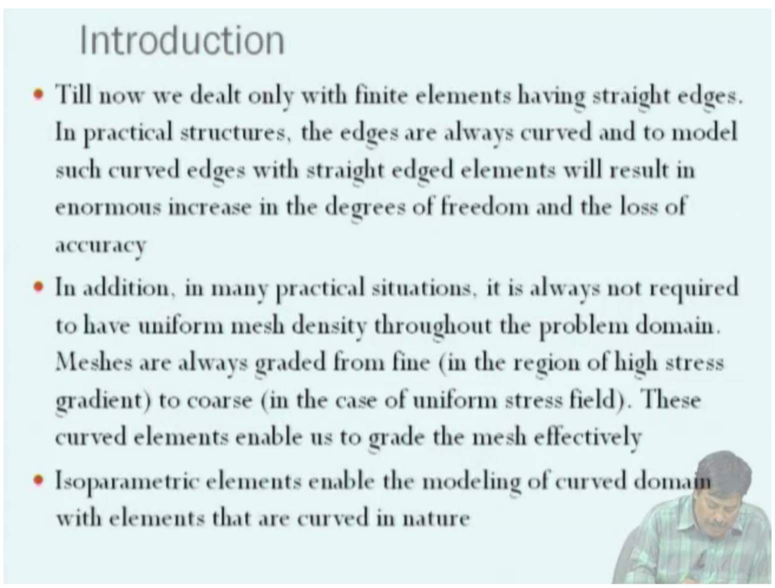
Micro and Smart Systems
Prof. S. Gopalakrishnan
Department of Aerospace Engineering
Indian Institute of Science - Bangalore

Lecture - 29
Isoparametric FE Formulation and some Examples

So this is lecture number 29 of the Smart and Microsystem course, so here we talking about the second part of the finite element formulation which is an extension of what we did in the lecture number 28. Here, basically we are going to deal with those kind of elements which are curved in shape, which are necessary for various applications, so today we will see how we can formulate the elements with curved boundaries, and in addition we will also solve some numerical examples to see how this method works.

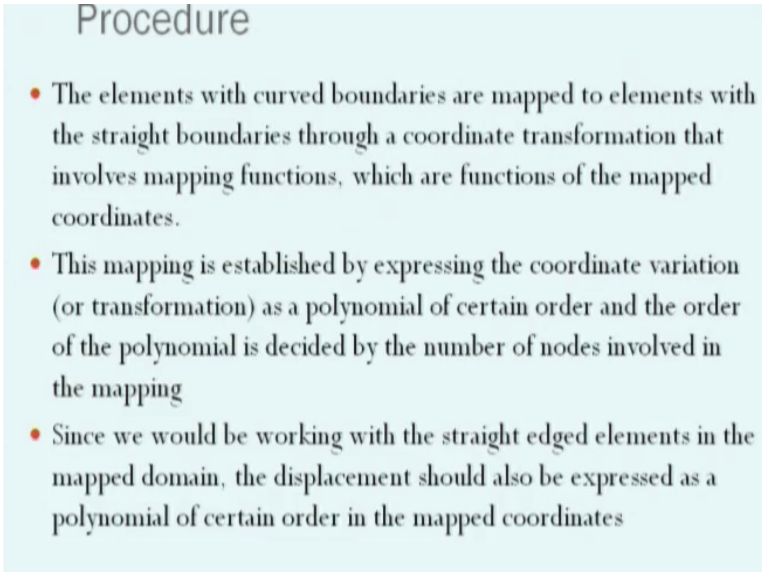
(Refer Slide Time: 00:59)



So as I said earlier till now we dealt only with straight edges elements. In practical structures, the edges are always curved and to model such curved edges with straight edged elements will result in enormous amount of degrees of freedom and also loss of accuracy. In addition, there are many practical situations in which the mesh density you will not actually mesh it uniform that is actually not advisable because it is going to increasing the mesh sizes.

And also when the regions where the gradients for examples stress gradients or the magnetic flux or any such things are very high and those regions require very high mesh density, whereas the other regions where the gradients are small or uniform we can just use a smaller set of elements. So in order to do this gradient, curved elements are absolute necessity. So how do we actually do this, there is a method called isoparametric finite element formulation which help us to model this curved elements.

(Refer Slide Time: 02:15)

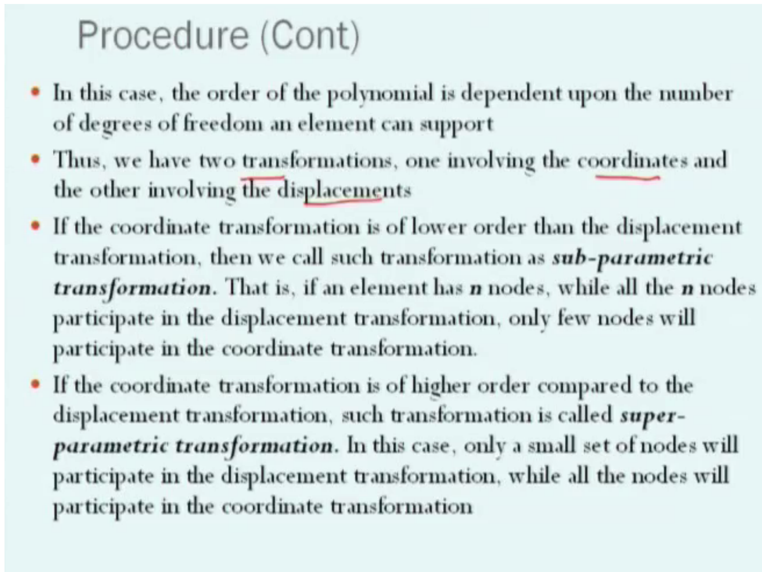


So how do we do this, the procedure that we adopt here is the elements with curved boundaries are mapped to the elements with straight boundaries through a co-ordinate transformation that also involves mapping functions, which are basically functions of the mapped coordinates. So we do have a 2 sets of coordinates one is the regular x, y, z coordinates and other is mapped coordinates.

So we try to map the curved boundaries to straight boundaries do our computation on the mapped coordinates and come back to the original co-ordinate system. The mapping is basically established through a co-ordinate variation or transformation as a polynomial of certain order, and the order of polynomial is decided by the number of nodes involved in the mapping.

Suppose, you have an 8 noded elements then the mapping is going to be higher order as opposed to 4 noded quadrilateral elements. Since, we would be working with this straight edged elements in the mapped domain, the displacement should also be expressed as a polynomial of certain order in the mapped coordinates. So everything the displacement, the strain displacement will all be doing on the mapped coordinates.

(Refer Slide Time: 03:30)

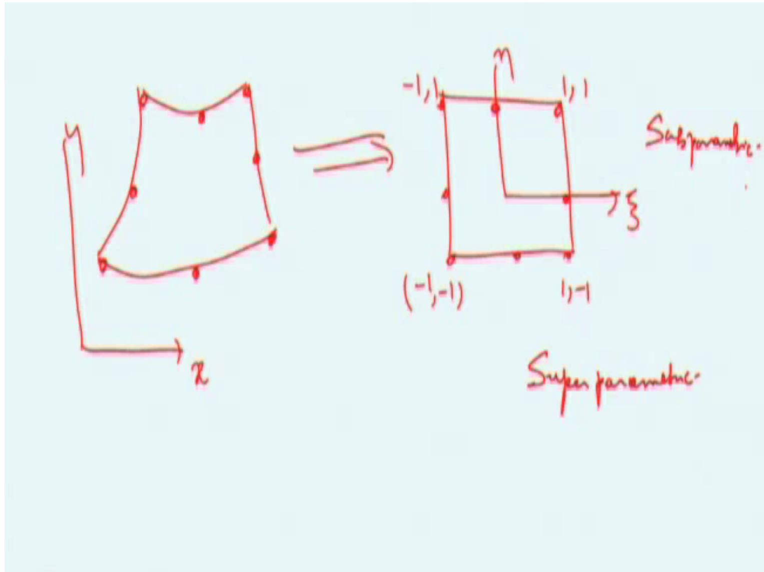


So in this case the order of polynomial is dependent upon a number degrees of freedom an element can support, we will come to that a little later now. Thus, in isoparametric formulation there are 2 sets of formulation, which is here one involving the co-ordinate, and the other involving the displacements. If the co-ordinate transformation is lower order than the displacement transformation, then we call such transformation as sub-parametric.

That is if the element has n nodes all the n nodes participate in displacement transformation, but only a few nodes participate in the co-ordinate transformation such a element is called sub-parametric element. On the other hand, if the co-ordinate transformation is of the higher order compared to the displacement transformation such a transformation is called super-parametric transformation.

In this case a small set up nodes will participate in the displacement transformation, while all the nodes participate in the co-ordinate transformation, let us explain this a little bit.

(Refer Slide Time: 04:56)

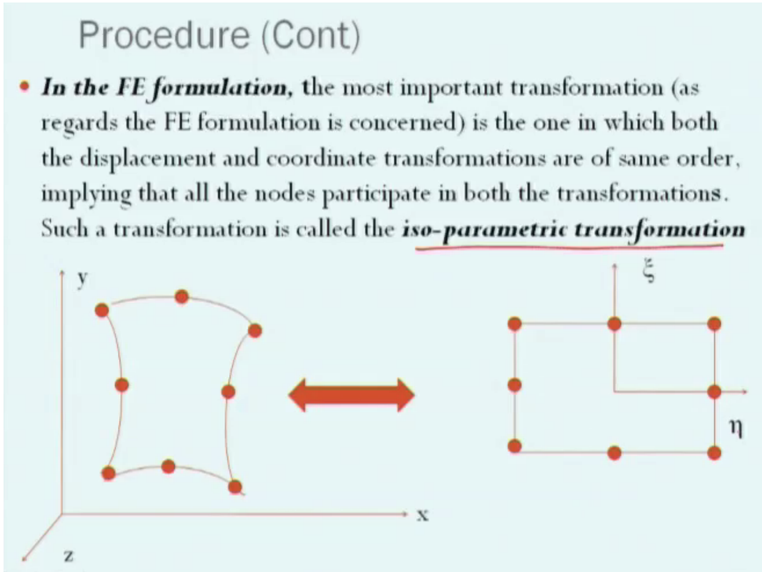


Suppose, we have an element we call this an element curved element having 8 nodes, which I write here, we map this onto a straight edged elements which is a perfect square, so this would be in basically xy co-ordinate system, whereas this would be in psi eta co-ordinate system. This is basically transformed to -1, -1, this is 1, -1, 1, 1 and this will be -1, 1.

