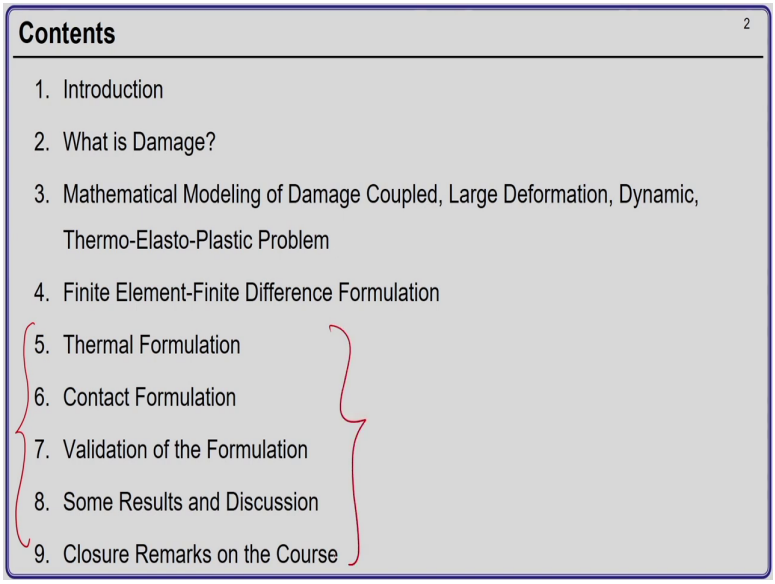
Computational Continuum Mechanics
Dr. Sachin Singh Gautam
Department of Mechanical Engineering
Indian Institute of Technology, Guwahati

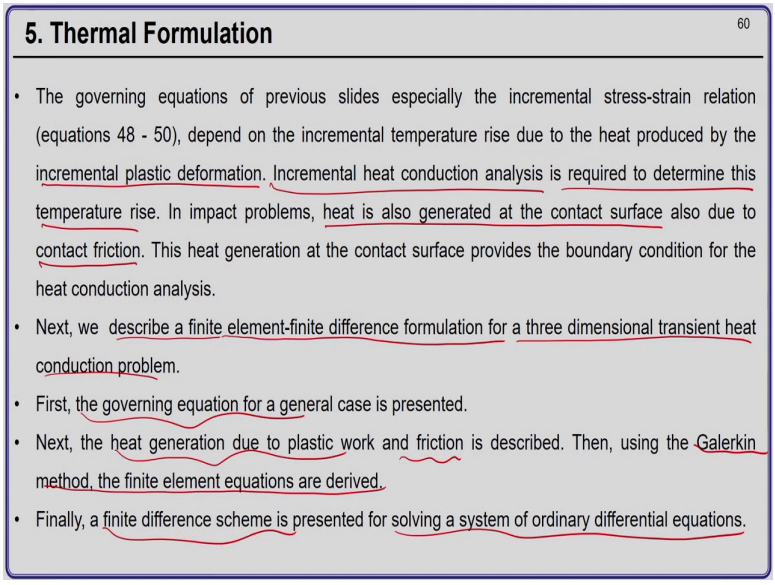
Lecture - 40
Finite Element Formulation of Ductile Fracture in Coupled-Thermo-Elastoplastic Dynamic Contact Problems

So, welcome to this last lecture on Finite Element Formulation of Ductile Fracture in Coupled-Thermo-Elastoplastic Dynamic Contact Problem ok. So, in today's lecture, we will discuss the thermal formulation, the contact formulation, we will see some validation problems ok; finally, some results and then, we will conclude this course by discussing what we have covered and what we have not covered ok.

(Refer Slide Time: 01:13)



(Refer Slide Time: 01:19)



So, in today's lecture, we are going to cover following five topics ok. So, till last lecture, we had covered the finite element, finite difference, discretization ok for coupled-thermo-elastoplastic problem. Now, during that discretization, we had treated temperature to be at time T ok. So, how do we calculate temperature? Ok.

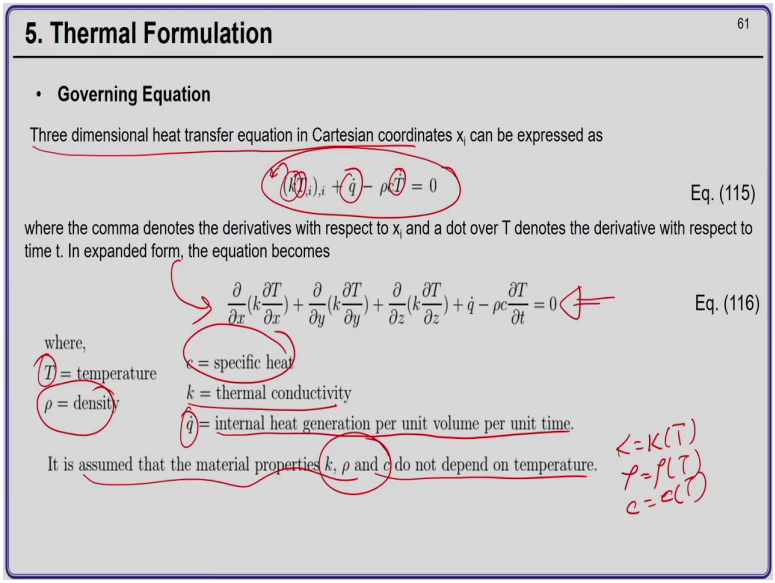
So, for this ok what we have to do is we have to do the incremental heat conduction analysis to determine the temperature rise ok. Now, in impact problem, you will have heat which is generated at the contact surface because of the contact friction ok. So, in elastoplastic problem, you also have heat generated because of the incremental plastic deformation ok.

