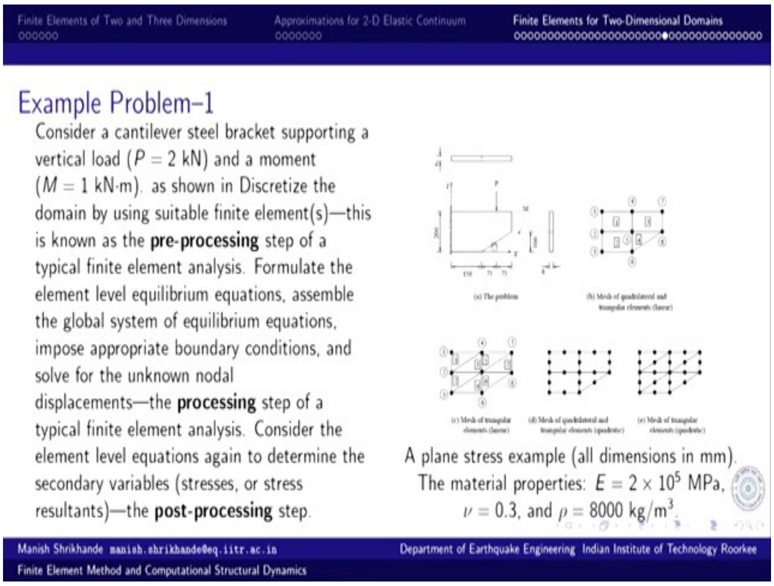
Finite Element Method and Computational Structural Dynamics
Prof. Manish Shrikhande
Department of Earthquake Engineering
Indian Institute of Technology, Roorkee

Lecture - 29
Finite Elements of C^0 Continuity in 2-D and 3-D-XI

Hello friends. So, we have seen how two-dimensional finite elements using in rectangular geometry are derived using Lagrangian interpolation, and also using serendipity formulation, serendipity family of elements. So, now, we try to solve one problem demonstrate the entire process of finite element analysis by using going through the entire process step-by-step, and solve one problems of stress analysis. And let us see how it progresses and that should help in crystallizing the concepts that we have covered so far in this course.

(Refer Slide Time: 01:16)



So, the problem is that of a cantilever steel bracket which supports a vertical load some magnitude let us say 2 kN, and a moment 1 kNm. And we see this is the bracket problem, and the dimensions all dimensions are in mm. So, its depth is 200 mm at the support. And then it is gradually tapering to have 100 mm depth at the free end. And then there is this point load P at a distance of 75 mm from the free end, and this hugging moment at the end.

And thickness is 6 mm and that you can see in the plan and side view. So, we discretize this domain for analysis, we, of course need to divide it using finite elements. So, we there are several possibilities of doing that. And we dis few of the possible meshes are shown here. So, one mesh is of combination of rectangular elements and a triangular element, all linear complete up to first degree polynomial, so four node rectangle and three node triangle.

So, 3 elements of 4 node rectangle, and one element of three node triangle; or alternatively we could model this entire problem using triangular elements by dividing it into all these rectangles also into two triangles, and that is the mesh that we see here 3 noded triangle all through. So, while in previous case the entire problem domain, we could model using 4 elements. By using triangular elements, we need 7 elements to cover the entire domain. So, 3 elements extra in this; such a small trivial problem.

Now, we can also consider for higher accuracy and we can consider using elements. So, which is approximation is wherein the approximation of primary variable is complete up to second degree. So, we use here in this particular mesh we have 3 rectangular elements, 8 node rectangular, so serendipity element. And then 6 nodded triangle so linear strain triangle. So, this is all these elements are complete up to second degree polynomial approximation variation. Again we can possibly divide these rectangles into triangles. So, model the entire domain by using 6 noded triangles.

And there would be obviously, same number of elements 6 7 elements in triangular discretization, and 4 elements in case of combination of rectangle and triangle, but the degree of interpolation is higher. So, number of variables are also going to be much larger. So, this discretization of the domain in this by using finite element mesh, this is what we call as preprocessing step of finite element analysis that is preparing the data for actual finite element computation.

So, this discretization and then mapping the geometry, locating the specifying the coordinates of the nodes, and how each element is defined by which curve which nodes, and what are the coordinates, and what are the order in which the nodes are aligned nodes are defined for the element, these are all the steps that are covered under what we know as pre-processing step of a typical finite element analysis.

