Finite Element Method

Prof. C. S. Upadhyay

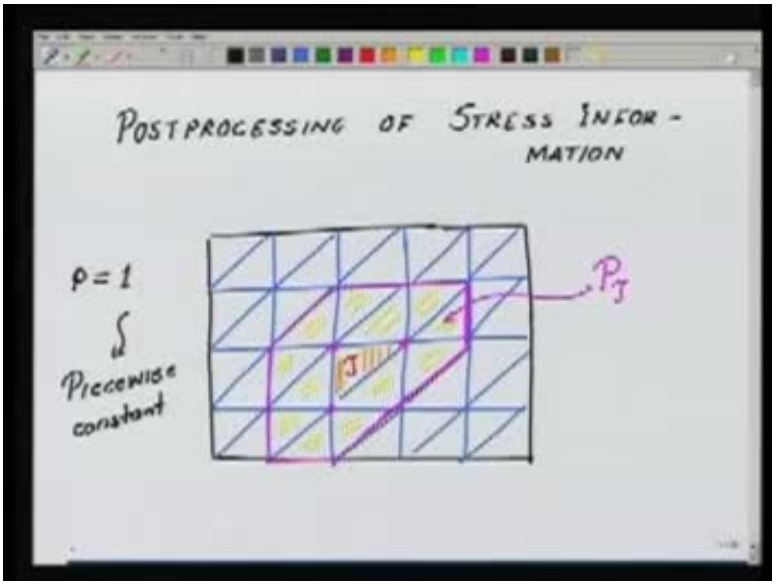
Indian Institute of Technology, Kanpur

Module No. # 10

Lecture No. # 1

So, in the last class, we discussed the beam stress problem and the beam strength problem of elasticity, planer elasticity, and there we saw how we can do the finite element formulation from the weak formulation that we derived. We also discussed how to handle inclined boundaries, boundary conditions imposed on inclined boundaries, because in an actual structure, we are going to have tractions and displacements, specified normal and tangential to the edge.

(Refer Slide Time: 01:28)

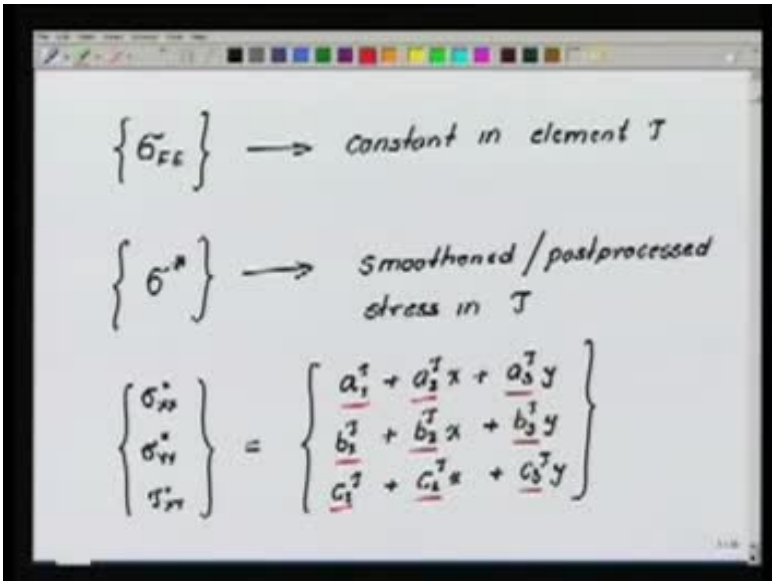


Here, we are going to continue a discussion further with a planer elasticity problem. And, as we had done in the one-dimensional problem of the bar and the beam, we will continue here with the desire to obtain a better state of stress from the finite element data that has been obtained; that is, we are interested in the post processing of the stress information.

So here, what do you mean by post processing of stress information? So, for that, let us take a simple example; that is, I have a domain, let me draw a big domain; let us, in the domain, I make a mesh; so, I make a mesh, I will be making a mesh of triangles. Here is mesh of triangles. Let us say that I am interested in obtaining the stress information in this element tau, in this element tau; so, what do I do? I take a one element neighborhood of the element, that is, I will take all these elements; this one element neighborhood of the element of interest forms a patch; this will be patch corresponding to the element of interest tau. So, if you understand that here what we have done is, you see that I have taken all the elements which share either a node or an edge with a given element.

All these, the setup, all these elements including this element, forms the patch which I have marked this line, it forms the patch P tau; so, this is the one element neighborhood of the element tau. Given this patch P tau, now, let us see how we can construct a better stress information in the element tau. So, remember, our goal was the element tau. So, let us say, just to fix ideas, we take the order of the approximation p is equal to 1; that is linear approximation. In this case, my state of stress would come out to be, because the strain will be piecewise constant; that is, constant in the elements, so, the state of stress would be piecewise constant.

(Refer Slide Time: 05:26)



So, now what do we do? We say that we will have the stress vector, we are writing in the engineering connotation, coming out of the finite element solution sigma FE. This is

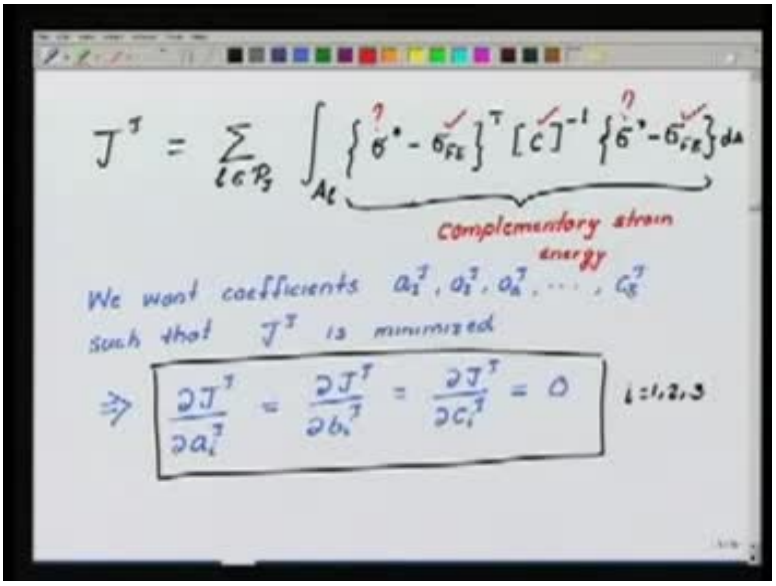
constant in element tau, and it is piecewise constant in each of the elements, so, in the patch P tau. So, then we say, that out of this we want to get a so-called better or post processed state of stress in the element tau, given by sigma star; smoothened or post processed stressed/stress, in tau. How do I represent the sigma star? So, sigma star will have sigma star XX, sigma star YY and tau star XY; these are its components; and what we do?

We are going to represent them using the same order polynomials as the order of approximation. So, in this case, the order of approximation p was 1, so, for fitting the sigma star, we are going to use a linear functional representation over the element tau, actually, over the whole patch P tau. So, what we are going to do is, so here it will be equal to a 1 tau plus a 2 tau x plus a 3 tau y. Similarly, sigma YY star will be b 1 tau plus b 2 tau x plus b 3 tau y, and this is c 1 tau plus c 2 tau x plus c 3 tau y.