And suppose, so we said that if we go back here we said that in the co-ordinate transformation is lower compared to the displacement transformation. That it is in the transform we again have 8 nodes it can be 8 nodes and if all the 8 nodes participated in the deformation, but only the corner 4 participate in the co-ordinate transformation such a transformation is called sub-parametric.

On the other hand, if there are only 4 nodes that are participating in deformation or the displacement transformation, whereas all the 8 nodes participate in the co-ordinate transformation such a transformation is called super-parametric. However, in the finite element formulation we neither use sub-parametric or super-parametric.

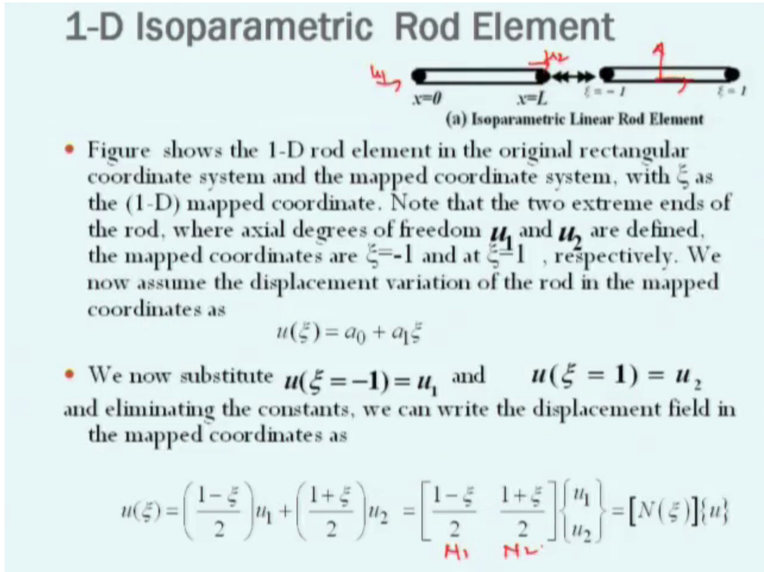
(Refer Slide Time: 06:55)



What we use here is, we use the same co-ordinate transformation for both displacement as well as co-ordinates that is all the nodes in the original domain will be the participating both in the displacement as well as in the co-ordinate transformation. So hence, the order of polynomial transformation for both displacement as well as in the co-ordinate will be same and hence it is called isoparametric formulation or transformation.

So this is precisely what we shown here, you have the original domain in x, y, z, and you have a mapped domain in psi and eta, and each one of them will participate both in there is one to one correspondence between the nodes in the xy domain to the nodes in the psi eta domain as the isoparametric domain or the transform domain. And the coordinates of xy will participate in psi and eta all the nodes and so is the displacement.

(Refer Slide Time: 08:06)



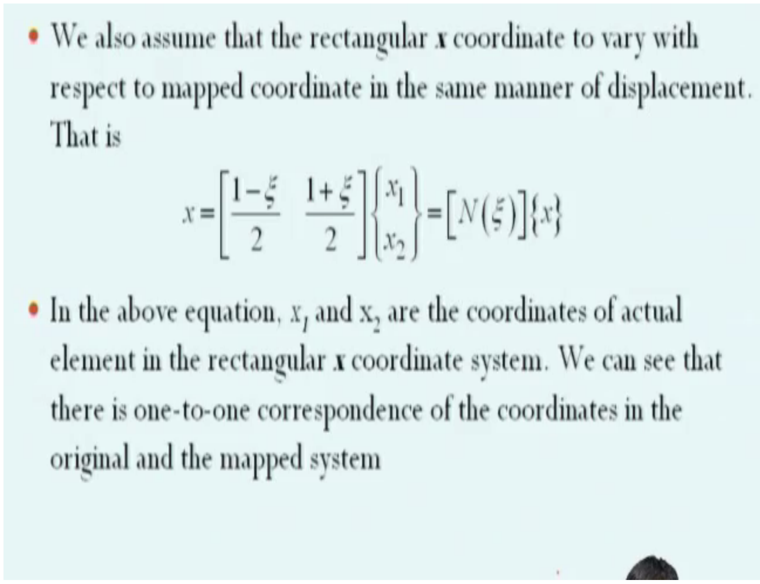
Let us now think about how we can actually formulate an isoparametric element, to demonstrate it let us take the simplest example that is the rod element, we had in the last lecture derived the stiffness matrix for the rod element. Now we will do the same using the isoparametric transformation. So the figure here shows a rod element which can have only 2 degrees of freedom that is you have u1 and u2.

And the left figure is the domain in the x direction and the right will be in the psi direction which is isoparametric co-ordinate. Isoparametric co-ordinate will always have the origin exactly at the center, so this is of unit 2 psi=-1 correspond to left node, psi=+1 correspond to the right node. Now we will now start the formulation by again assuming a displacement variation, now in the mapped coordinates.

Because we are not going to work on the left system here which is xy system, we are going to work on the right system which is the psi system, so we need to assume our displacement variation and on the psi system, so the psi system u of psi will be a0+a1 psi, because we have 2 nodes so corresponding to that we need to have 2 constants in the assumed variation and a0 and a1 are the constant.

So as we did for the regular finite element, now we are going to substitute at u at psi=-1 it is u1 and u at psi=1 is u2. Now relate the coefficients find the coefficients exactly the procedure which we followed to construct the shape functions in our previous lecture. So in doing so we can write u psi is Ni*ui where Ni which is a function of the psi, which is the mapped coordinates is 1-psi/2*u1 and 1+psi/2*u2, so when we put this so this becomes my shape function matrix, which is N1 this is N1 and this is N2.

(Refer Slide Time: 10:37)

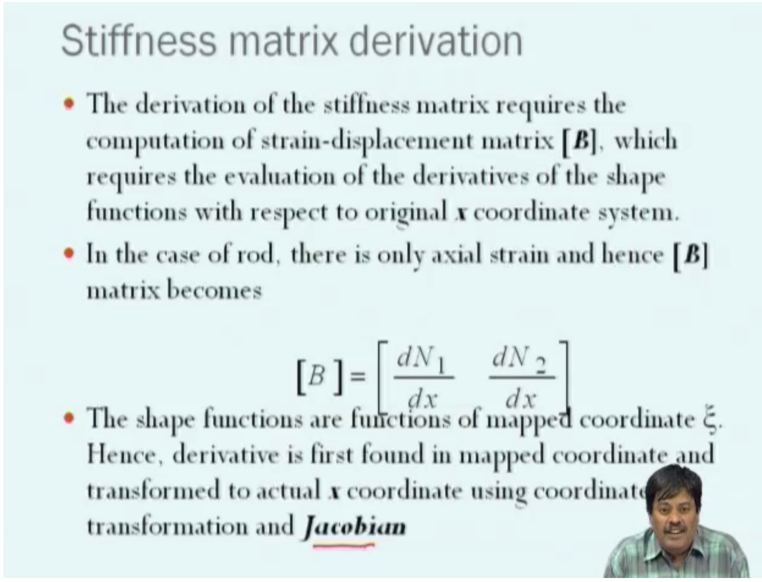


We also assume that the rectangular x co-ordinate, now we need to because we also transforming from xy to psi, so we need to have a transformation for x because it is a one-dimensional problem so only one co-ordinate will participate in the transformation. So we need to have the rectangular x coordinate also vary as a function of the mapped coordinates what we have defined for the displacement.

so we said u=Ni ui, x=Ni xi the same N will be used because the order of transformation is same because it is an isoparametric transformation, so x can also be written as 1-psi/2*x1+1+psi/2*x2. In the above equation x1, x2 are the coordinates of actual element in the rectangular co-ordinate system, we can see that there is a one to one correspondence of the co-ordinates in the original and also the mapped system.

And note that these shape functions also satisfied the properties of the shape function that is it takes a value 1 at the node where it is evaluated in the mapped coordinates and sum of the 2 shaped functions always equal to 1.

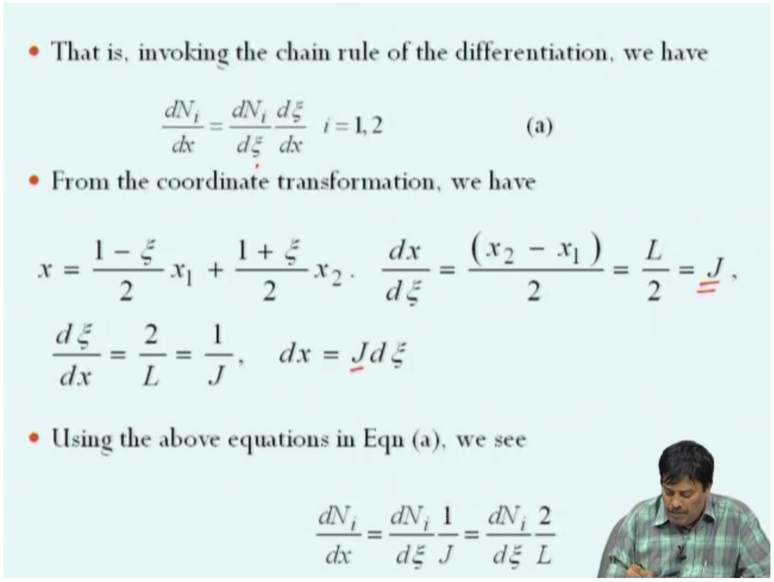
(Refer Slide Time: 12:02)



So now next see how we can derive the stiffness matrix, the derivation of the stiffness matrix obviously requires the computation of the strain displacement matrix B, which requires the evaluation of the derivatives of the shape functions which is with respect to x. Now we have a problem, because we always be have the shape function in isoparametric formulation in which is a function of only the mapped co-ordinate which is psi co-ordinate in this case.