Then after this discretization effort is completed, all these discretization data is all in place, then we take out take up one element at a time, and establish the element equilibrium equations. And this is the basic processing step. The finite element, finite element processing, the conversion the process of conversion of the differential governing differential equation into algebraic equation that happens in this particular process.

And also after the algebraic equations, element level algebraic equations, we assemble them into global equations impose the boundary conditions, and then solve for unknowns of the problem. All these are referred to as the processing basic finite element processing.

And subsequently after the primary variables nodal values of the primary variables are determined, then we go back to element level equations, and substitute these computed primary variables nodal variables. And compute what are the secondary variables of the problem that is the stress components or attractions in this case or whatever I mean appropriate secondary variables. And that is called what and then we can present the results in graphical format or whichever way.

So, the those are the steps which are covered under what we call as post processing steps, post processing of the finite element analysis. Typical finite element analysis run ends at the with the computation of the solution of primary variables of the problem. And subsequent analysis is covered under post processing.

And these post processing is also very important in the sense that it gives us very good idea, very useful idea about the possible errors in the solution because step we need to keep in mind that at the end of the day finite element analysis is an approximate analysis. And for any approximate analysis, it is very crucial, it is crucially important very important to keep track of what is the error of approximation.

And this error of approximation is monitored is can or it can be assessed by using these post processing steps, the error analysis is based on these first results of the post processing step. And then it is a call is taken whether a further refinement is necessary whether this mesh that we started with whether it is good enough, or whether it needs to be refined.

For example, refinement can be of two types, either I can reduce the size of the element and increase the number of elements to cover the whole domain that is referred to as H refinement. H referring to the size of the element. So, we can reduce the element size. Or second possibility is what we have covered here that is by keeping the same mesh, but by using higher degree interpolation, so that will necessarily involve incorporation of additional nodes in the mesh.

And that will again require re calibration or a preparation of the mesh connectivity data, and, but this is brought since the number of elements remain the same, and the element geometry remains the same. It is only the additional nodes are added and that helps in increasing the degree of polynomial interpolation. So, this is referred to as P-refinement, refinement with respect to polynomial degree of interpolation.

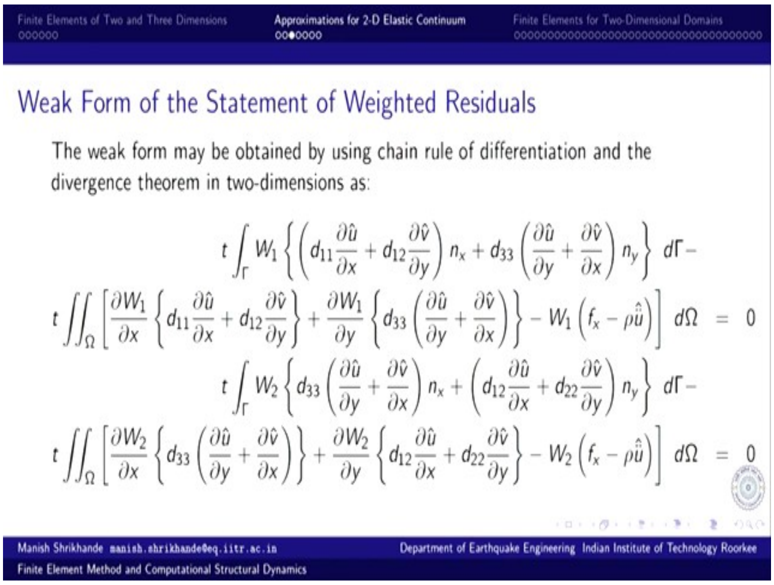
So, both of them are often adopted, more common is H-refinement, P refinement is not very often encountered, but it is quite often a matter of choice and convenience, but that is how we refine the finite element analysis based on the results of post processing step. So, let us get on with the analysis. So, these are the material properties.

So, Young's modulus that is the Young's modulus of steel that is given steel bracket, then Poisson's ratio 0.3, and density row is the density. Now, what kind of problem is this? We model this. I mean we can look at this problem as this dimension thickness is 6 mm compared to length of 300 mm and depth of 200 mm.

