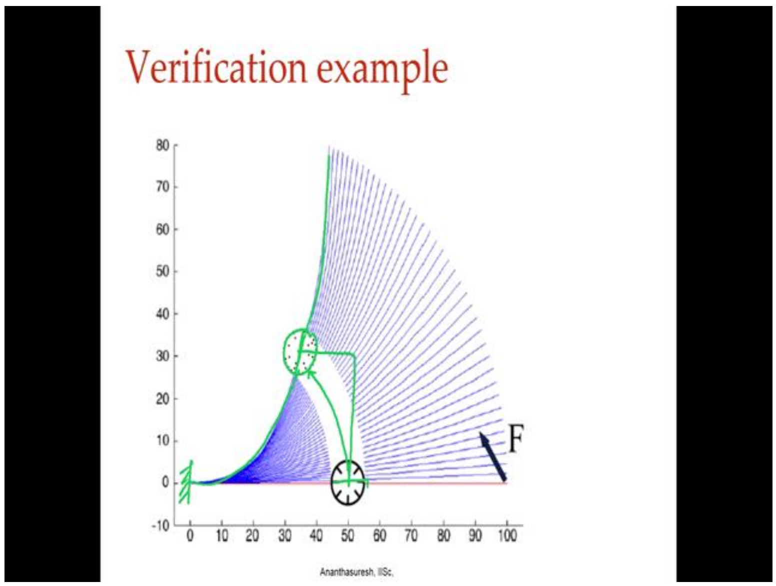
(Refer Slide Time: 30:02)



So, lambda little close up to use you can read the things better this rotation matrix is there is a incremental displacement delta U L and delta U g and delta U L are related by this rotation matrix and so are the internal forces will be two forces in x and y directions as well as moment that will be there in this vector. So, global coordinate system forces local global system forces are related again in the same rotation matrix and this one we talked about U L and U g how they are related.

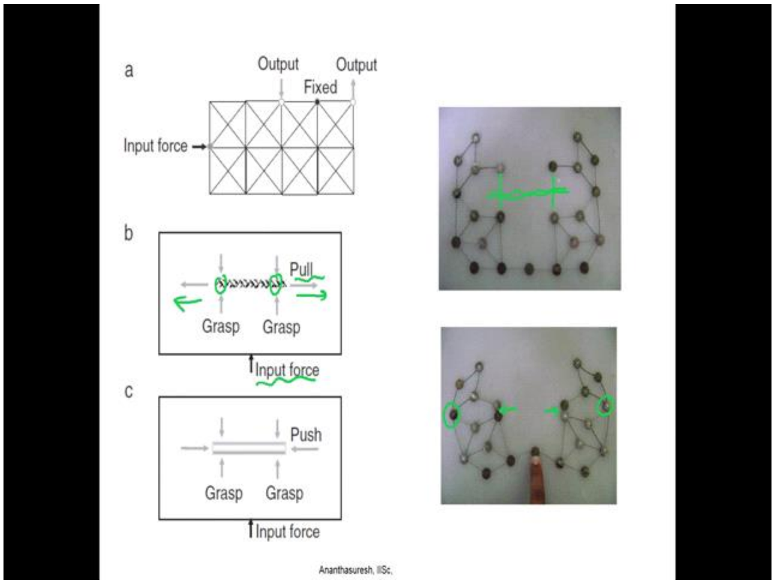
So, F internal if you what to do local coordinate system with a local stiffness matrix that if you know finite element analysis you would know what stiffness matrix means. So, for that you have stiffness matrix and we can derive that global stiffness matrix tangent stiffness matrix t g t here refers to tangent when you do nonlinear, again will mention this in second lecture from this one you can get this rotation matrix these are all the details of finite elements analysis if we do that we can verify.

(Refer Slide Time: 31:12)



So, if I have a connector like this let us say this is fixed and apply a force in the red line is the undeformed apply a large force is going to deflect and this how the real device also would deflect, now if you have something like this a connector in the middle apply a force you can imagine something deforming like that in the rigid connector now has rotated translated. So, it go to here, but also there will be a rotation of this that is probably not apparent over here when I put numbers; so this one would have then it goes out there it would have gone to a different angle instead of being horizontal line like this it could be like this, that there is a rotation is well, that is why I talked about the transformation matrix rotation matrix.