So we need to convert this psi coordinates into derivative in to x co-ordinate derivative, so we need to get the derivative with respect to x co-ordinate. So in this case the case of a rod the B matrix will dN1/dx and dN2/dx. So in order to get the transform the psi co-ordinate derivative to x co-ordinate, we need what is called the Jacobian and that is where we need the co-ordinate transformation variation, let us see how we can get this.

(Refer Slide Time: 13:13)



So we invoke the chain rule of differentiation which is used in the partial differentiation chapter of the mathematics where we say dNi Ni can be N1 and N2 dNi/dx= dNi/d psi*d psi/dx i could 1 and 2. Now in order to find d psi/dx we need the isoparametric transformation of the co-ordinate that is why we need the co-ordinate transformation. We already said the co-ordinate transformation always uses the same transformation as that of the displacement.

So we have x=1-psi/2*x1+1+psi/2*x2, so we take a derivative dx/d psi which is basically x2-x1/2 and the x2-x1 is basically is the length of the element L/2 which we designated by J which is we called it as a Jacobian. So we get d psi/dx=2/L or 2/J and from this we can easily say dx= to converter our integration we also have the integration with respect to x so we convert x to psi by multiplying with the Jacobian J.

So now using the above equation in equation ‘a’ we can say dNi/dx= 1/J*dNi/d psi=2/L.

(Refer Slide Time: 15:01)

- Substituting the shape functions, which is given by

$$[N]=[N_1 \quad N_2]=\begin{bmatrix}\frac{1-\xi}{2} & \frac{1+\xi}{2}\end{bmatrix}$$

we can obtain the shape function derivatives with respect to mapped coordinates and hence the **[*B*]** matrix becomes

$$\frac{dN_1}{d\xi}=\frac{-1}{2}, \quad \frac{dN_2}{d\xi}=\frac{1}{2}, \quad [B]=\frac{1}{J}\begin{bmatrix}\frac{-1}{2} & \frac{1}{2}\end{bmatrix}$$

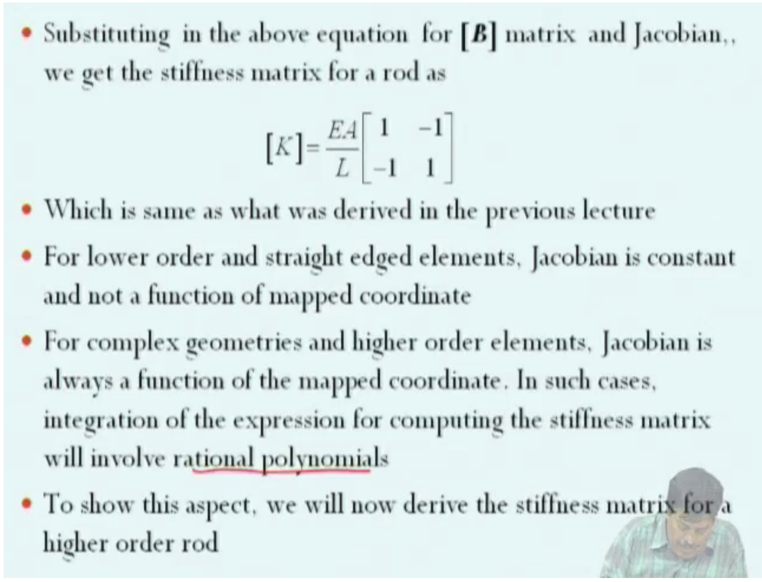
In the case of rod, there is only axial stress acting and as a result **[C],** the material matrix for evaluating the stiffness matrix will have only ***E*,** the Young's modulus of the material. The stiffness matrix for a rod is given by

$$[K]=\int_V [B]^T[C][B]\,dV=\int_{0}^{L}\int_{A}[B]^T E[B]\,dA\,dx=\int_{-1}^{1}[B]^T EA[B]\,J\,d\xi$$

So now we have the N matrix here, so we obtain dN1/d psi=-1/2 from here dN2/d psi=1/2, B=dNi/dx so which is 1/J of dN1/d psi and 1/J of dN2/d psi which is given by this. Now we have all the quantities what we want in order to find out the stiffness matrix except one. Now as in the case of beam we need to find out this matrix C, the only predominant stress is axial stress axial strain, so the axial strain is du/dx on which we derive the strain displacement matrix.

So the matrix is replaced by a single quantity called E, which is Young's modulus of the material, so the stiffness matrix which we know is this. So now this is on the volume on the mapped domain, and the mapped domain we have a coordinate of -1 to +1, so we say that we can actually change this to 0 to L in the B transpose E B into area of cross section of the rod same dA/dx and this because -1 to +1 B transpose EA*B*dx is nothing but J times d psi we derived this in last lecture.

(Refer Slide Time: 16:33)

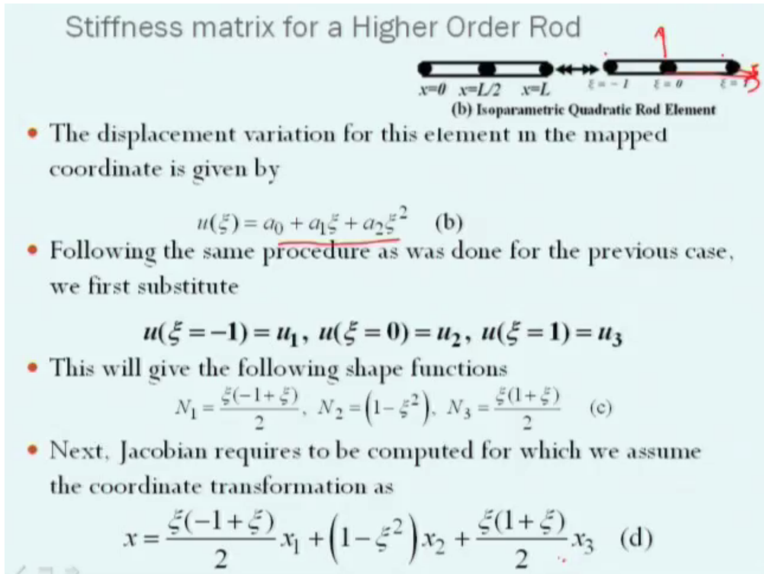


So we need to integrate this, so in doing so we get the K which is EA/L 1 -1 -1 1 which is same as what we derived in the previous lecture number 28. So here are some of the observations we could see, for lower order and straight edged elements Jacobian is always a constant value, which is normally equal to half the domain length and not a function of the mapped coordinates.

However, for complex geometries as we go along we will see or higher order elements Jacobian is always a function of the mapped coordinates, in such cases integration explicit integration like what we did here is not possible exact integration is not possible as it is done for a simple rod, then we need to because basically the integration will involve what is called rational polynomial, even today it is very difficult to exactly integrate the rational polynomial.

So this becomes very difficult in such situation what is the solution, before finding out the solution is let us see how we can get these kind of matrices by adding a little more complexity to our rod model by introducing an additional node at the middle, so it becomes a higher order rod.

(Refer Slide Time: 18:01)



So this is a higher order rod which is shown here on the left is the xy co-ordinate system and the right is the psi co-ordinate system, so the origin is exactly here, so we have this psi in this direction, so psi=0 at the middle and it again the unit is equal to 2 varying from -1 to +1. So here we have 3 nodes, the nodes are located at x=0, x=L/2 and x=L in xy system and psi=0, psi=-1 and psi=+1 in the psi system. As I said once we are mapping it we will work only on the psi system.

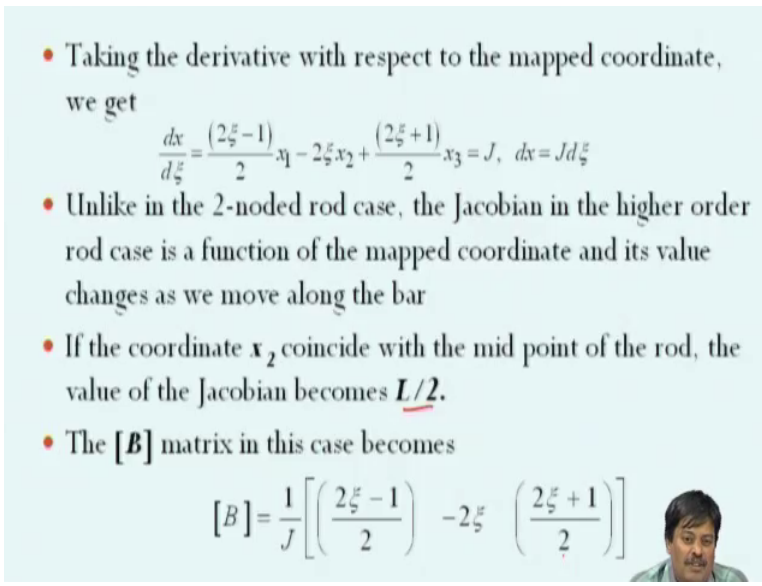
So we need to assume our displacement polynomial in the psi system which is given by equation 'b' and we note that we have 3 constant corresponding to 3 degree of freedom x=0, x=L/2 and x=L, so we have there is one to one correspondence between the psi system and x system, we need to have 3 constants here and the 3 displacement variation can be given as given in the equation 'b' here, and you see that it is not linear it is quadratic.