So, remember, what we have done. We have taken the post process or the smoothened stress in the patch P tau to be a polynomial of the same order as the order of approximation, while the finite element stress would be of an order one less than the order approximation, because if the order approximation is p, the derivative of the functions u, v, w will be of order p minus 1; so, the stress will be of the order p minus 1. So, this is one order less; then the order of approximation, this is of the same order. We could do different things also, but this is what we are going to stick with; this is the simple procedure and that is what we are going to use.

Now, what do we do need to find, as far as this is concerned? If we can get these coefficients a 1 tau, a 2 tau, a 3 tau, b 1 tau to b 3 tau and c 1 tau to c 3 tau, then I have the representation of sigma XX star, sigma YY star, tau XY star in the element tau; but remember, that to get these coefficients, we are using the information from the patch P tau; so, what do we do?

(Refer Slide Time: 09:50)



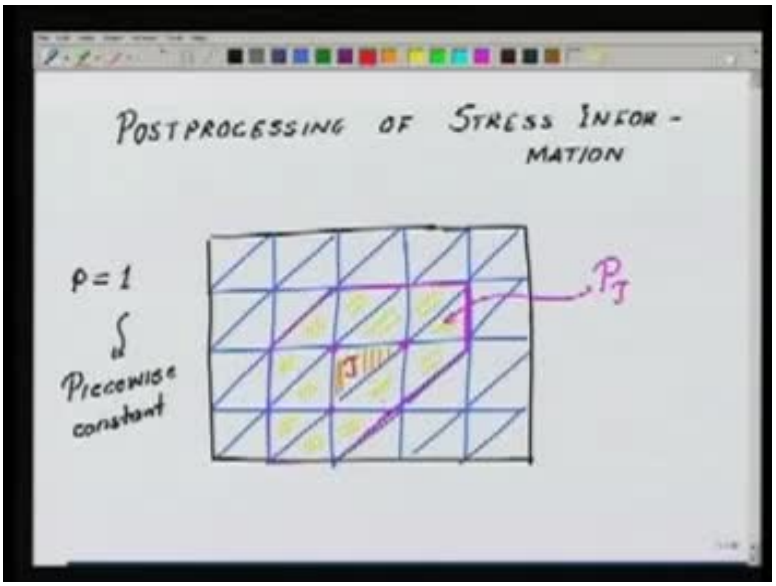
Then, we define a functional associated with the element tau; this is equal to sum over all the elements l belonging to the patch P tau, which includes the element tau of integral over the area of the element l into the vector sigma star minus sigma FE transpose into c inverse into the vector sigma star minus sigma FE dA. See what we have done. This quantity is actually, this is the complement energy corresponding to the stress given by sigma star minus sigma FE; so, this is the complementory energy of what we would say as the error between the smoothened stress and the actual finite element stress.

So, we have defined this function. What do we want? We want our coefficients; so, we want our coefficients a 1 tau, a 2 tau, a 3 tau, up to c 3 tau, such that, J tau is minimized. So, you see that we have defined a functional, this functional will always be positive whatever be your stress, state of stress, right? So, what do we want? We want these coefficients a 1 tau to a 3 tau, b 1 tau to b 3 tau and c 1 tau to c 3 tau, in such a way that it leads to the minimum value of J tau.

What does it mean? That, this implies, we have del J tau del a i tau is equal to del J tau del b i tau; this is equal to del J tau del c i tau is equal to 0. This is the condition that we have for i is equal to 1, 2, 3; so, you see, this case, me 3, 3, 9 equations, it gives me for the 9 unknown coefficients a 1 tau, a 3 tau; and remember, that in this patch, this quantity, this and this are known data.

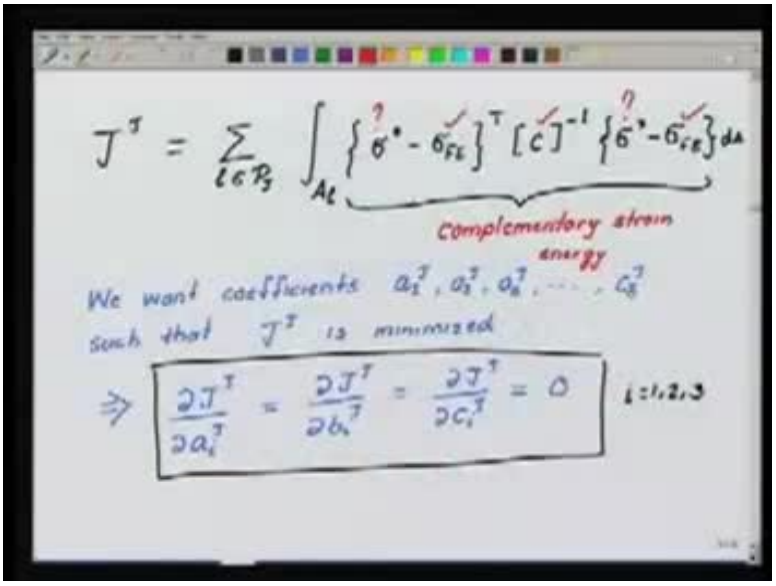
This sigma star is the one that we are trying to solve for. So, just like with did in the one-dimensional case and for the bar problem and the beam problem, we can do the same thing here, to set up the system of equations in terms of the coefficient a i tau, b i tau, and c i tau, solve, get the representation of the state of stress, recover state of stress in the element tau.

(Refer Slide Time: 14:08)



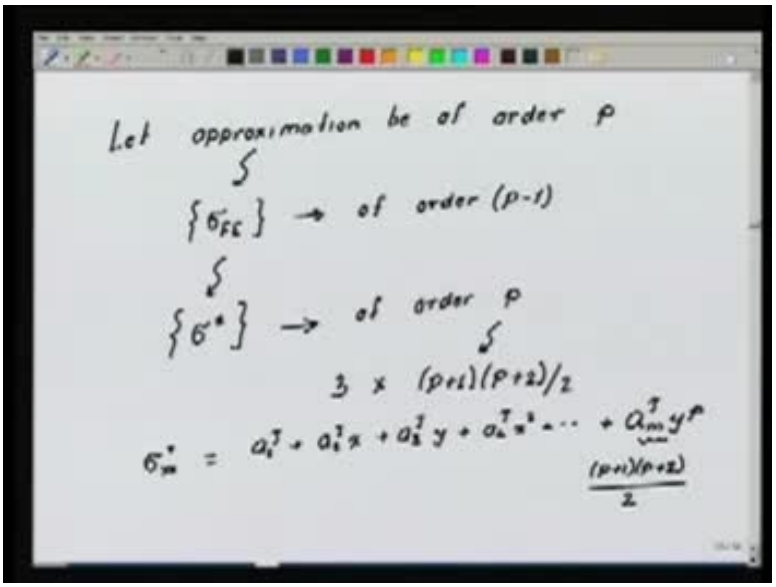
This way I keep on moving, so, I go from one element to the next element. So, if I am interested in this element tau, I take the one layer neighborhood of that element tau. If I am interested in the next element, I take the one layer neighborhood of that element. So, this can be nicely written in an algorithmic way.

(Refer Slide Time: 14:31)



And you see, what again is the advantage? Let me reiterate it, that here, we are solving small problems over each element; for example, here the size of the problem is 9 by 9. Solving the small problems does not take any time; in fact, it takes negligible time with respect to the solution of the actual finite element problem. So, this job gets done very quickly as compared to the cost of obtaining the finite element solution. And that should be one of the criteria used for deciding whether oppose processing technique is going to be useful to us or not. If I have explained a lot of effort in post processing, it is better to resolve the solution with a higher order approximation.