(Refer Slide Time: 32:01)

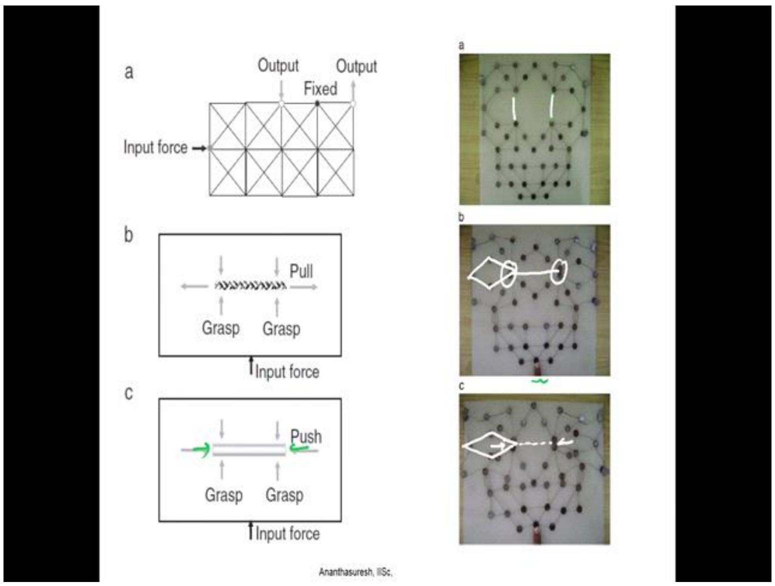


Now, let us come to the point of how we can actually if you are given a kit with beam segments and rigid connectors how would you solve this problems that is, I am saying that you have to fix somewhere let us look at this problem first fixing there when I apply input force I want these two points to move that one up this one down now, if you have all these things everywhere we can you can imagine the rigid connectors, you can assemble all these beam elements which one of them would you keep which one of them would you remove, that is the kind of design problem and that can be done something that you see here in this particular one; you apply a force they should go up and this should come down that is exactly what happens. We can see rigid connector there are snap connectors like Lego blocks we can get something like a very large deformation moving this will be instantaneous nonlinear finite element analysis will takes finite time whereas, here we can just move it and see how it does.

So, you can add and subtract these elements in the this kit of parts that you have rigid connectors and flexible beam elements we can see what is fixed here this has gone up, this has come down, let us look at the second problem this is interesting because we just want to apply a force over here just keep on pushing up, if I have a some string or something or a wire I want to first grasp it with like two fingers.

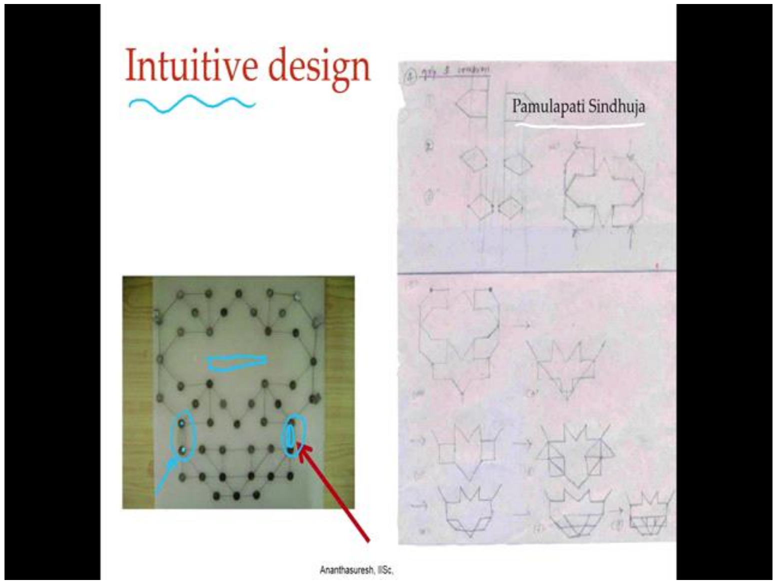
So, grasp over here and here and then pull this way, all I want to do is continue to apply input force up. So, for that the device is there that you can assemble and it was fixed at some of this points. I think it was fixed over here and here when apply force these two things that are separate will come together if you continue to press is going to go like this. So, if there were to be an object that is over here like that that can be grasped and then stretched.