So we go by the procedure what we adopted for deriving the shape function, so we substitute u at psi=-1 is u1 that is here, u at psi=0 is u2 that is here and u=psi at psi=1 is u3 that is here. So we will substitute it and go through the procedures what we have adapted in the previous class, we can construct the shape functions and these functions are again quadratic, because we have quadratic variations in psi which is given by psi*-1+psi/2 will be N1, 1-psi square will be N2 and psi*1+psi/2 is N3.

And we note that this shape function again well takes a value of unity at their node and it will be 0 at other nodes, and sum of these 3 shape functions is equal to 1. So next we need to compute the Jacobian because that is required to convert the psi derivatives to x derivatives. So since it is an isoparametric transformation we assume the same variation as used in the displacement, so we can write x=Ni*xi where xi is the coordinates of the elements.

So that is psi*-1+psi/2, 1-psi square/2 and psi*psi+2 all multiplied by x1, x2, x3 has showed here, so equation d is the co-ordinate transformation.

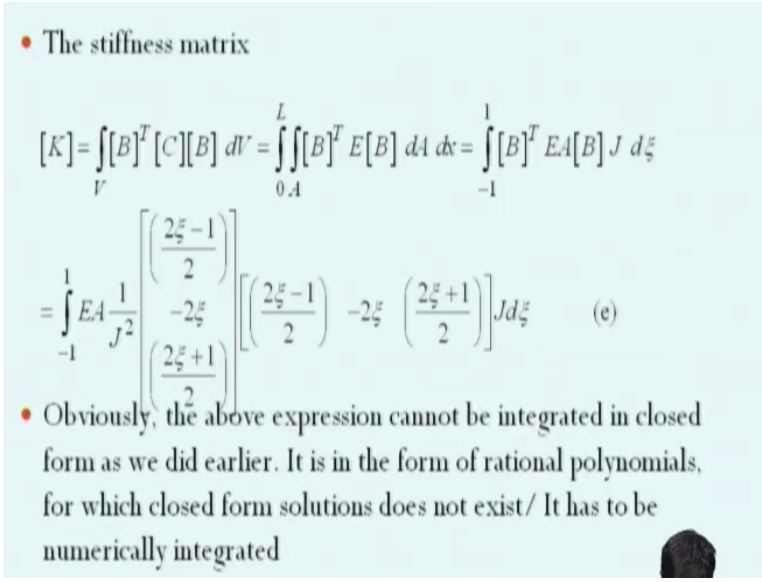
(Refer Slide Time: 21:00)



And next we will see need to take the derivative and we will do the same thing in the mapped coordinates as we did before, so dx/d psi will be equal to which is given by here by taking the derivative of the co-ordinate with respect to psi which is given by here. So we see that unlike the 2 noded rod case where the Jacobian was a constant the Jacobian in the higher order case is a function of the mapped coordinates and its value changes as we move along the bar.

And suppose we choose my coordinates x2 exactly coinciding with the midpoint then because of the symmetry we find the Jacobian becomes a constant equal to L/2, then we do the same thing what we did that is we convert this the derivative with respect to psi 2x by multiplying with J, so the B matrix will become now 1/J*2 psi-1/2-2 psi and 2 psi+1/2. So once we have the B matrix then we have all the quantities to evaluate the stiffness matrix.

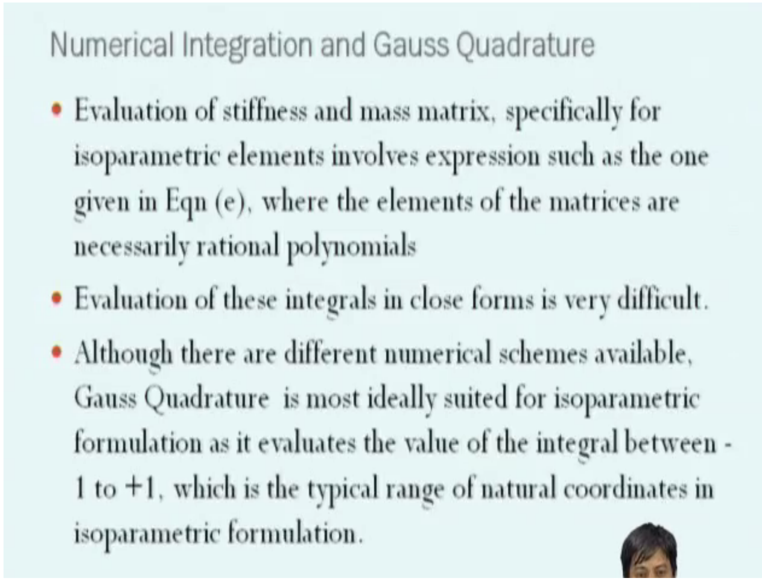
(Refer Slide Time: 22:35)



So the stiffness matrix is given by integral volume B transpose CB converted in to remove the area into this and converted into line integral 0 to l by multiplying this with the area, so you will get this and again dx=J times d psi, so everything is in the psi co-ordinate system and integration limit is -1 to +1, when we substitute this into this we see that we have a J square and J is also as see here is a function of psi.

And we have a coordinate complex functions here which becomes very difficult to integrate because the exact solution for this may not exist, so the integration has to be done through numerically integration and that is a standard procedure for all isoparametric elements. So let us now self this for a while, we will revisit later after we described the numerical integral procedure.

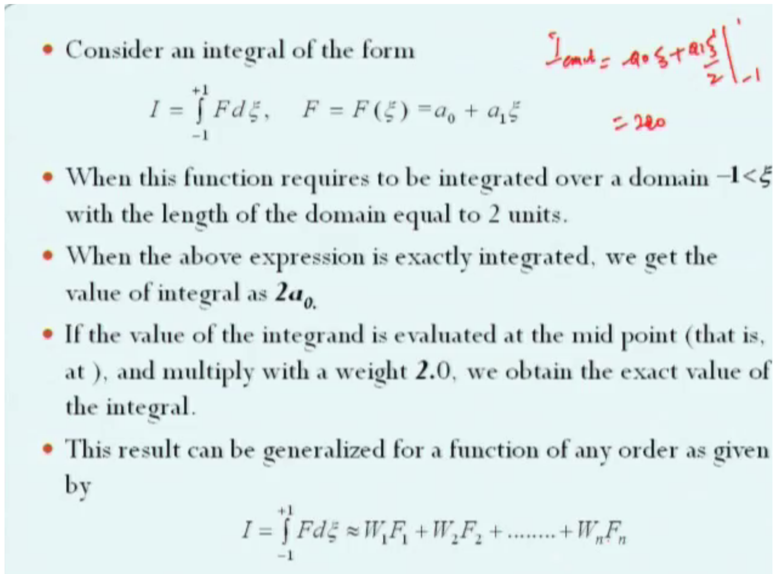
(Refer Slide Time: 23:49)



So we have seen that how complex the expressions become as we increase the complexity of elements add more and more elements and becomes make it more higher order, so the evaluation of the stiffness and also the mass matrix for dynamic problem specific for isoparametric element involved expression such as the one shown in equation 'e' here, so where the elements of the matrices are necessarily rational polynomials.

Evaluation of these integrals in close form as I said earlier is very difficult. Although there are different numerical schemes are available, the one which is more popular among the finite element specialized or the people working with finite element is the Gauss Quadrature method, it is ideally suited because for same reasons that it is a formula given where the integration limit is -1 to +1, which is the case with most of the finite element formulation, and it is also exact at certain points which has been theoretically proven.

(Refer Slide Time: 25:03)



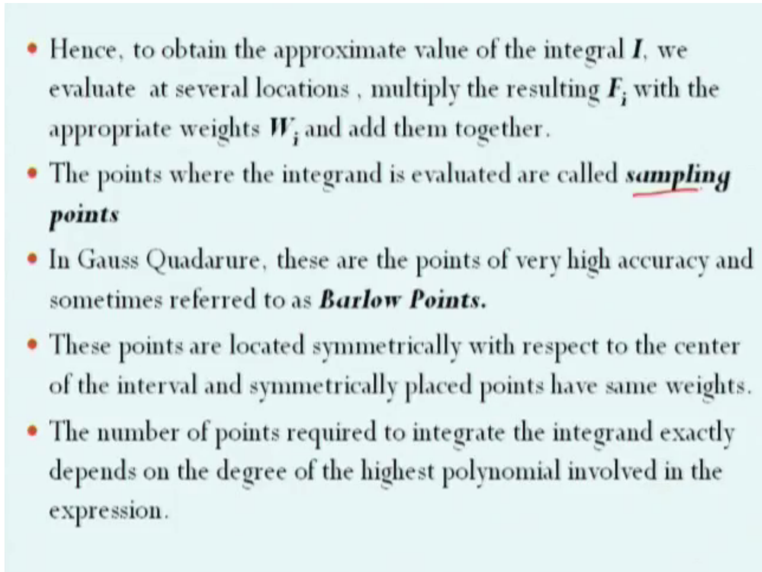
So what is this numerical integral let us consider a simple integral of the form F=a0+a1 psi, so if you want the exact integral I exact will be equal to you have a0 psi+a1 psi square/2 evaluated between -1 to +1 which will be equal to 2a0, this is exact we all know we can integrate it. Now if you want to integrate this function numerically within the limits -1 to +1 how do we do this, what does this Gauss Quadrature do?

So basically there are 2 ways to do it, suppose we design a method where we take we evaluate at certain point and multiply with certain weights can we get the exact answer. For example here if we say that we evaluate the expression at the midpoint, the midpoint is 0 that is at psi=0 and multiply the resulting with weight factor 2, then we get the exact answer.