(Refer Slide Time: 15:25)

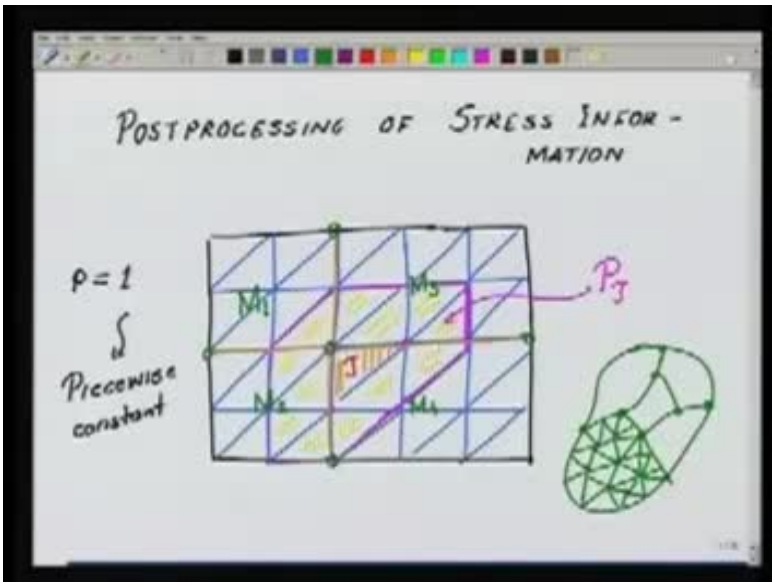


So, how do we generalize? So, generalization is, if approximation, let approximation be of order p; if the approximation is of order p in all the elements of the domain, then this implies, the finite element stress is of order p minus 1 as we said; because, the strain is of order p minus 1, so, the stress becomes of order p minus 1; so, then, what we do? We want sigma star, the recovered stress in the element tau, to be of order p; so, in this case, each component of the recovered stress vector, as the way we have written, will be a polynomial of order p.

So, how many terms will each one will have? So, it will have essentially, so, each components will have p plus 1 into, from the Pascal's triangle, in p plus 2 divided by 2 terms, this into 3 will be total number of unknowns for which we have to solve in order to get the representation for the recovered state of stress. So, what will these, for example, sigma star XX will be a 1 tau plus a 2 tau x plus a 3 tau y plus a 4 tau for x squared, so, we continue, plus a m tau by to the power of p.

And, the same representation in terms of b, b i's and c i's for sigma YY and sigma and tau XY; and what is m? m is p plus 1 into p plus 2 divided by 2. So, we follow the same procedure that I had given earlier in order to get the recovered state of stress. What you can see then is this recovered state of stress will look a lot better and smoother as compared to the actual finite element data that you have obtained.

(Refer Slide Time: 18:11)

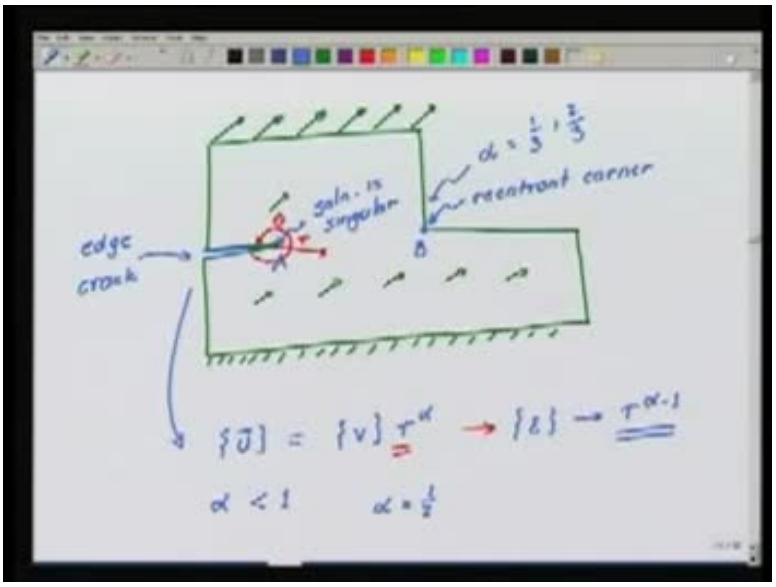


Let, me go back here and pose another problem. Let us say, that in this domain, I had taken a different material in this part, different material in each of this part; that is, I will have here, a material M 1 in this region, material M 2, M 3, M 4; here we have taken the boundaries of the material's domains to be nice straight lines. In general, they need not be like that; when you have a multi material domain, we will have a domain like this, with different material, in the various regions. In that case what should you do? You should make a mesh in such a way, that the boundaries of the material domains are honored, that is, the elements do not cut across the material boundaries.

And at the point of intersection, like these points of the material domain with a domain boundary or between two material domains, for example, here, at these points, you have to have a node. So, when you make a mesh, make sure that these points are nodes; then only you can go ahead and make the rest of the mesh; for example, I am making in this piece here; this should be very clear, otherwise, a finite element solution will be very far from the actual solution to the proper.

So, this may be could complete the mesh here, in this part, and we continue the same thing in the other region. This is one more thing that you have to keep in mind when you are doing the finite elements approximation, because, nobody tells you that the material will be always be have to same type; you could have different types of materials.

(Refer Slide Time: 20:38)



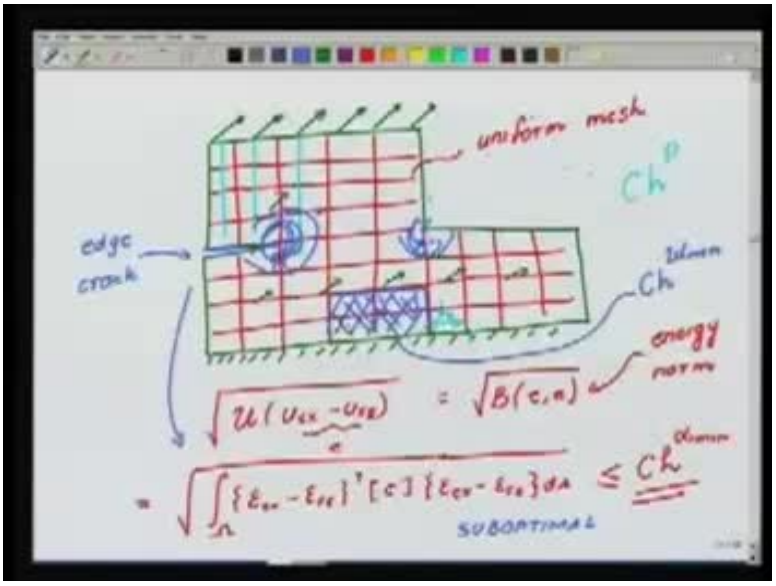
Let us go to another issue which now we have can handle. Let us say, that I have a domain like this, this is the domain; let us say, I have, for simplicity, fix this edge and I have some distributed forces and some tractions acting in different parts of the domain. What do we know? That this, this edge is actually a crack, this is an edge crack. At the crack tip, you will see that the solution is singular. What about this point? If you see, so this, let me call this point A, you see, this point B. Here, this is a so-called re-entrant corner; even here, the solution is singular. What does it mean, the solution is singular? It means the state of stress goes to stress components, go to infinity at these points; some of the stress components go to infinity.