(Refer Slide Time: 34:16)



Now, you have designed with your intuition now I change it now I want to grasp and then push for that there is a different mechanism, we can take this kit or parts and put them together and make it now we can see these the undeformed now we have pushed a little bit you see the little finger there and then these two things have come down that is the gap here and here has closed. So, let me use white color if you see this gap has closed over here and now, if you continue to press further compared to the distance there the distance here has reduced meaning that it is actually pushing we can actually make out something like that there has been become longer here pushing it in.

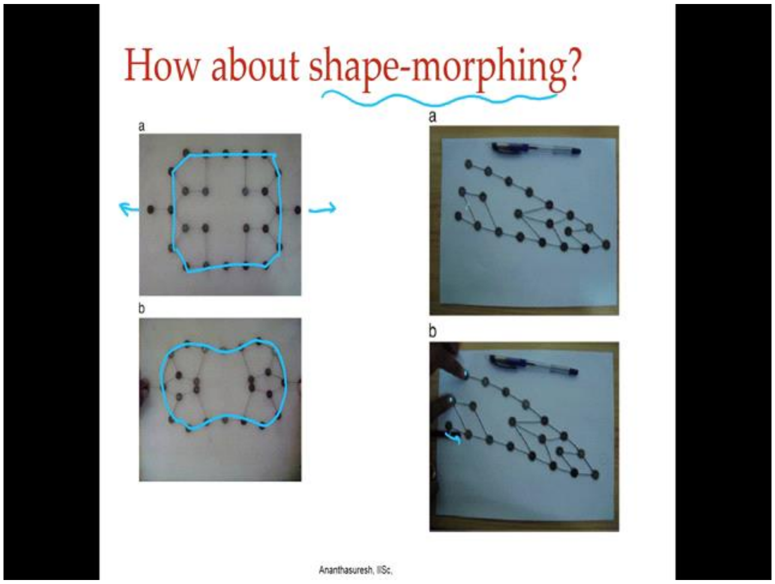
(Refer Slide Time: 35:06)



So, were able to design these things this was actually done by a under graduate intern in my laboratory some years ago and her name was Sindhuja and that is what we have put here and I kept asking how do you design. So, he actually gave me a piece of paper in from her note book how she designed this complex mechanism that can grasp something that is over here and then actually compress.

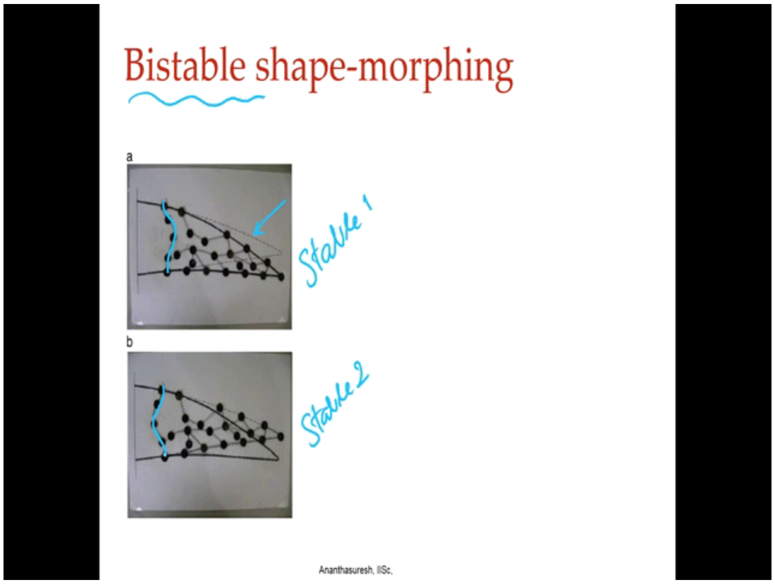
So, her thought process is here which is not easy for anybody else to figure it out, but if you were to do with this you have your own way of synthesizing using your intuition and if you notice over here and here what Sindhuja did was to have elements like this she wanted something rigid there, if you put one element there it like this over here it was not working out for her. So, she figured out that she can use little longer elements and bend them make it stiff between these two points that is exactly what a kit will do somebody makes a few parts and gives it to you and you figure out innovative ways of using it and that is what she and other people in my lab started doing.