So, there is a two order of magnitude difference between the thickness dimension and other two dimension. And there is no force or no applied force normal to the plane or along the thickness direction. And there is no variation with respect to thickness in the application of the load because thickness is such a small 6 mm thickness really very, very, very small magnitude small distance.

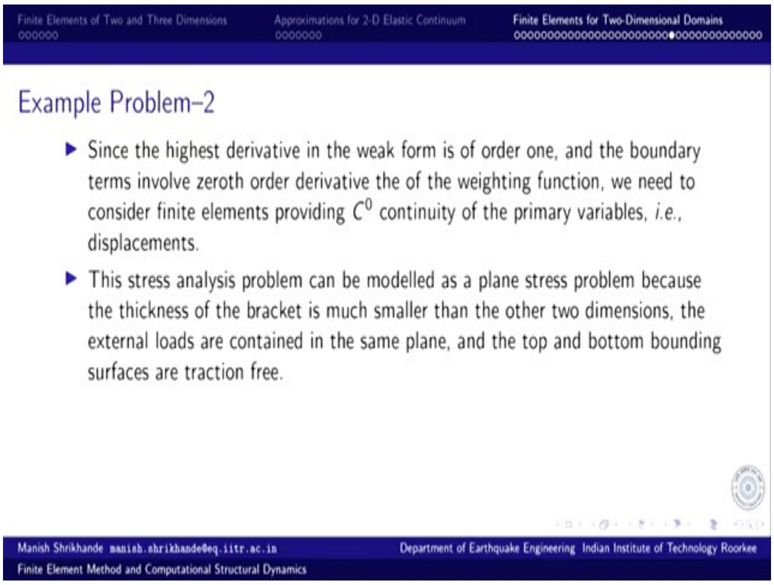
So, we do not expect nothing anything significantly changing in this span of 6 mm. So, all the assumptions of plane stress conditions are quite valid in this particular problem. So, we model this as a two-dimensional plane stress analysis model. And it is a stress analysis problem. So, second order or differential equation that is the governing differential equation of motion of the problem. And the weak form, in the week form the highest degree of derivative is order 1. And boundary terms involve 0th order derivatives of the waiting function right.

(Refer Slide Time: 12:12)



So, we can look at this. So, this is the weak form of the weighted residual statement for two dimensional problem. So, this is the boundary term. And waiting function has the appears in the form of 0th derivative. So, the primary variable is the displacement, unknown variable of the differential equation, so u. So, primary variable is u and v. So, displacement along x and y. And the first degree so the continuation of 0 degree interpolation, 0 degree of the primary variable is required across the element boundaries.

(Refer Slide Time: 13:02)

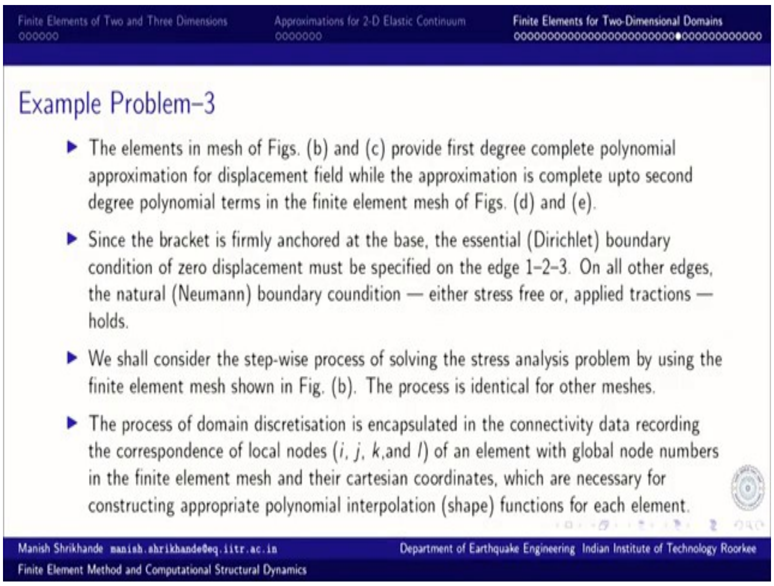


So, C 0 continuity elements are required, primary variables. And primary variables or of course, the displacements along x and y direction, and x direction we consider it to be in this direction and this is of course y. So, x and y; x is along the length span of the cantilever, and y is transverse direction.