So, in general, if I take this crack, the crack will have a representation as the vector u, the exact one will be equal to some vector v into r to the power of alpha. Now, this v is the function of alpha in theta into r to the power of alpha. What is r? r is the distance from this point to any point in the vicinity of the crack, so, this is the distance; and I can take the angle to go theta like this. So, I use a local pooler corner system in the vicinity of the crack, or again of the re-entrant corner, then the solution has this form; then because of which, the strain, the strain will have r to the power of alpha minus 1. And generally, this alpha will be less than 1; so, what happens? This quantity now becomes r to the negative power; if its r to the negative power, then as I come closer and closer to this point or to this point, this strain will go to infinite; that is what it means by singularity of the solution.

This alpha in the vicinity of the crack is half; if it is an isotropic material. In the vicinity of this corner, it will be alpha is, it could be one-third, two-third different quantities; so, you see that this tells me, depending on the various profiles of the re-entrant corners, even the crack can be, edge crack can be considered as a re-entrant corner, I will get different intensities of the singular behavior of the solution.

In this case, what do we do in order to get an effective finite element solution? So, remember, that in this case, if I used the standard mesh that we have, the standard mesh that we have then that form mesh, then the solution will be horrible. Actually, the rate of convergence of the solution is said to be sub optimal. Let me erase all these, so that I can make the mesh write here.

(Refer Slide Time: 25:34)

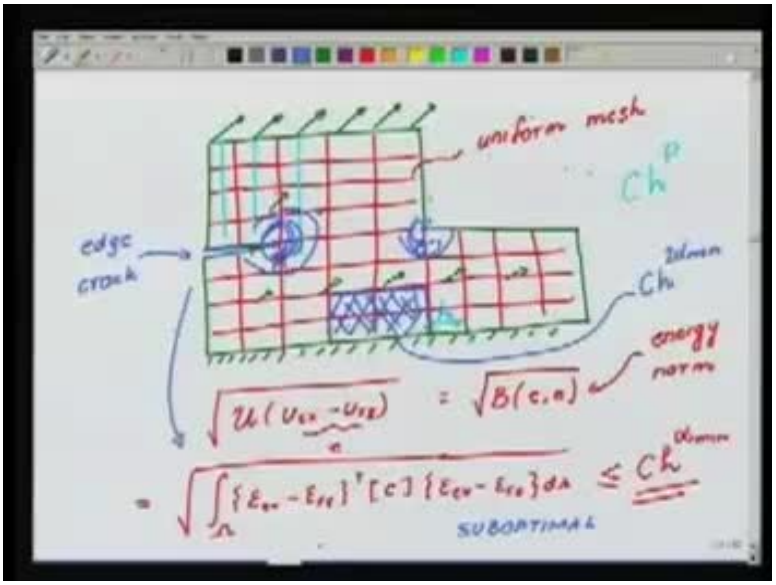


So, what we are talking of is the following; that we have a mesh, let me make the mesh some other pen. Let us say, I am making this kind of a mesh, it does not matter whether it is a mesh of quadrilateral rectangles; if I use this mesh, so-called uniform mesh, then the solution will converge at a sub optimal rate; that is, we define the quantity, let me erase this also, we define the quantity, strain energy of the u exact minus u FE, square root of that, this is actually equal to square root of the bilinear of form of; this is nothing but the error (e, e). This is equal to square root of integral over the domain of strain exact minus strain finite element transpose into c into strain exact minus strain FE dA; this is what we move mean by the so-called, energy norm; it is called the energy norm, but you understand it as a strain energy of the error, that is, the difference between the exact solution and a finite elements solution. I do not know the exact solution, but if I knew it, this is what it would mean.

This will converge; in this case, a C h to the power of alpha, where alpha was the dominant, alpha, I will it, alpha meant; so, it will be the minimum of the alpha corresponding each of these singular points. This is a very strong result which tells me, that in the actual, in the other case, when there were no corners and singularity, the solution was smooth; that is, if I refine this mesh. So, this mesh as size, mesh size h, that is, the elements are of size h, so, if I keep on refining this mesh, add more and more elements, then the error converges as C h to the power of; in the optimal case, h to the power of p, where p is the order of approximation that we have of taken.

But, in this case, if I have a domain with a crack and re-entrant corner, I will only get C h to the power of alpha min; that is, the rate of convergence is going to be sup optimal, so, this is sub optimal rate of convergence. So, in order to get a good solution, finite element solution, you have to refine the mesh a lot; a lot, which means, a mesh with which I may not be able to compute at all. Because, the number of unknowns will be so high that it will be impossible for my computer to handle that many number of unknowns.

(Refer Slide Time: 29:33)



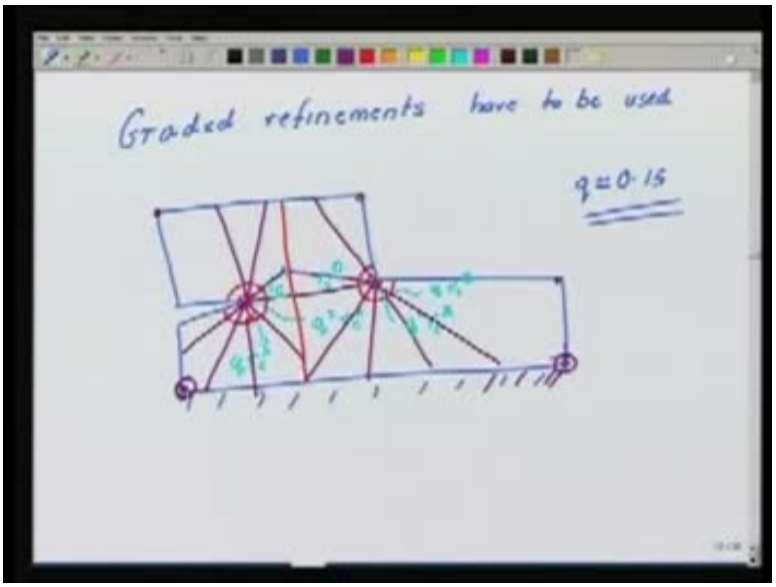
Next question, that people say is, well, I am not interested in what is happening here, I am interested in a region far away from any of these points, either from the crack or from the re-entrant corner; so, I know from Sangamon's principle, that the influence of this crack is only in the small vicinity of the crack, influence of this corner is only in the small vicinity of this corner.

So, does it matter? If I am here, whether there is a crack or not, a finite element solution should be good. The problem is that, this solution, this badness of the solution here, pollutes the solution here. So, if I am looking at strain energy of the solution here, and again, if I am looking at the error in the strain energy of the solution here, then that will converge as C h to the power of 2 alpha min. So, what it tells me is that, it does not leave me; the nature of the solution, the un-smoothness of the solution, elsewhere, affects the numerical solution here. Though, physically I know this does not happen, but numerically it is going to happen. So, my numerical solution is going to suffer. So, even

I am interested far away from this crack depth and so on, my solution is bad; at the crack tip, it is certainly bad.

And, as an engineer, we are mostly interested in what is happening in the vicinity of these critical points. So, even that argument is not so strong; so, we are interested in what is happening in the vicinity of the crack, so that I know whether I want to know, whether the crack under the given load, increasing load, dynamic load becomes critical or not; so that information, I would like to know out of the finite element computation.