And this is the philosophy, why is at the midpoint and where are the points where we need to evaluate these or those which are established after considerable mathematical rigor people have established it. So in generalized form of numerical integration especially with Gauss Quadrature is if you have F integral of this term, so you multiply with the function F1 evaluated at psi 1, this F2 is evaluated at psi 2 multiplied by W2.

And if you have n points you have n number of weights. So this is the principle on which we work with the numerical integration for finite elements and see how best we can actually achieve this.

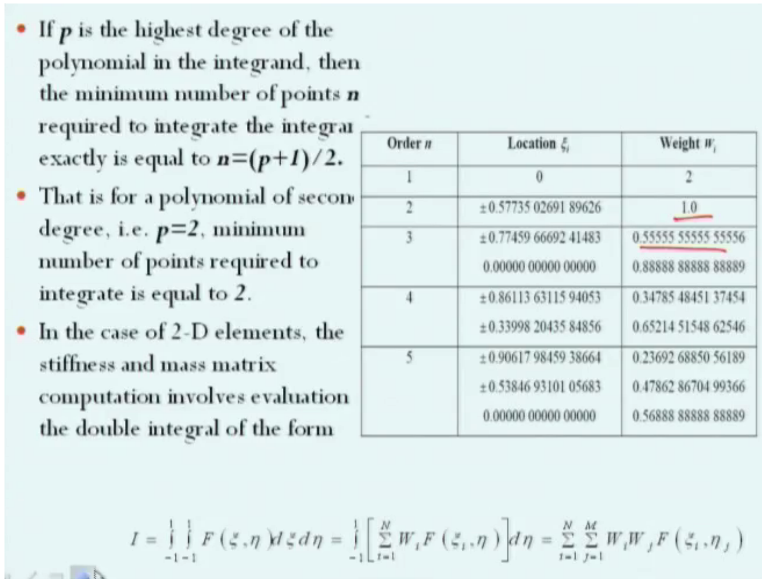
(Refer Slide Time: 27:18)



So hence, to obtain the approximate value of the integral I, we evaluate the integral at several locations multiply the resulting Fi's with the appropriate weight Wi and add them together. The points where the integrands is evaluated are called sampling points, and in Gauss quadrature these are the points of very high accuracy, people have actually proved that this have very high accuracy points and sometimes referred to as Barlow points.

So these points are located symmetrically with respect to the center are on the mapped co-ordinate center of the interval and symmetrically placed points have the same weights that is the beauty of the thing, and for integration up to certain order people know exactly what are the locations and weights. And the number of points required to integrate the integrand exactly depends on the degree of the highest polynomial involved in the expression.

(Refer Slide Time: 28:21)

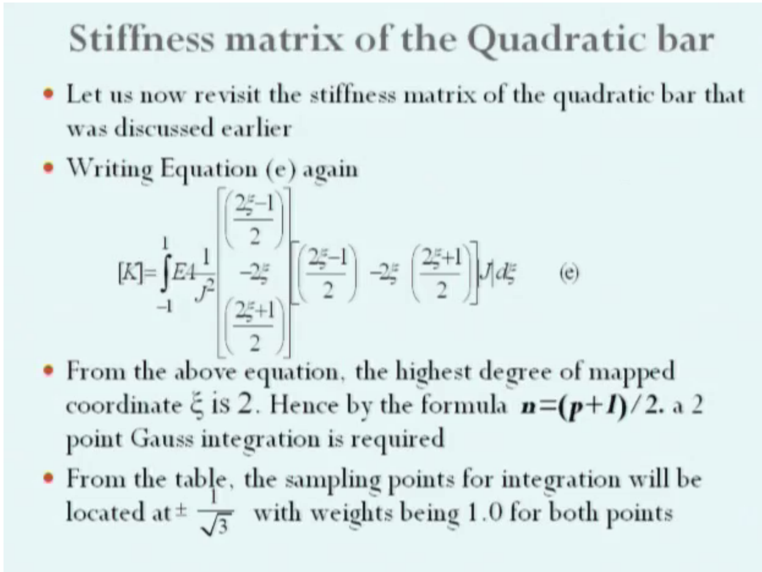


So let us see if p is the highest number degree in the polynomial in the integrand, then the minimum number of points n required to integrate is exactly equal to n=p+1/2, this has been mathematically proven, so we are not going to prove here, we will take it as a phase value that this is true. Suppose we have the highest order of polynomial in the integrand is 2 that is p=2, so minimum number we need 2 point integration.

So if it is a 1 point integration as I said the location is exactly at the middle with the weight 2, if you are want to have a 2 point integration location is 0.57735 which is basically +-1/root 3 and the weight is 1, so if we want the third order of integration this is 1 is at the middle and 2 others are the root of 0.6 and the weights are given here, so like that people have derived where the location where are this Barlow points, where these solutions are accurate.

And we just used it and the operations become very simple to actually integrate this, these are true if it is 1 dimensional structure, in the case of 2 dimensional elements the stiffness and mass matrix computation involved the evaluation of the double integral of this form, so we use 2 summation one on psi and one on eta and eventually this becomes in this form.

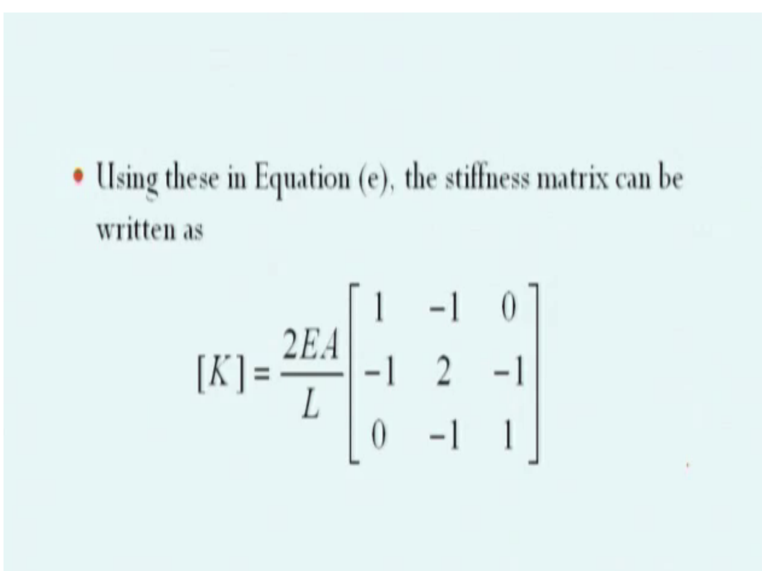
(Refer Slide Time: 30:08)



Now let us revisit our quadratic the stiffness matrix of the quadratic bar, we wrote this before and we said it is very complex and very difficult to integrate. Now let us examine this expression, we have J which is linear J square is quadratic and this product is quadratic, so basically we have and this is multiplied again with the determinant the Jacobian, so if you look at this we need the order of the highest integrand is 2, so we need n=p+1/2 in 2 point integration.

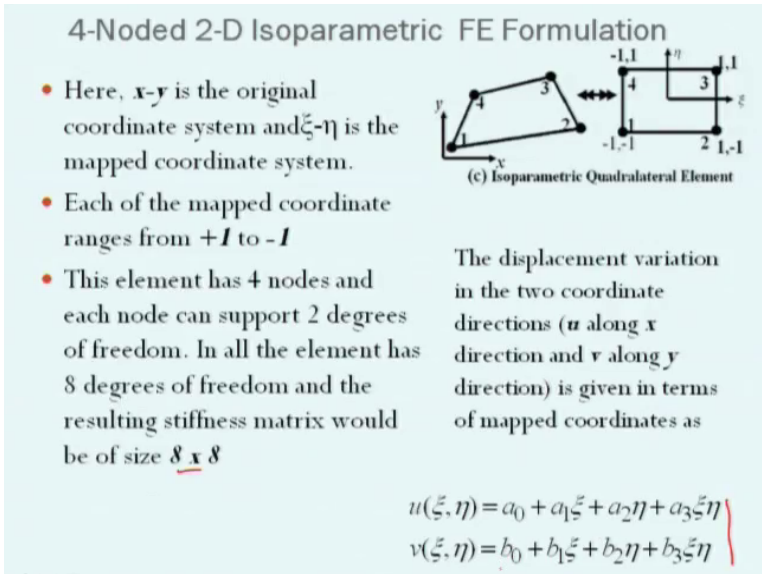
So the 2 point integration the points where we need to sample or the sampling points we need to integrate +-1/root 3 and the weight is given in the previous table which is given here, so the weights are given here the weight is equal to 1 here.

(Refer Slide Time: 31:15)



So when we do that and sum it up we get a very simple matrix which is symmetric and this is the stiffness matrix of the higher order, so we see that once when the order of the polynomial increases because of the increasing number of nodes then the integration become complex, the element formulation becomes really complex and the third numerical integration it is not possible to evaluate this stiffness matrix.