(Refer Slide Time: 36:21)



Now, if you see this particular device when you apply when you pull this here and here it actually becomes something like a groundnut shape or sometimes we call it peanut shape. So, something like that looks like this shape has now become this shape you can do shape morphing with this kit we can make things that when you deform they assume a shape that you want; what shapes do you want you can play with aero foil shapes. So, if you have something like this some where you apply a force like in this particular case these two are held fixed as you can see with the fingers when apply force there it is going to deflect.

(Refer Slide Time: 37:08)

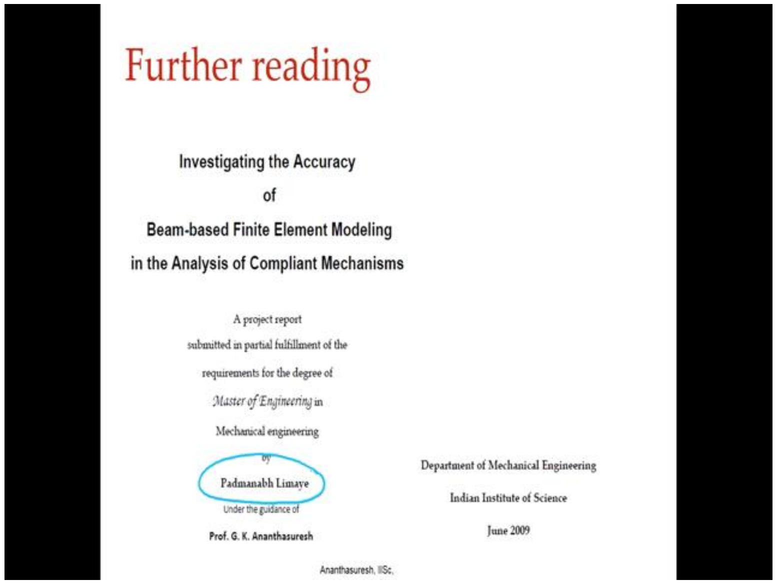


And in fact, Sindhuja went ahead, she was really playing with this kit she made it to be bistable as well, this is the trailing edge of aero foil. So, what was in the dash line here can be moved here it can be held in both configuration as you can see it is bistable. So, it is stable here, stable 1 state and then stable second state, first state second state show you able to use this kit in a particular way.

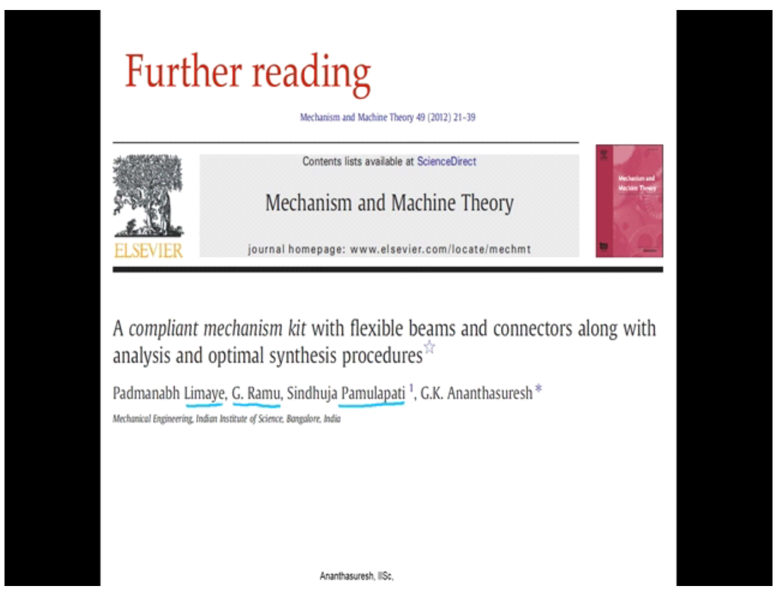
(Refer Slide Time: 37:45)

So, she could actually get bistable elements here and she went on to do multi-stable well as you can see this one has 16 states, if you look at any of them closely you can see how like for example, what is the difference between this and this is this element there that has become like this every one of them here can be turned like for example, if you see what difference between this and this. So, this is inwards this is out here. So, you can of course, this has gone back to this one. So, everyone of them is slightly different we can makes 16 stables states you know I should also say that there are 16 stable states basically once you have a kit we can do lots of wonders with it.