(Refer Slide Time: 31:38)



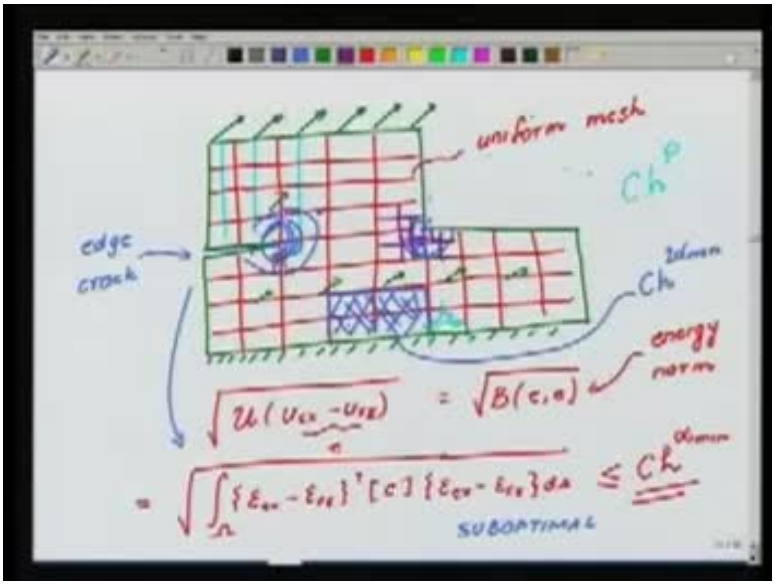
To do that, what is it that we have to do? To do that, we have to do graded refinements. The job is easy but it has to be done, so-called graded refinements; that we had discussed earlier also, have to be used. So, what we do have? Let us say, we have a domain here; so, what I will do is, in the vicinity of this cracks, I will make rings; these are called, these a graded rings, in such a way, that the radius of the inner ring and the outer ring are related. So, I take some characteristics distance which could be, let us say, the middle line; distance from the middle line to this crack tip; so, this could be my characteristics distance.

So, then this radius, this outer radius will be some q times r characteristic, well, the inner one is q squared r characteristics. Similarly, this one is r c, this is for a; this is r c for b, this guy will be q squared r characteristics; similarly, this one is r c, this is for A, this is r c for B and this is q square r c B. So, I will make these rings which are geometrically

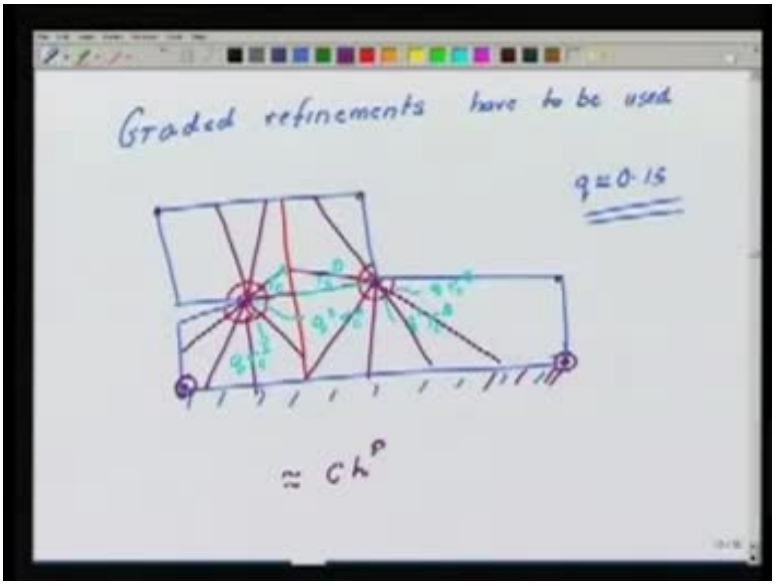
graded were q is equal to 0.15, generally, approximately, this will be what is taken; this is called the grading factor.

Now, I will make the mesh, such that, these rings are boundaries which have to be honored. So, now, you make the mesh, so, one way I can make the mesh is, let us make elements going like this and so on. I am not trying to complete the whole job, so, I will make these rings in this way. So, and then, I complete the mesh; so, what do I do? I do this graded refinement towards the singular corners. Actually, if we see in your solution, even this point will be a, these two points is also be a singular corner. These will be weak singularity points, will be logarithmic kind of singularities, but these two points will definitely also be singular points; so, I have to use graded mesh, such that the graded refinements are used in the vicinity of the singular point.

(Refer Slide Time: 35:02)

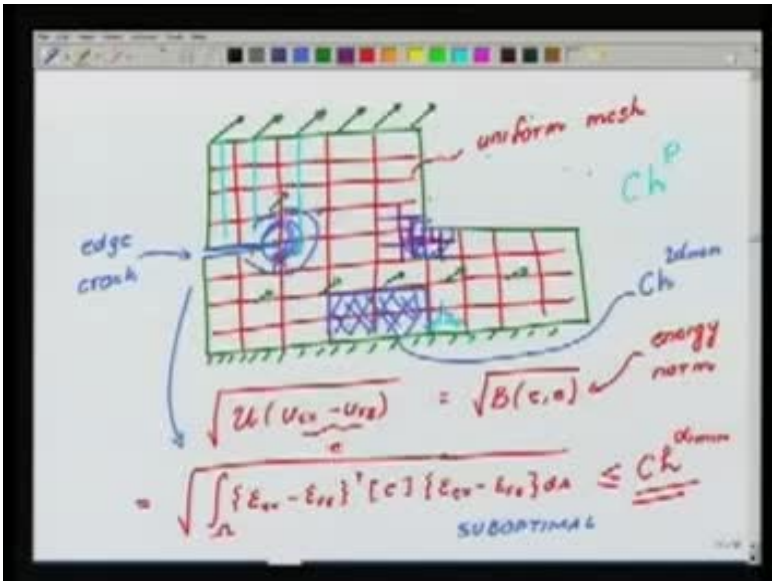


(Refer Slide Time: 35:20)



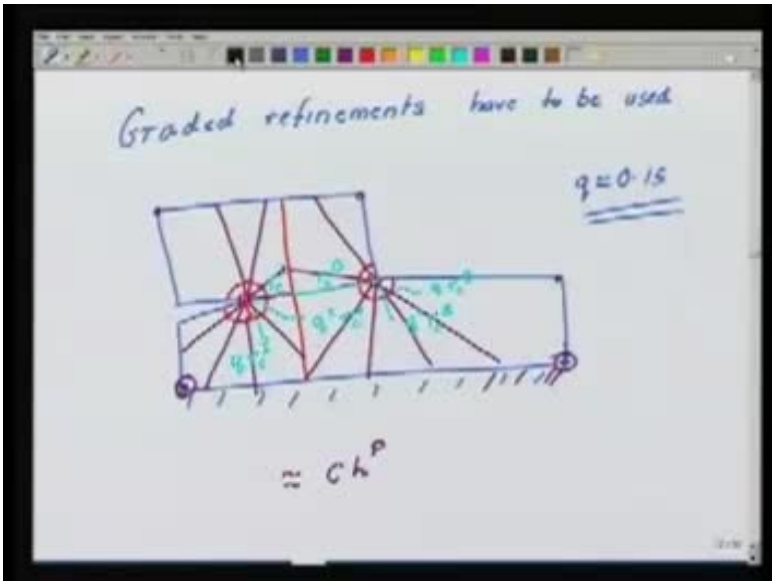
And, another way I could do the mesh is, take the previous mesh here and in this mesh, use some kind of grading here, in the vicinity of the corner and so on. There are various ways I could do it and if I do it, then I will get an essentially, C h to the power of p rate of convergence, that is, I will get the optimal rate of convergence, that is… Rate of convergence is not such an important issue as for as a user is concerned, it, what does it mean? It means that, for a given mesh, for a given number of unknowns, with this kind of a meshing arrangement, I will get a much better solution as compared to the uniform mesh with the same number of onwards and that is what we are after, that is, economy of computations.