(Refer Slide Time: 31:43)

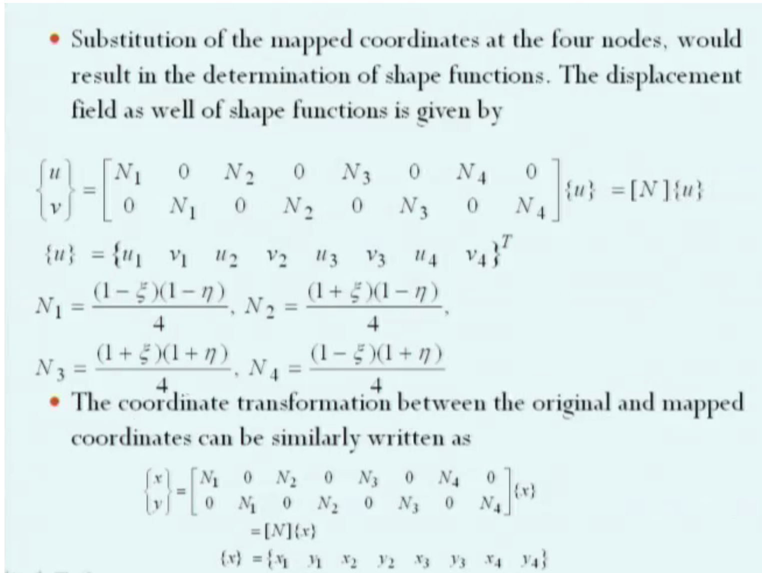


Now let us see how we can actually do a more complex by formulating the 2-D isoparametric finite element formulation, so you have a very arbitrary element which may be a quadrilateral in this quadrilateral we have straight edges of arbitrary shape, so to handle this in the conventional method becomes extremely difficult, this is the form of some kind of an arbitrary trapezium, so we mapped it to a completely a square of psi eta coordinates.

So now we have the isoparametric coordinates both in the psi and eta, so each of these can support 2 degrees of freedom, so there are 8 degrees of freedom per element, so they result in stiffness matrix should be 8 by 8. So now we will forget about the xy coordinates system since we have mapped it, we will work only on the mapped system.

So we start with the assumed displacement field on the mapped system in the psi eta coordinate, which is given by these equation here, so u psi and eta = aa0+a1 psi+a2 eta+a3 psi eta and b0+b1 psi+b2 eta+b3 psi eta. And you see that since there are 8 degrees of freedom 4 on u and 4 on v each of the u and v will have constants and psi and eta, and it is a linear in both psi and eta.

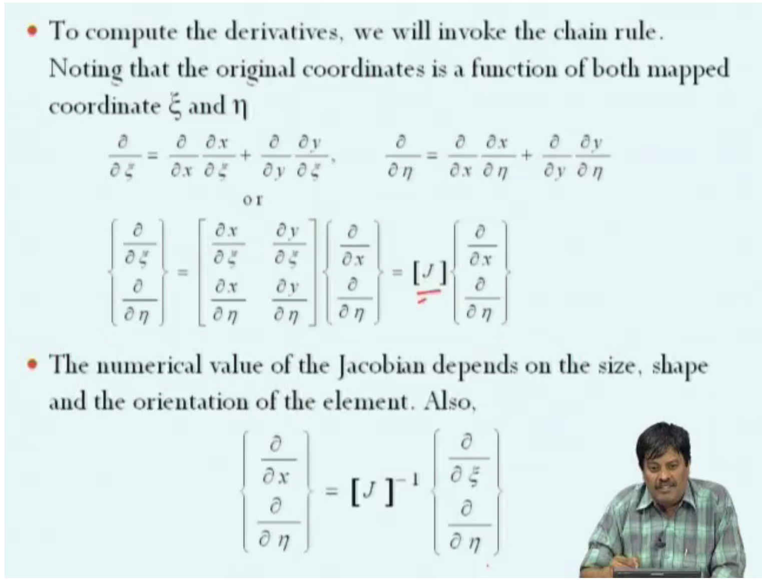
(Refer Slide Time: 33:14)



So this is the shape function matrix which can be arranged in this form where the u vector is u1 v1 u2 v2 etc. and if you go through the motion what we did for the rectangular element xy exactly following the same procedure using the assumed polynomial here. We can write the N1= 1-psi*1-eta4, N2= 1+psi*1-eta/4, N3= 1+psi*1+eta/4 and N1= 1-psi*1+eta/4.

Each of these again follows the properties of the shape function that is takes the values of 1 at the points where it is available 0 at all other points and sum of these 4 shape functions will be equal to 1. And now coming back to isoparametric formulation we assume the same variation of displacement to the coordinates, now there are 2 coordinates x and y. So here u and v are replaced by x and y, and u vector is replaced by x vector which contains the coordinates of the original element in the xy system.

(Refer Slide Time: 34:35)

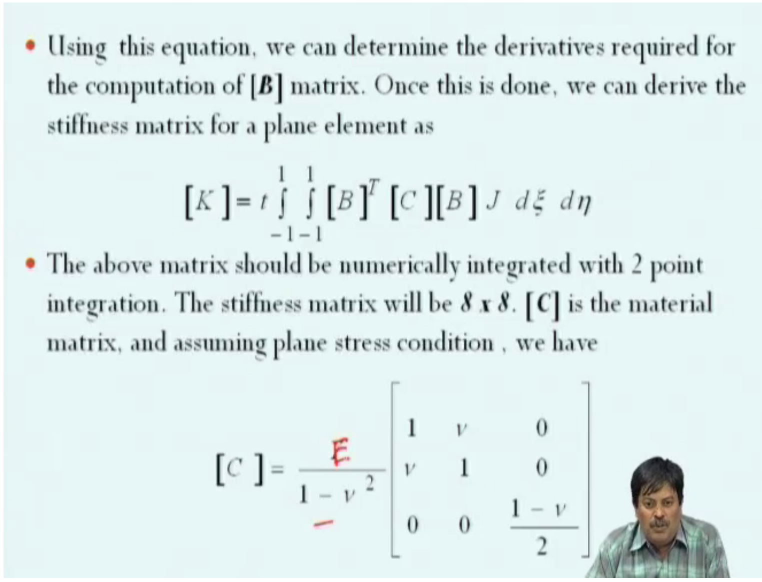


So next we need to convert the psi derivatives to x derivative, so we use the chain rule on both x and psi I mean the psi and eta, so d/d psi=d/dx*dx/d psi+d/dy*dy/d psi similarly, d/d eta=d/dx*dx/d eta+d/dy*dy/d eta, so this can be written in matrix form that {d/d psi and d/d eta = 1 matrix*d/dx, so that is this matrix is the J matrix is the Jacobian matrix, so this is the Jacobian matrix.

So now we are interested in only in d/dx and not d/d psi, so d/dx basically will be equal to J inverse of this into eta, one of the things that we have to be very careful is the J should always be positive, the determinants of J should exist it cannot go to 0, it can go to 0 when we are trying to mapped a regular domain to a completely 0 domain.

So one of the fundamental things in the isoparametric formulation is for the formulation for the mapping to exist Jacobian always be positive definite, that is the fundamental thing that is the J inverse should exist, so when I only when the J inverse exist we can get the psi derivatives and eta derivatives converted to x and y and derivatives, so this is y derivatives.

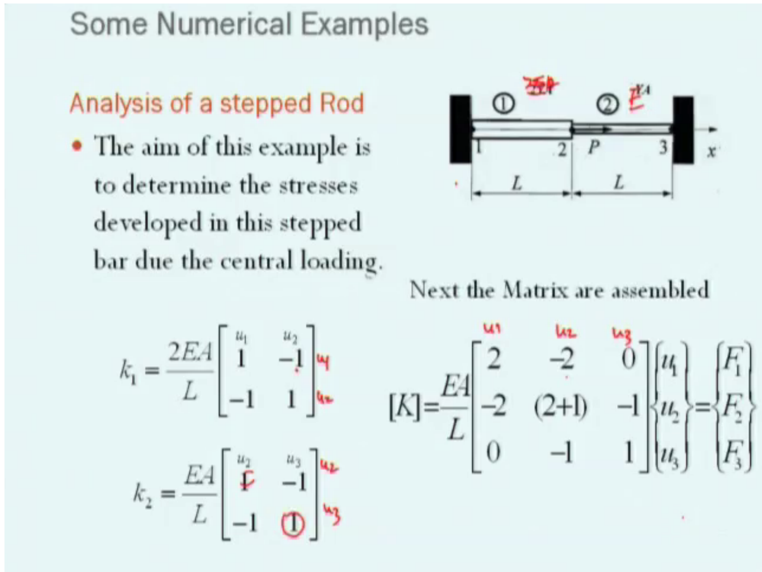
(Refer Slide Time: 36:07)



So once we have all these things, once we have evaluate the B matrix, so from which we can easily get the B matrix because this is just the strain displacement matrix, and here in the 2-D that the third dimension is only the thickness, so that are removed out and we have -1 to +1 in the x direction or psi direction and -1 to +1 in the eta direction, again dx and dy which is =J*d psi*d eta.

And of course we do not have a single element in the case of a 2-D element because there is a 2-D State of stress, so the constitutive matrix will be a full matrix, which we have already derived it which is given by this, where Y or it can be said as E is the Young's modulus and mu is the Poisson's ratio.

(Refer Slide Time: 36:59)



What we have seen till now is how we can formulate the regular element with a straight edges we did for the rod beam and also rectangular element, and we also said if the domain is curved or even straight how we can actually use the isoparametric formulation and formulate the element. And we have shown for rods and probably even beams we can show that what is formulated in the basically in the regular domain and the isoparametric domain, we will always have the same stiffness matrix.

Now we will actually use this developed stiffness matrix and solve some problems of practical interest, these are all very simple problems as we know that is the complexity increases we cannot do this problems by long hand, a simple problems are basically taken to demonstrate these methods. Let us take a simple analysis of stepped rod which is fixed at 2 ends which is shown here and there is a central point load.

This of course a problem which can be solved by regular analytical methods but here we will try to use the finite element methods to actually solve this, using the developed stiffness matrix, so you have a central point load here each is of the length L and you have the properties that this is 2 times YA, Y is the Young's modulus and this is YA. So let us actually say that so here we take it has 2EA this is 2EA and this is EA.

So now we take so totally we have 1, 2 and 3, 1 degree of freedom per nodes, so we have a 3 degrees of freedom element. What is the aim of this example? The aim of this example is to determine the stresses developed in the stepped bar due to a central point load P, so let us take this split up into 2 elements and we will write the stiffness matrix for each one of the elements.