So, this stress analysis problem can be modelled as a plane stress problem, because the thickness of bracket is much smaller than the other two dimensions. There is a two order of magnitude difference. The external loads are contained in the same plane as the x y plane, and the thickness direction both the planes are free from traction. There are no loads acting along the thickness direction on either side.

So, some of the possibilities that we have seen what are the possibilities of discretizing this domain using finite element. We can use finer mash if required, but that will increase the problem size. So, we make use of the simplest possible mesh that is this three rectangle and one triangle. And use this for further analysis just for the sake of demonstration. Of course, the steps of finite element analysis remain identical. It is only the number of calculations that will increase if I we take more number of elements or more number of nodes in the analysis.

(Refer Slide Time: 14:44)



So, elements 4 noded rectangle and 3 noded triangle they divide they provide the first degree complete polynomial approximation for displacement field. And for these 8 node rectangle and 6 node triangle, the approximation is second degree complete, and it is more accurate finite element more accurate approximation.

And since the bracket is firmly anchored at the base, let us say restraint at the base. So, the essential boundary conditions or Dirichlet boundary conditions are implied or zero displacement are implied at the edge 1 – 2 – 3. What we are referring to as 1 – 2 – 3, this one. So, edge 1 – 2 – 3. So, all these three nodes 1, 2 and 3, the displacements u and displacements y along on these nodes, they are to be restrained to zero values to correspond with this fixity condition in the original problem.

So, on all other edges, there is a natural boundary condition on Neumann boundary condition, it is either stress free or some applied tractions. I mean only one element has the applied load as we can see element number 3 will have this applied traction applied load P and this moment M. So, these are all these are actually covered by in the sub domain represented by element number 3. So, element number 3 will have range from this point to this point and in this particular rectangle.

So, we shall consider I mean as I said step wise process or solving this stress analysis problem by using the finite element comprising 4 node rectangle and 3 node triangle the process is of course identical for all other cases all other measures its only the number of elements will increase or number of nodes will increase and that is all it really does not make any difference to the user apart from preparing the input data. So, preprocessing step, rest of all is taken care of by the computer.

The process of domain discretization is encapsulated in connectivity data. As I said we need to make sure that the how the problem is decomposed using finite element individual sub domains representing each element, and how they are juxtapose together to cover the entire domain that needs to be captured accurately.

And this entire domain representation or domain discretization, discretization of the problem domain into smaller sub domains or finite elements and how these elements are placed together with respect to each other, how they are interconnected or what kind of boundaries they share, what are the common nodes between these elements, these all this information needs to be encapsulated and in a readily accessible data structure in a computer program, so that this information can be derived as and when required during the process of computations.

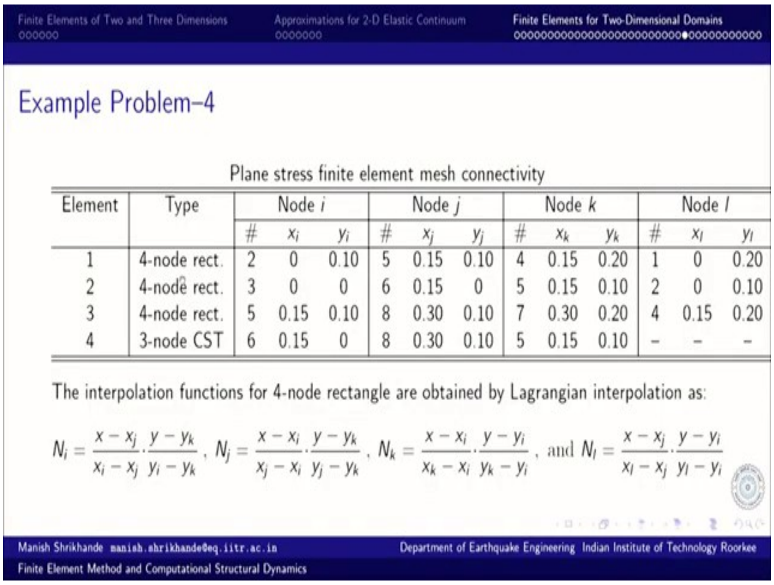
So, this is what we commonly refer to as connectivity data or connectivity array. Array or the arrays or the simplest data structures provided by any programming language, and it is very conveniently I mean all these data finite element data is very conveniently stored in one-dimensional or two-dimensional arrays as the as per requirement.