(Refer Slide Time: 36:12)

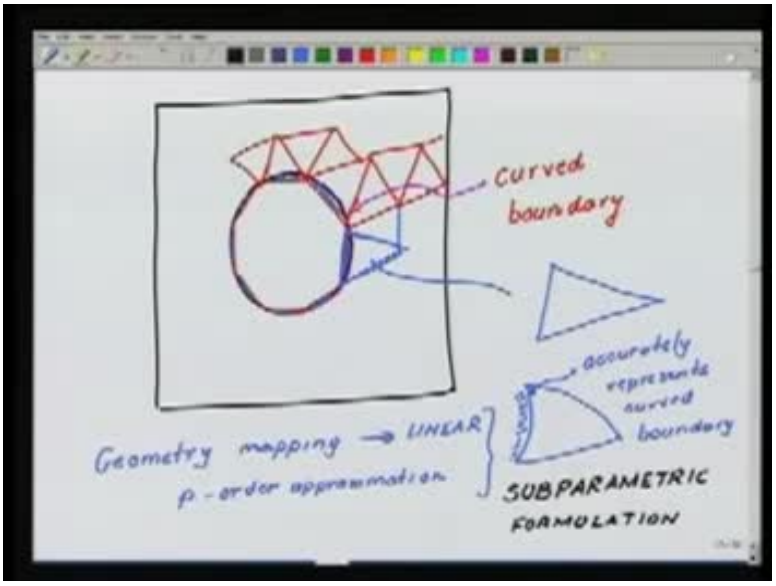


So, with this approach, with a graded refinement, we can capture singular behavior of the solution and then, just like I talked about earlier, in the crack tape, in the vicinity of the crack tape, there are methods by which I can abstract, what is known as the stress intensity factor, the k, that we can abstract with much better accuracy if I used this kind of a graded machine.

(Refer Slide Time: 36:35)



(Refer Slide Time: 36:50)



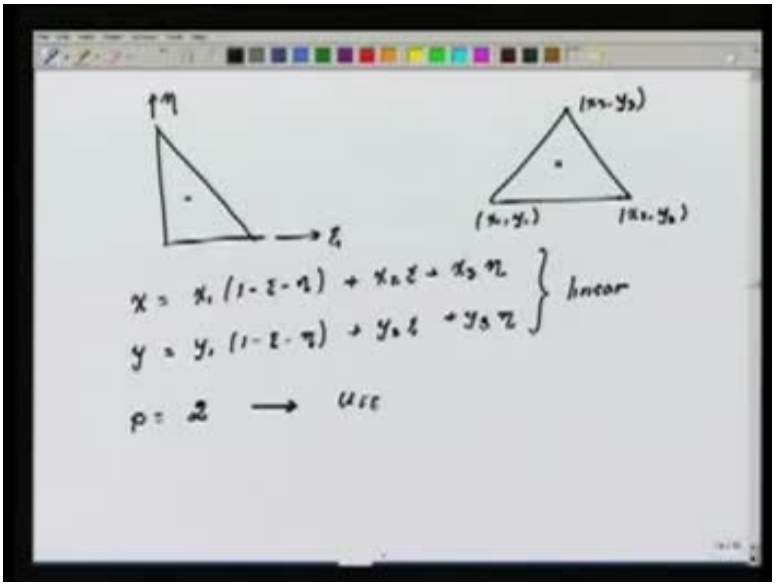
Let us look at another issue which is of interest to us as engineers. What happens if I have a domain with curve boundaries? I will take a very simple case of a domain with a cutout; so, this cutout profile, it is, say, it is a circular cutout, this represents a curved boundary. In this case, if we use the geometric mapping that we have talked about earlier, that is, if we make a mesh like this, if I make a mesh starting like this, let us say I make making a mesh here and so on; I am continuing the mesh in all directions. Then, you see that here, I am making, I am committing a lot of crime in representing the boundary of the curved surface. Depending on how fine or course in the mesh, the error in representing the boundary will be small or big.

In order to, if I use these linear elements, linear geometry, that is, the straightest elements I used. The straightest element here would lead to a crime in representing the curved boundary. In order to overcome that error, we will have to use a very finite element in the vicinity of the curved boundary, which is again un-economical. So, can we use a different way of representing the geometry of the elements, that is, can we use curved elements right away in order to capture the geometry better?

So, the question is that, here, instead of using the straightest elements, can I use a curved element, so that, this edge is accurately representing the curved boundary here. In case, when I have do not have curved boundaries, then all that we have done works perfectly fine; it is only when we have to deal with curved boundaries or curved domains, then,

you have to talk of curved elements. So, if you see what we had done earlier, as for as a representation of the geometry, so, geometry mapping was through linear earlier, while my finite element approximation was a p order approximation. So, remember that the representation of the geometry had nothing to do with my choice of the approximating functions. We use p order of approximation for the finite element solution, we represent the geometry using the linear map, because that gives as the exact representation of the geometry.

(Refer Slide Time: 41:25)

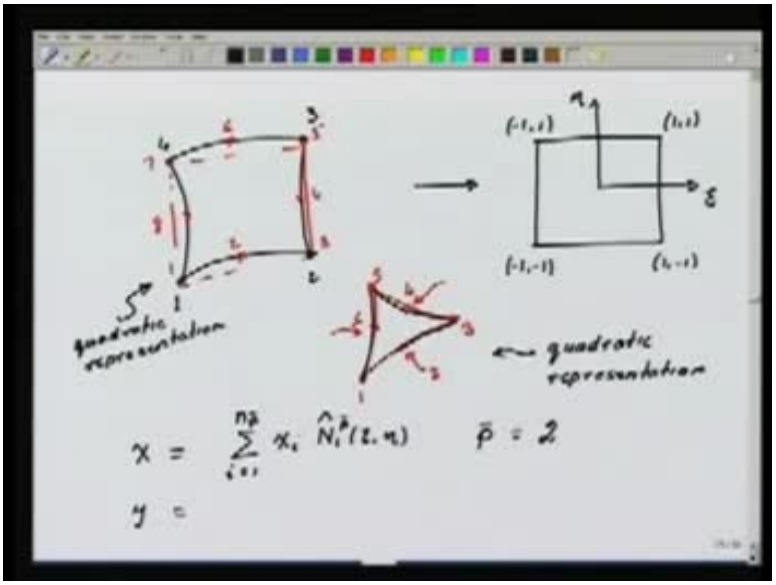


Such an arrangement in the language of finite elements is called a sub-parametric formulation; that is, we had representation of the geometry mapping from physical to the master and remember, that mapping was… this is my master element with a psi and the eta, and this is my generic physical element with coordinates (x 1, y 1), (x 2, y 2), (x 3, y 3). So, we had said that x was equal to x 1 into 1 minus psi minus eta, which turned out to be this basis function, of the shape function corresponding to p equal 1 plus x 2 into psi plus x 3 into eta and y was equal to y 1 into 1 minus y minus eta plus y 2 psi plus y 3 eta; so, this is for any point here, going to any point here.