The stiffness matrix for the element is EA/L 1 -1 -1 1 but EA is 2EA in the first element, so the stiffness matrix of the first element will be 2EA/L 1 -1 -1 1 and we will write that there is a local system and the global system, locally this is u1 u2, this is a global system so this will be u1 and u2. The second element will have only EA here, so EA/L u1 u2 u3 corresponding to the degrees of freedom u2 and u3, so we write it has u2 u3.

So now in order to make sure that the force equilibrium exist at the interface between the 2 elements we need to assemble this, so when we assemble this you will have u1 u2 u3, now while we will assembling we will go and see u1 u1, so there is a u1 u1 is here which has 2EA so EA so 2 goes here. Then you have u1 u2 so u1 u2 is 1, so basically this will be u1 u2 is -2 which is coming from here and 2 is multiplied which is there and u1 and u3 are not connected so it becomes 0.

Then similarly, it is symmetric so u2 u1 will be -2 and u2 u2, so the u2 u2 is a combination of the middle degree of freedom where there is a contribution from element 1 as well as element 2, so the element 1 contribution is u2 u2 is 2 which goes here, and element 2 is u2 u2 is this one 1 so 2+1 will get added up here, and u2 u3 is only here there is no u2 u3 here which is -1, and u3 u3 there is u3 here, so u3 u3 is only this component that goes here.

And the 2x2 system is upgraded to 3x3 system, next we need to see where the loads are, the load is the degree of freedom is u1 u2 u3 correspondingly we have F1 F2 F3, in the first node there is no load is there only the middle node F2=P and we need to see that these are a fixed end, so that means there is no moment of this in the horizontal direction, so u1 and u3 will go to 0, so you need to solve only for one unknown here.

(Refer Slide Time: 42:06)

- Load and boundary conditions (BC) are $u_1 = u_3 = 0$,
- FE equation becomes,

$$\frac{EA}{L}\begin{bmatrix} 2 & -2 & 0 \\ -2 & 3 & -1 \\ 0 & -1 & 1 \end{bmatrix}\begin{Bmatrix} 0 \\ u_2 \\ 0 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ P \\ F_3 \end{Bmatrix} \qquad F_2 = P$$

- Deleting the 1st row and column, and the 3rd row and column, we obtain,

$$\frac{EA}{L}[3]\{u_2\} = \{P\}, \quad u_2 = \frac{PL}{3EA}$$

- Stress in element 1 is $\sigma_1 = E\varepsilon_1 = E[B]\{u\}_{element1}$
- The $[B]$ matrix derived earlier is used here and

$$\{u\}_{element1} = \{u_1 \quad u_2\}^T = \{0 \quad u_2\}^T$$

is the nodal vector of element 1. Using these in the above equation, we get stress in element 1 as

So which is given here so we apply these boundary condition and so as I said, so here because there is a constraint we do not know what is the load here, so we eliminate this 2 rows F1 and F3 where we have applied the boundary conditions, then we have only one equation EA/L*3*u2=P, so u2=PL/3EA. If there is a uniform segment then it will be PL/EA now we will have a stepped rod so that will be PL/3EA.

The stress in the element sigma= Y or E this is E*epsilon 1 which is E*B*{u}, so B matrix was derived earlier is used here.

(Refer Slide Time: 43:01)

$$\sigma_1 = Y\frac{u_2 - u_1}{L} = \frac{Y}{L}\left(\frac{PL}{3YA} - 0\right) = \frac{P}{3A}$$

- Similarly, stress in element 2 is given by

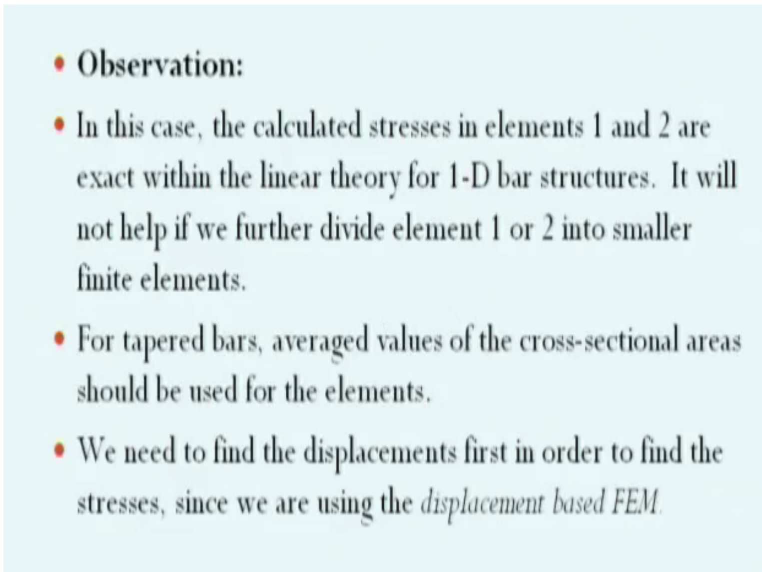
$$\sigma_2 = Y\varepsilon_2 = Y[B]\{u\}_{element2}, \quad \{u\}_{element2} = \{u_2 \quad u_3\}^T = \{u_2 \quad 0\}^T$$

Substituting the $[B]$ matrix and the displacement, the stress in element 2 is given by

$$\sigma_2 = Y\frac{u_3 - u_2}{L} = \frac{Y}{L}\left(0 - \frac{PL}{3YA}\right) = -\frac{P}{3A}$$

So when we use, this is what would be, so by doing this we can find the stress is given by P/3A. And similarly, we can find the stress into and which is = -P/3A, because the stress state is constant in a rod. So using a simple finite element method by using matrix approach we were able to find the stress in element without going resort in 2 equations, so it was purely used based on the stiffness matrix formulation.

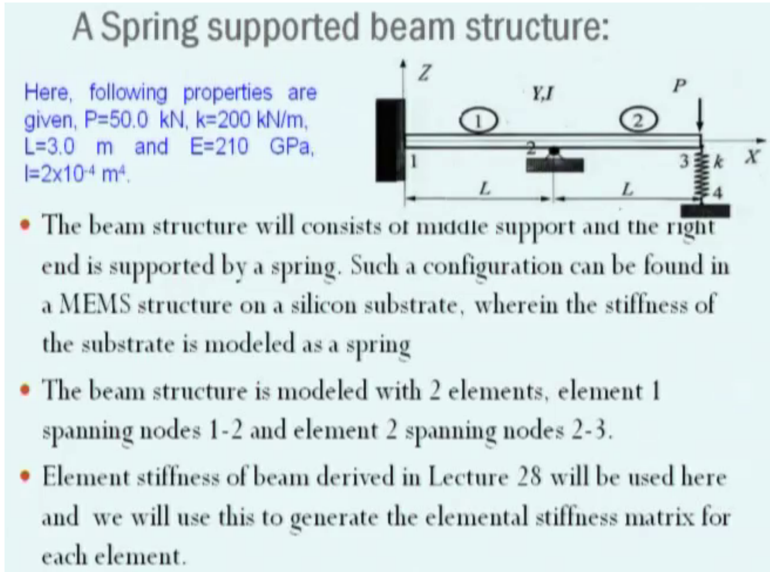
(Refer Slide Time: 43:29)



So what are the observations we see here, in this case the calculated stresses in element 1 and 2 are exact within the linear theory of bar structure why? Because we assumed polynomial u=a0+a1 x or a0+a1 psi exactly satisfies the governing bar equations, so even by increasing the element you will not help improving the results, whatever the results you get with one element is the converged results.

And if the bar is tapered if we take the average of the areas of the end bars you will get a very good results, we need to find the displacement first, before we find the stresses later, so stresses are always found by post processing the displacement.

(Refer Slide Time: 44:22)



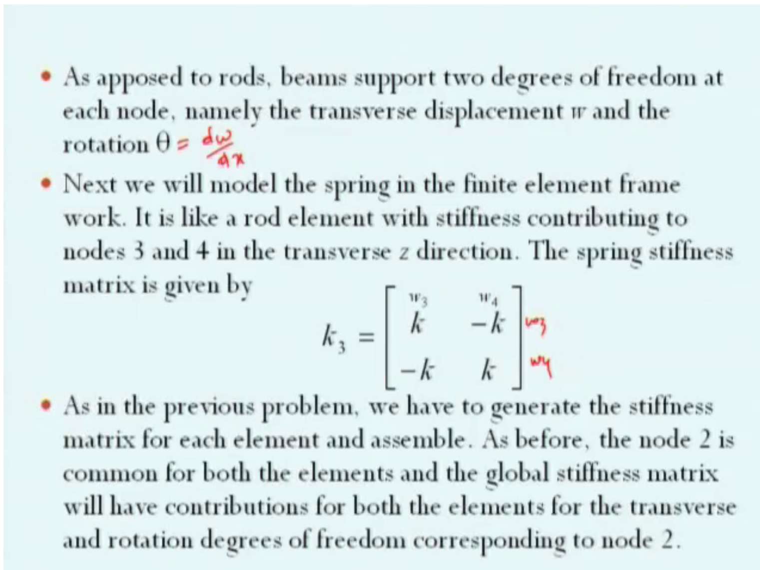
Let us take about the next example which is a Spring supported beam structure, we are given some properties here, the load is applied at the top and we the property the material property E is given, the cross-sectional property the area moment of inertia is given, where do you find these kind of structures, these are very common in the case of microsystems, the microsystems basically has the silicon wafer over which these beam structures rust.

So now in order to find the bending of the beams will create a capacitance which is a basically depends upon the type of sensor and measuring the capacitance will basically give the sensing components of what you are trying to measure. So typically the material the silicon substrate is basically the effect of silicon substrate on the beam structure are basically simulated through a spring, so basically we can actually evaluate the spring stiffness by looking at the stiffness of the silicon substrate.