(Refer Slide Time: 38:27)

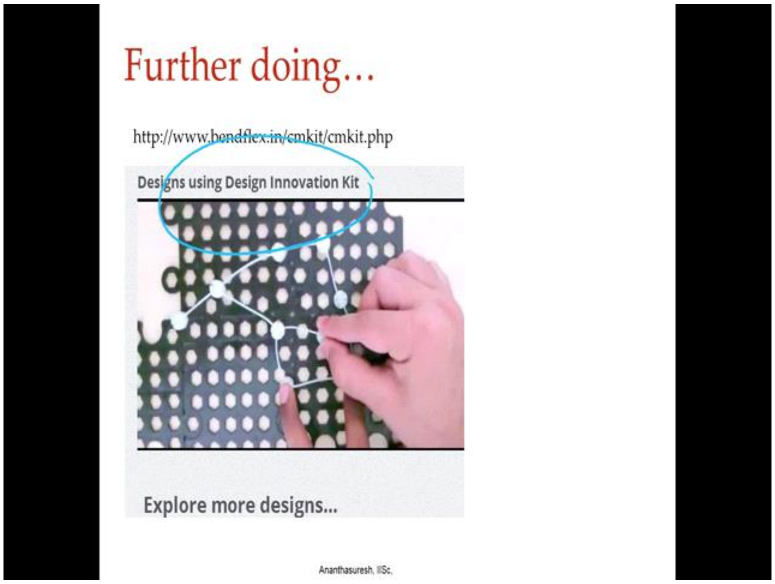


(Refer Slide Time: 38:36)



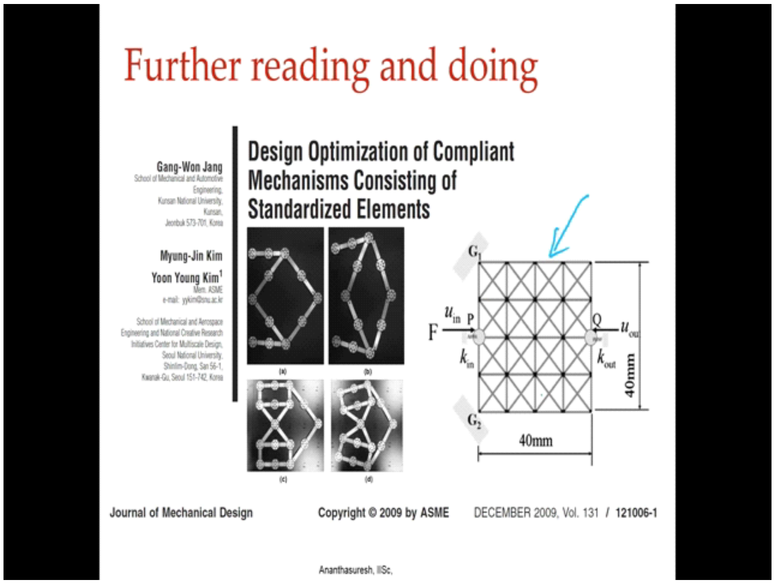
And more about this can be read in the Master thesis of Padmanabh Limaye and also there is a Journal paper where student Ramu had prepared all these elements and Sindhuja under graduate at that time really played with it and showed us how this kit can be used.

(Refer Slide Time: 38:53)



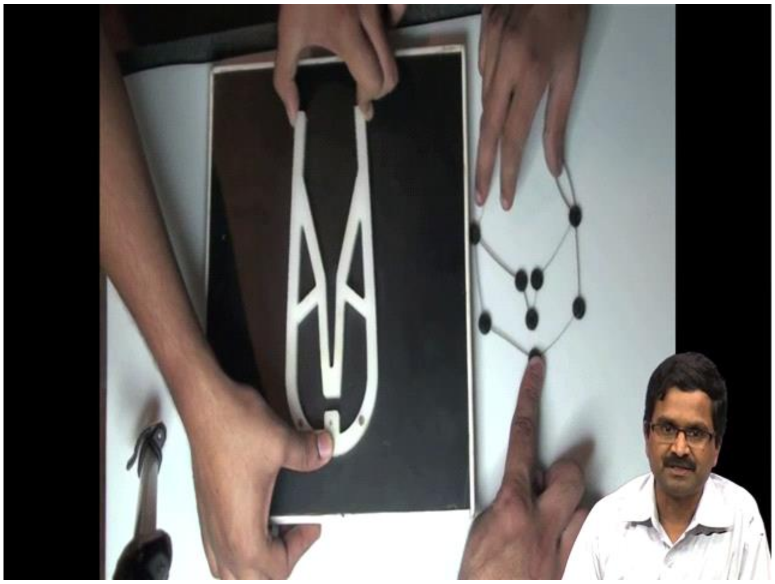
And little later on we also had one of our students actually make like a design innovation kit that I will just show you a little bit here, we can it comes up with you know this connectors and like breadboard like electronic breadboards we can have this it comes up with little beam segments as well as connectors that one can use to make a things like this it a little breadboard and in fact, we can connect multiple things here that go we can make a bigger breadboard as big as you want and make these things let us say I apply force something moves.