So, this was a linear map; while for the approximation of the finite element solution, we used, let us say, p equal to 2; so, this was linear, that is, in terms of linear shape function, I could represent the mapping, while for the approximation, this is for the U FE, I could use p equal to 2, 3 and 4. We used it happily without any concern about all these things.

That is what we mean by a sub-parametric approximation; that is, the stress functions at the element level which I have used to represent the geometry, are of a lower order as compared to the safe functions which that are used to represent the finite element solution.

(Refer Slide Time: 43:24)



Now, there could be a situation, especially, this is something which is very popularly used; that I have this kind of a curved element which maps to this master element, which is again, if you remember, this is the point (minus 1, minus 1); this is the point (1, minus 1); this is the point (1, 1) and this is the point, (minus 1, 1). So, these are the four nodes, we can say this is node 1, 2, 3, 4; so, if added just using these four nodes, then I end up getting this representation; that is, the curved edges will not be properly represented.

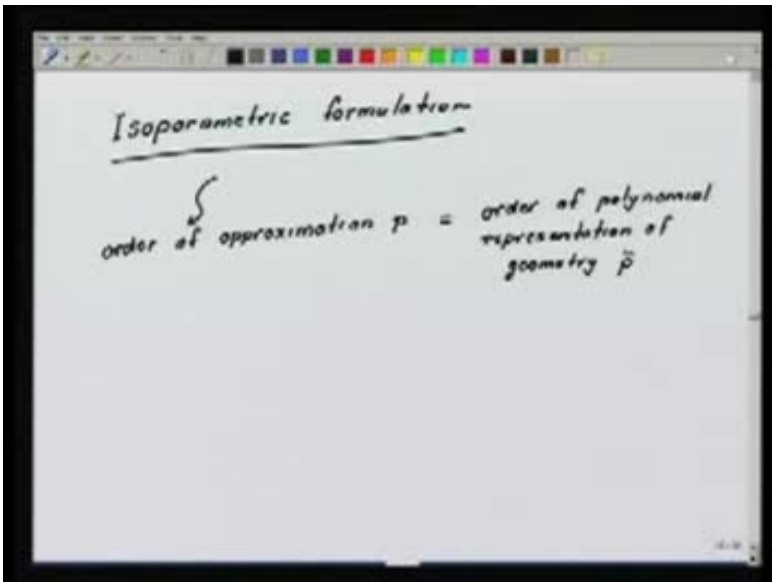
So, what we do? We say it is fine, let us put another node somewhere in the middle; we could do a serendipity mapping or if I have a triangular element, I will do, take these middle nodes, that is, I need to know these points in the middle of the these curved, is just, I need to know their coordinates, so, this will become point 1, this will become a point 2, this will becoming point 3, 4, 5 and 6; similarly, this will be 1, 2, 3, 4, 5, 6, 7, 8.

Then, in this case, I will have the representation of X is equal to sigma X i N i as the function of sign eta; here, I will call it as p bar, N i corresponding to the order of the approximation or order for the geometry given by P bar. If I have taken 6 points,

obviously, this will mean a quadric fit, quadratic representation of geometry; if I have taken 8 points here, it is actually quadratic representation of the geometry here.

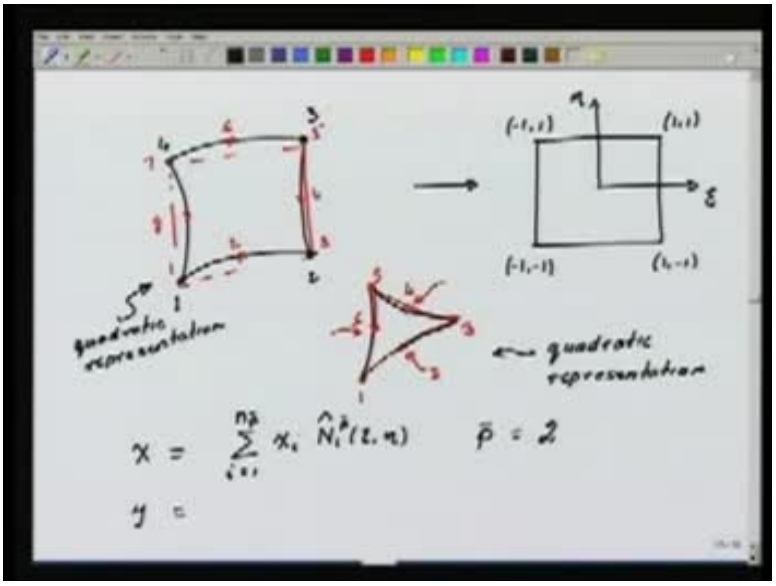
So, what we are doing is, you are saying that, in order to represent the curved edge, we take more points, line on the curved surface and do a fitting, as for as the geometry is concerned, of higher polynomials; that will be interpolating these curves using a higher order polynomial, each of these curves on these edge using higher polynomials; p bar, in this case, is equal to 2; sigma i is equal to 1 to n, corresponding p bar, number of terms corresponding, which for the case of triangle will be p bar plus 1 into p bar plus 2 divided by 2; while for the case in the quadrilateral, it is going to be dependent on what we have done for the serendipity representation. Similarly, I will have the representation of 5; if my finite element solution is also represented using quadratic shape functions, while we are, at the same time, using quadratic shape functions for representing the geometry or in the case when the finite element solution is represented using linear functions, and I will be using linear shape function to represent the geometry. In all these cases, the formulation said to be Isoparametric formulation.

(Refer Slide Time: 47:41)

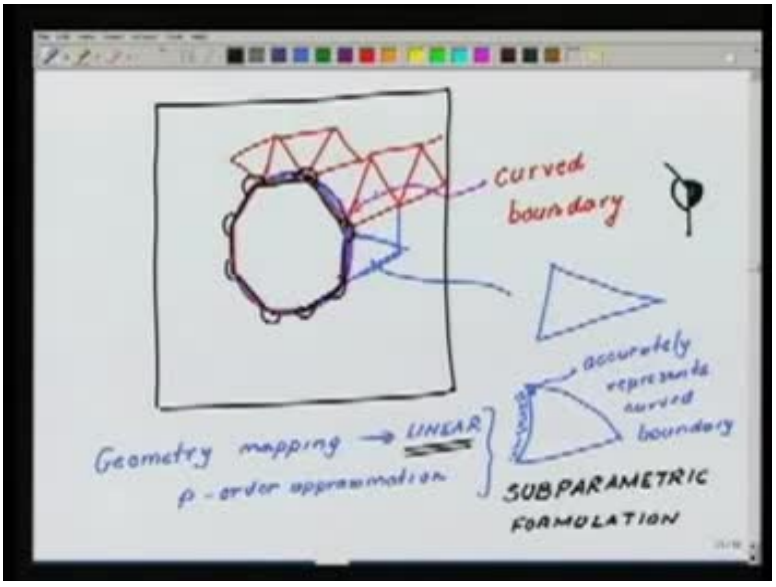


So, we have something called the Isoparametric formulation; that is, the order of approximation for the finite element solution p equals order of polynomials representation of geometry p bar.