So this is a typical problem which can be used in many of the MEMS structures such as accelerometers, so here there is a discontinuity here and we are going to model this again with 2 elements. In the structural analysis term this is an indeterminate beam, that is there are more number of equations than the number of statically equilibrium equations available, and hence this structure is statically indeterminate structure.

So we take these 2 elements model this structure with 2 elements, we have already derived the element stiffness, we take this element stiffness for a beam structure of course the beam structure is the element stiffness is 4/4, so it has more degrees of freedom that is based on the transverse and the rotation, then we assembled the stiffness put the boundary condition solve it and get all the post quantities what we want.

(Refer Slide Time: 46:44)



So as apposed to rods beams can support 2 degrees of freedom as I said at each node, namely the transverse displacement and theta which is basically is = dw/dx, as I said earlier the beam element based on this theta assumption is essentially a C1 continuous element. Next level we will model the spring first, what is the spring? What is the effect of the spring? So basically spring is like a rod element but the stiffness EA/L is replaced by k.

And basically it has 4 elements in addition to 2 beam elements we have to have a rod type element here which movement is only push and pull in this transverse directions, so we modeled this the element stiffness for a rod is EA/L 1 -1 -1 1, so instead of EA/L we put the k there and we can say the stiffness matrix for a spring element is k -k -k k and this corresponds to w3 w4 and the w3 w4 is shown here okay.

So, now as in the previous problem we have to generate the stiffness matrix for each of these as before we see that that node 2 is a common point for the element 1 and 2 that means the contribution is the transverse displacement to 2 and rotation at 2 will be there for both elements 1 and 2.

(Refer Slide Time: 48:34)

- Now adding all the stiff nesses, the global FE equation is given by

$$\frac{EI}{L^3}\begin{bmatrix} 12 & 6L & -12 & 6L & 0 & 0 & 0 \\ & 4L^2 & -6L & 2L^2 & 0 & 0 & 0 \\ & & 24 & 0 & -12 & 6L & 0 \\ & & & 8L^2 & -6L & 2L^2 & 0 \\ & & & & 12+k' & -6L & -k' \\ & \text{Symmetric} & & & & 4L^2 & 0 \\ & & & & & & k' \end{bmatrix}\begin{Bmatrix} w_1 \\ \theta_1 \\ w_2 \\ \theta_2 \\ w_3 \\ \theta_3 \\ w_4 \end{Bmatrix} = \begin{Bmatrix} V_1 \\ M_1 \\ V_2 \\ M_2 \\ V_3 \\ M_3 \\ V_4 \end{Bmatrix}$$

where $k' = \frac{L^3}{EI}k$. We now apply the boundary conditions $w_1 = \theta_1 = w_2 = w_4 = 0$ and $M_2 = M_3 =$ 0 Deleting the first three and seventh $V_3 = -P$ equations (rows and columns), we have the following reduced equation,

$$\frac{YI}{L^3}\begin{bmatrix} 8L^2 & -6L & 2L^2 \\ -6L & 12+k' & -6L \\ 2L^2 & -6L & 4L^2 \end{bmatrix}\begin{Bmatrix} \theta_2 \\ w_3 \\ \theta_3 \end{Bmatrix} = \begin{Bmatrix} o \\ -P \\ 0 \end{Bmatrix}$$

So let us see, now we generate the stiffness matrix, so the total number of degrees of freedom what we have if you look at it this becomes w1 theta 1 w2 theta 2 w3 theta 3 w4 theta 4, so it has 8 degrees of freedom why? Because there are 4 nodes here if you look at it, there are 4 nodes node1 2 3 and 4.

And so this is the assembled stiffness matrix and you can clearly see that for the theta component here there is the addition of the stiffness that is coming from the transverse displacement here is added here that is coming here and that can be clearly seen in the addition of the matrices. So now this is the assembled stiffness matrix where we add the component and we say k prime= (k L cube/ EI.

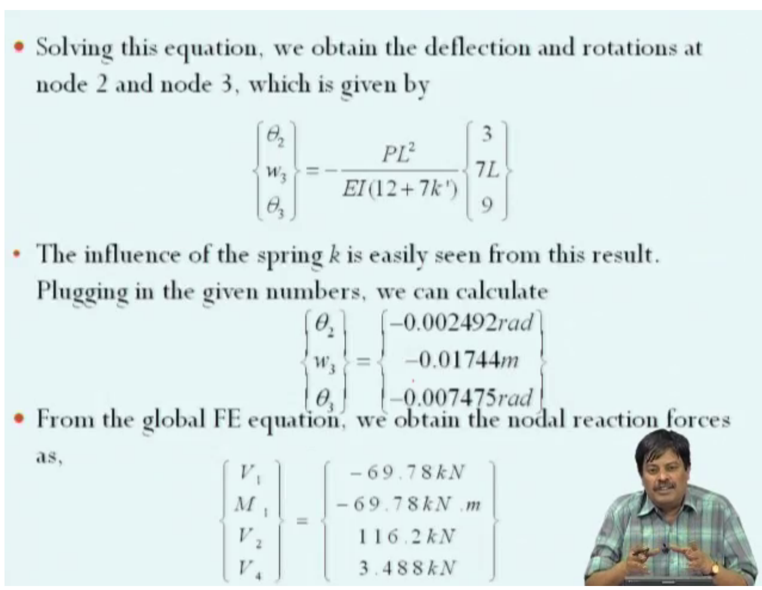
And now we assume we put that this is a so here the boundary conditions are it is a fixed end, so we have the transverse displacement w=0 here and rotation is 0 here. Here, this is a hinge there will be rotation but the transverse displacement is 0 here, here nothing is 0, in addition here the rotation will be non-zero but the moment is 0, it cannot take any moment because it is a hinge.

So we apply these boundary condition here we say that w1 theta 1 w2 and w4 the w4 here is the fixed, so here there is nothing happening here both the w4 is 0 because this has only theta 4 there is only w 4 here that is 0, and we also apply M2 and M3 =0. So here because we are putting a spring here it cannot undergo any moment, so we say that the moment here is 0, so these are the boundary conditions we apply here.

And after applying the boundary conditions we delete the corresponding rows and corresponding columns and get the reduced stiffness matrix and the reduced stiffness matrix is given by here, and you can clearly see the w3 there is a component of the transverse k and that is why you have a k here, and it is a simple 3x3.

And you can also note that the load at if you look at it the only active degrees of freedom are theta 2 w3 theta 3 that is the moment at 2 is 0 because I cannot take it, the moment at 3 is 0 I cannot take it, but the shear force at 3 is present there, so that is the concentrated load that is coming from here this is the node 3.

(Refer Slide Time: 52:13)

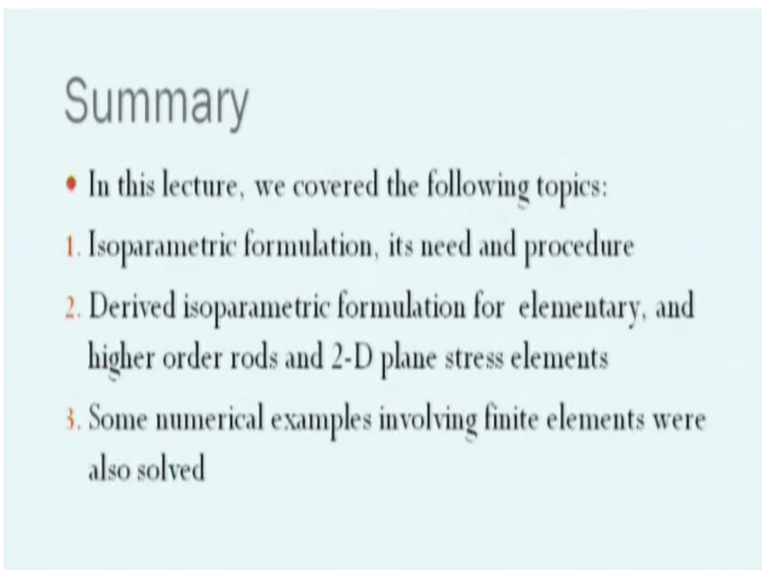


So once we have this, we can solve for the equations and once we solve this we can get the displacements at the rotation at node 2, the displacement at node 3 and the rotation at node 3 given by this expressions. And one thing which can clearly see that when k=0 it acts like a fixed free cantilever beam, because there is no restraint here, when k becomes infinite it becomes like a fixity there, so there is nothing to solve theta 2 w2 theta 3 becomes 0.

So you can see the k is an intermediate kind of a support which can simulate many many conditions depending upon what the value is, so after solving this we plugging those values which we gave at the starting of the problem and we see that these are the displacements, so once we get the displacement we go to the elemental equations.

And the elemental equations we substitute these equation, the elemental equation relates the shear force and the moment in each of the beam element to their respective displacements, we plugging there and we can get the shear force and moment at each one of these segments, so it is very elegantly a complicated problems such as this was solved using a simple finite element method using the developed stiffness matrix.

(Refer Slide Time: 53:48)



So now let us summarize, what we have studied in this lecture, in this lecture we covered the following topics, we said how the isoparametric formulation takes place the need for it, what are its procedures. How do we formulate the element for the simple rod the higher order rod and the 2-D 4 noded isoparametric element formulation we saw that. We derived the procedure the emphasized the need for the numerical integration.

And we also showed how the numerical integration works especially in the case of the higher order rod element, and showed that in conventional isoparametric formulation especially using for the complex geometries which we will be dealing with a many real life problems, the formulation leads to highly rational polynomials for which exact integration is not possible.

We showed that Gauss integration is very useful, we also found how to do that what are the integration points and how to choose the order of integrate integration and such other things. And finally we also said that how to solve some numerical problems using finite elements where we took a typically problems that are normally encountered in some of the microsystems analysis that one will be dealing with, thank you.