So, in case of 4 noded rectangle, there will be 4 nodes. And as I said 4 nodes that we refer to i j k l or 1, 2, 3, 4, local numbering, we number them into in the counter clockwise direction. For rectangle, it will be 4 nodes; for triangle, it will be 3 nodes; i j k or 1, 2, 3 in counter clockwise direction. But these numbers are local to an element.

So, whether I consider element 1, element 2, element 3, the generic element definition is same. What differs is the placement or the coordinates of the nodes which define these elements, and which node takes which position i, jth, ith position or kth position in different elements. We will see that.

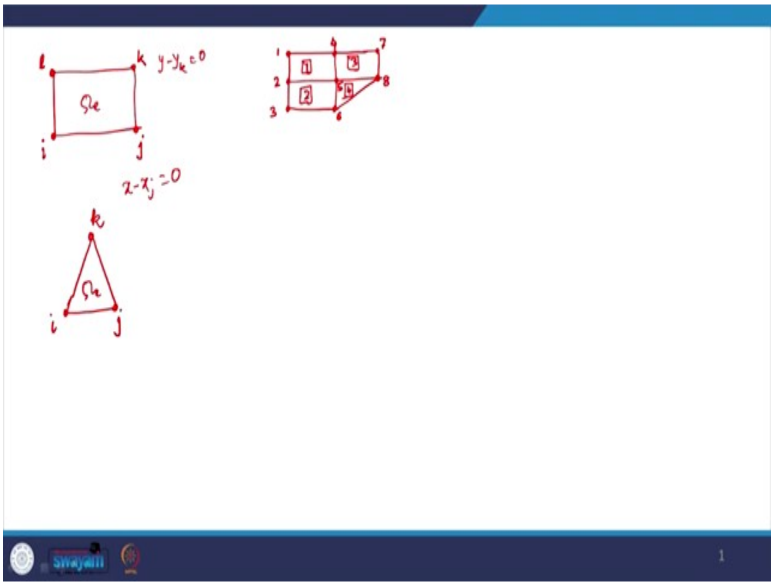
And these Cartesian coordinates I mean the position of these nodes and they are defined by the coordinates of the nodes in the chosen frame of reference. And this will allow us to construct the suitable approximation polynomial for each element uniquely. So, the connectivity data, the nodal connectivity data or mesh connectivity data that I was referring to.

(Refer Slide Time: 20:21)



So, there are 4 elements 1, 2 and 3 or 4 node rectangle, 4th element is 3 node constant strain triangle. So, what are the possible nodes?

(Refer Slide Time: 20:36)



So, what I am referring to here is there are for the rectangle, what we are referring to are these nodes. This node is called i node, this is jth node, this is kth node, and this is lth node, so 4 nodes in all. And as you can see the numbering is always done in counter clockwise direction. And similarly for triangular geometry, this is, so i, j, and k right. So, this is for element one particular element. So, triangular element, rectangular element.

Now, the element formulation inside the when we program the computer we only refer to these local references because that is how the coordinates what are the coordinates of node i of this element, what are the coordinates of node j of the element, what are the coordinates of node k of the element, and what are the coordinates of node l of the element.

And based on those variable, those are to be stored in the variable from the data that we are preparing. And the program the computations are programmed using these generic data in the reference of these local node numbering right. So, during the process of finite element data repression, what we need to first of all do what is the actual node number in the mesh discretization. For example, in this particular problem we are dealing with 1, 2, 3, 4, 5, 6, 7, 8.

So, these are the node numbers that we have. So, this is these are the node numbers total 8 number of nodes in the global system of problem the do total domain is discretized in 4 elements using these 8 nodes. Now, element number 1, for element number 1, node i refers to node 2 in this global mesh, total finite element mesh for the entire domain. Node j refers to node number 5; node k refers to node number 4; and node l refers to node number 1. So, this is for element number 1.

For element number 2, node 2 becomes node number l for element number 2. For element number 2, node 5 which it shares with all other 3 elements, so node 5 or node I mean in the case of element number 2 corresponds to kth node in this counter clockwise local ordering. And similarly node 3 refers to i, and node 6 refers to jth node for the 2nd element.