(Refer Slide Time: 39:53)



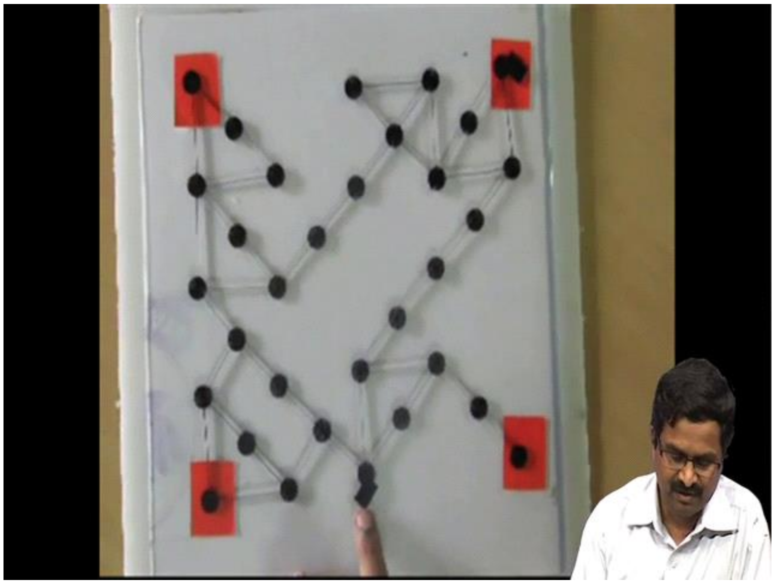
So, it has to actually fix here, but it can go like this. So, connectors are to be inserted and made these mechanisms. So, what I will show is that this has even more scope which is that there is a paper in the Journal of Mechanical Design where people came up with this standard elements as they called it to do design (Refer Time: 40:06). So, design meaning that you have all these things you do not have a real kit now you remove them mathematically speaking remove them or add them the way you want. In fact, Padmanabh Limaye also had done his masters work where you can actually synthesize compliant mechanisms which is what we call topology optimization.

(Refer Slide Time: 40:38)



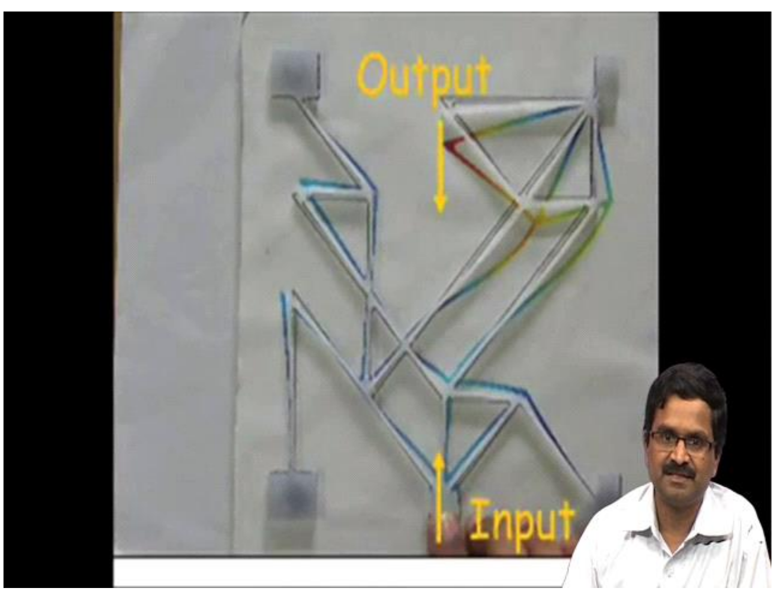
So, before I end I like to show you a little movie with which basically talks about everything that I have shown so far. So, it is basically showing a number of mechanisms and it shows how these connectors that you can take to make it out of steel it is much better, but of course, it will have sharp edges this beam elements and the rigid connectors which have those little snapping gaps with which we can put them firmly and make a mechanism as my student Ramu is showing in this video, where you can make these things and connect the way you want based on your intuition you do not had to do finite element analysis.