(Refer Slide Time: 48:42)



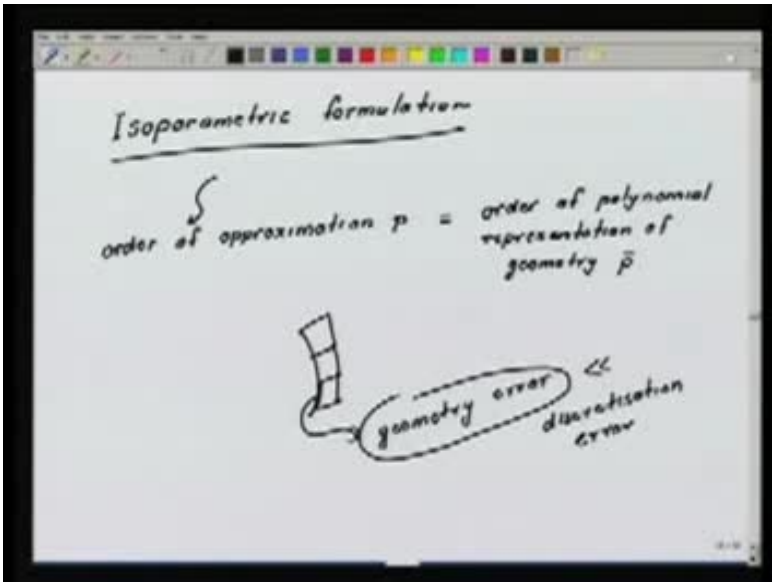
(Refer Slide Time: 48:48)



Let us go back here and see something interesting. My question is the following - that if this was actually polygonal cut out, then what will happen? In that case, even if we use a higher order representation for the geometry, the representation will degenerate in to the linear mapping that we had talked about. We will see that it will, even though you use higher order polynomials to represent the edges, you will end up getting all, these higher order terms will cancel out, and then getting linear approximation on the linear representation.

Second point, if I had used a linear mapping for a curved edge, what problem can it cause numerically? You see, that we had discussed a little while ago, that these corners are so-called re-entrant corners, because this angle, if you see this angle, sorry, this angle is an obtuse angle; you see, that all these angles here are obtuse angles, so, numerically, that is my finite element solution sees a domain which has these singular points, which has spurious singular points; it does not really happen in the actual problem, but my finite element solution does not know that; it sees this domain, the domain that is the mesh representation and it thinks, well, there are so many singular point, and then what happens? The solution, finite development solution will develop a spurious singularity and you will see that there will be lot of error everywhere because of this spurious singularity. That is why it is very important to represent the geometries, accurately for these curve edges.

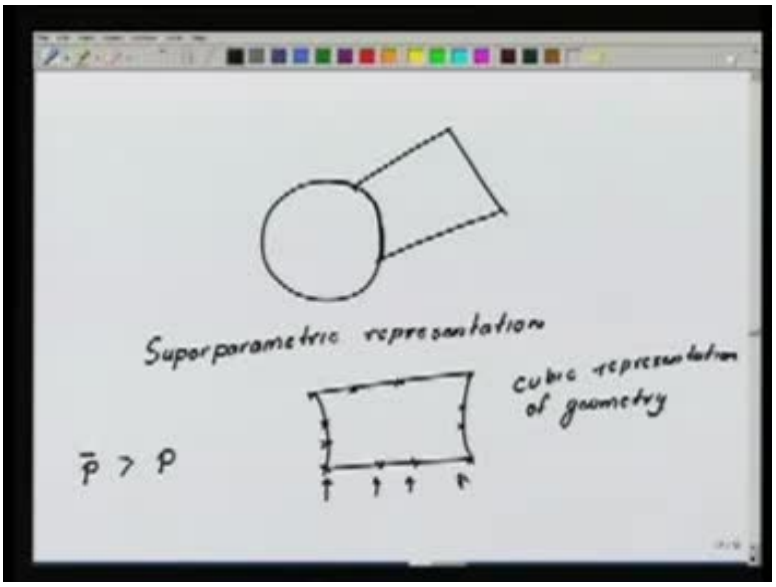
(Refer Slide Time: 50:44)



So, if I have this curved edge, I have to represent the geometry properly in such a way that the geometrical representation, the geometry error is less than the discretization error, in fact should be must less than the discretization error. Discretization error means, the approximation error; that what I get when I take the finite element solution as compared to the exact solution of the problem; so, this should not dominate my solution. So, generally, in the literature, isoparametric formulation is used, is a very popular representation, and for quadratic and by quadratic representation, this is often used; and

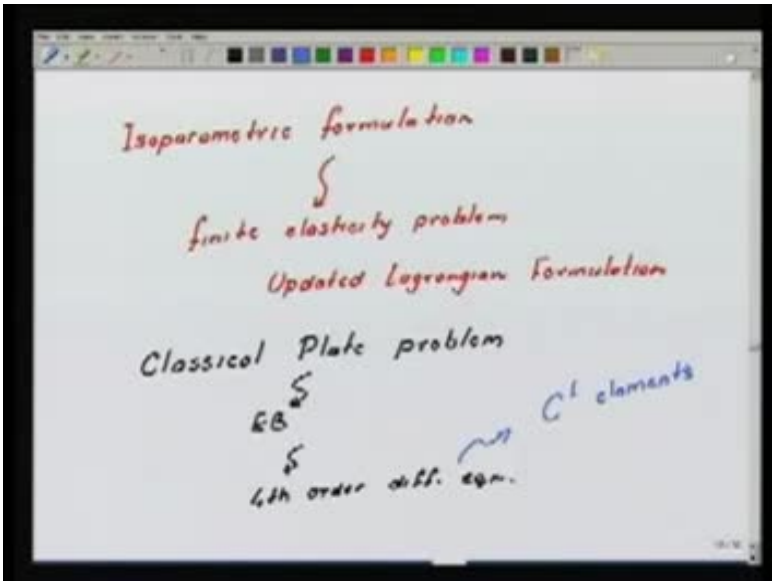
it does a good job as far as the representation of simple curve boundaries that we are talking about.

(Refer Slide Time: 51:50)



In some cases, when I am interested in using, let us say, large elements, so, I want to use large elements here; in that case, I may use a Super-parametric representation. In that case, I can take, let us say, have quadratic approximation. But, in order to represent the geometry, I use all these points; so, cubic representation of geometry, that is, p bar is greater than the order of approximation p. This is not used normally in the finite element applications, because the necessity does not arise; but not it may be needed in application, and if it is so desired, it has to be used. The bottom line should be, that my, the error induced due to inaccuracy of the representation of the curve geometry, should not be greater, should not be significant as compared to the error that I am enquiring due to the approximation of the exact solution.

(Refer Slide Time: 53:40)



So, this should be bottom line with which we have to choose the representation. A place where this naturally comes, this isoparametric representation is what is normally used; this comes in when we do the non-liner elasticity problem, the finite elasticity problem using what is known as the updated Lagrangian; those things we are not going to discuss here, but in the case of finite elasticity problems, isoparametric formulation is something that becomes almost mandatory. Naturally, it has to be used, because of the very way, we are defining deformations there.

In the next class, what we are going to do is, we are going to deal with, now, a different set of problems, just like we did in the one-dimensional case. We first went with the bar problems, then we went to the beam problem. So, next class, we are going to do the classical plate problem, which again, is just like the beam, the Euler-Bernoulli beam. This is an extension of the Euler-Bernoulli beam theory, it will lead to a 4th order linear differential equation; and because of it being 4th order we will have to talk about C 1 elements, C 1 continuous elements. In the case of such problems, how do you construct this C 1, continuous elements? What are the restrictions of such an approximation; those things we will discuss in the next class onwards.