And this is what is encapsulated here. So, the cord node i prefers to node 2 in the mesh global mesh, and these are the coordinates of these mesh, these nodes. So, 0, 0.1, so this these are in the in terms of meters. So, coordinates are in units of meters I mean we are choosing the frame of reference at the bottom of the node and support.

So, ith node, jth node, so node number 5 is the jth node of for element number 1; node number 4 is the kth node. And node number 1 is the lth node. And these are the coordinates. So, node number, so this is the node connectivity, nodal connectivity of element. And along with, we also have the coordinates of these nodes.

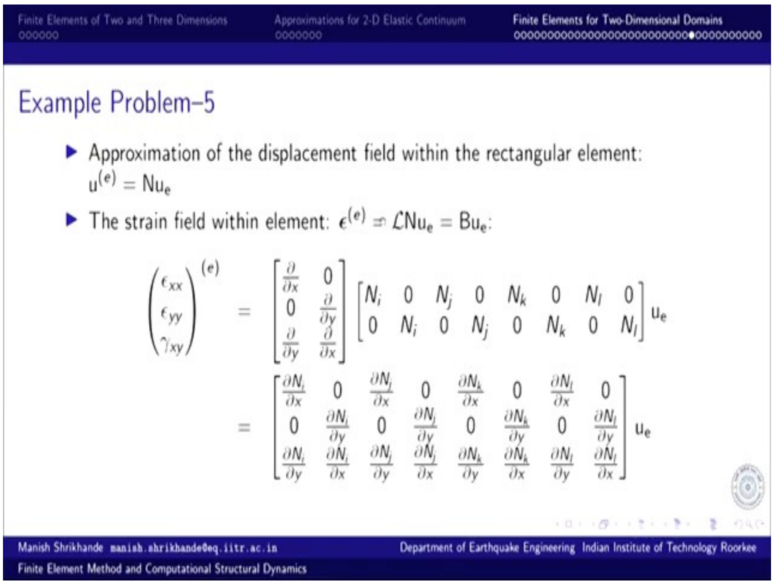
So, using these coordinates, we can define the interpolation function of 4 noded rectangle is defined by very easily defined by linear interpolation, linear in x and linear in y, and that is a simple Lagrangian interpolation formula of one-dimensional Lagrangian interpolation in x multiplied by one-dimensional Lagrangian interpolation in y. And that is what happens here.

Node i, for node i, we just multiply with the Lagrangian for node i. So, we x minus x j normalize to have unit length hat ith node. Similarly, for all other nodes in case of y direction, we take the coordinates of node k. So, this is what for node i for x coordinate, we take the x coordinate of j; and for y polynomial we take the y coordinate of node k and that is same as l.

So, with respect to y variation, this equation would suffice y minus y k is equal to 0. And with respect to x variation, this equation would suffice. So, we multiply these two equations and normalized to have this product x minus x j multiplied by y minus y k to have unit length at node i, and that is what we do here. Just evaluate this numerator at node i, and divide by that as a denominator.

Similarly, we do this for node j bilinear, and product of 2 first degree Lagrangian interpolation polynomials, and node k node l. So, with these, we have the interpolation model defined as you can see. Once we have these nodal connectivity and the coordinates of these nodes, the interpolation functions for all these nodes are uniquely defined.

(Refer Slide Time: 27:44)



So, using these interpolation functions, we can approximate construct the approximation of primary variable displacement field within the element domain. So, u within the element as a function of x and y is given by interpolation matrix, matrix of interpolation function, nodal variables multiplied by the variable nodal values of the primary variable. And this strain field is of course given by the derivative differential operator on this displacement field.

So, this is this matrix is referred to as the strain displacement matrix. It relates the nodal displacement to the strain field. So, this B matrix which is actually derived from the interpolation function by operating on it the differential operator of the governing differential equation, and that is what gives us strain displacement matrix.

So, the strain field is given by B times u, so strain displacement matrix multiplied by the nodal displacements. So, how this strain field with using this strain field we compute we arrive at we develop the finite element equations the equilibrium equations at the level of finite element, we will discuss it in our next lecture.

Thank you.