(Refer Slide Time: 41:26)



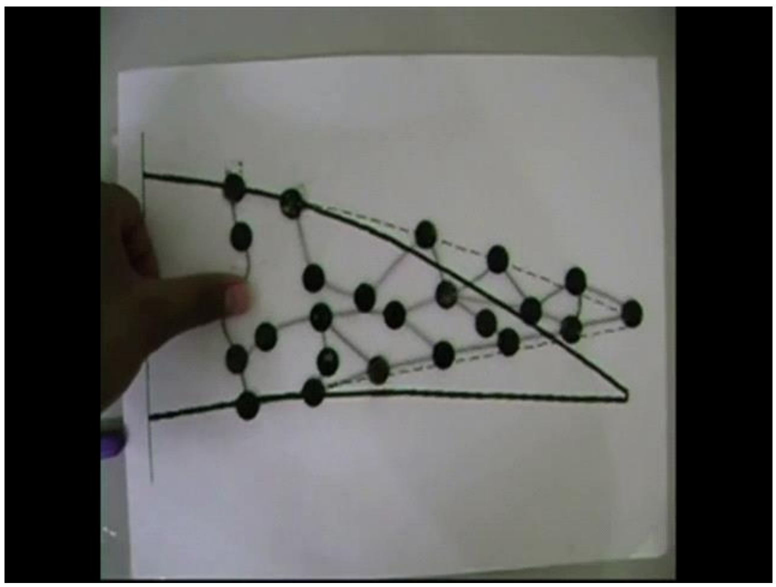
Now, we just move it right away its doing nonlinear realistic analysis whenever you want something rigid you make two elements like that and these two have correspondence the one that is there continuum compliant mechanism.

(Refer Slide Time: 41:39)



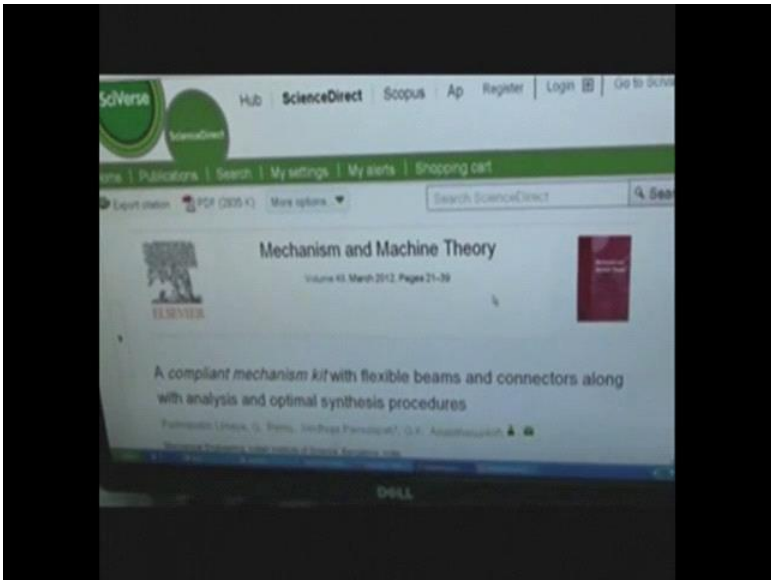
And the kit these example we showed when you have input and output here with the red ones where it is it was fixed and if you push it; it is coming down note how it deforming and then we will see the continuum version it deforms exactly the same way. So, you can conceive a compliant mechanism using this rigid connectors where you can have snapping beams going into it.

(Refer Slide Time: 41:51)



And see how it comes this the bistable one where the Leading edge of the aero foil structure and you can also do it for the trailing edge as we can see when we move that it is stable in both positions it goes it is like the flap changes the orientation and the paper has a lot of details and also the thesis.

(Refer Slide Time: 42:11)



So, with this we will end and then let us remember that there are some subtleties in finite element analysis when you model using beam elements and that is what we discussed today, and you have to be careful that there will be a rigid portion when you model using beam elements a real mechanism; real mechanism will have more rigid material compared to the beam element model.

Thank you.