So, now, you have two different ways in which the heat is generated in the system; one is because of the plastic work which is taking place and then, the other is because of the frictional heat which is generated at the contact surface ok. Now, these two heat generation ok

have to be taken into account ok. So, in this section, we describe a finite element, finite difference formulation for a three-dimensional transient heat conduction problem ok.

So, first we will present the governing equation followed by expression for the heat generation due to plastic work and frictional heat and then, we will use the Galerkin method for finite element discretization and then, we use the finite difference scheme for solving a system of ordinary differential equation, which is of first order ok. So, in previous chapters, we had a second order ordinary differential equation. Now, we will have a first order ODE ok.

(Refer Slide Time: 03:24)



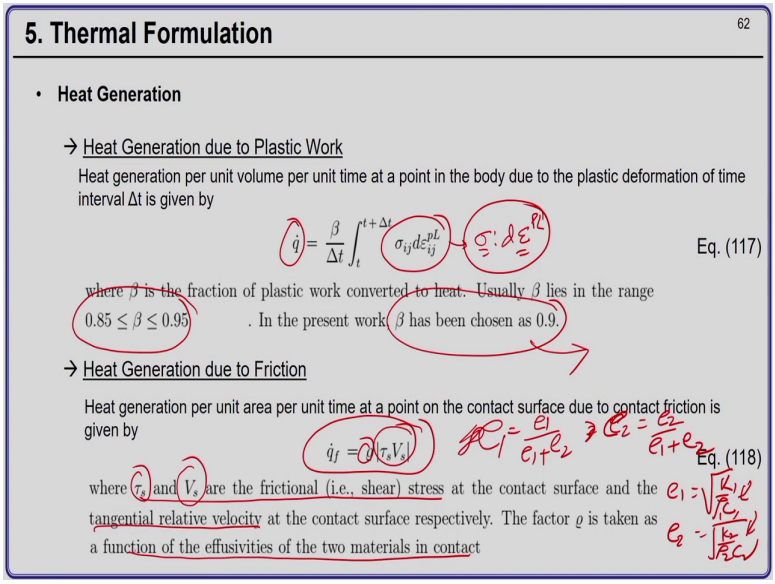
So, the governing equation for a three- dimensional heat transfer equation ok in Cartesian coordinate is given by following equation. So, this called the Fourier's law and here your k is

the thermal conductivity; T is your temperature; q dot is the internal heat generation per unit volume per unit time ok.

This is the heat generation because of the plastic work ok. Rho is the density; c is the specific heat ok and this dot over T denotes derivative with respect to time. So, if you write explicitly, you can write equation 115 in following form ok. Now, we assume that these material properties do not depend on temperature ok for simplicity.

However, they can be taken function of temperature ok, that would be more realistic; but then this adds lot more complication into the analysis. So, right now what we will take? We will take this material properties independent of the temperature ok. So, that means, k can be taken out here ok.

(Refer Slide Time: 04:43)



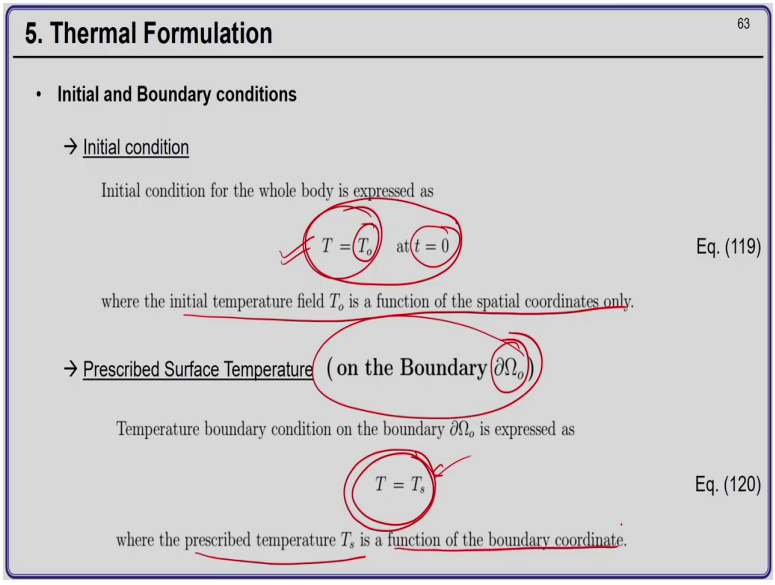
Now, the heat generation term q dot due to the plastic work is given by beta by delta t integral t to t plus delta t sigma contracted with the incremental plastic strain ok. So, this you can write sigma d epsilon PL ok. So, this is the plastic work that is being dissipated and a bulk of it ok, roughly 90 percent of it ok; so, this beta is a factor which says how much of plastic work is getting converted to heat and experimentally people have found that this beta lies in the range of 0.85 to 0.95.

So, here in all the simulation that will show, we will take beta as 0.9 ok. Now, the heat generation due to the friction is taken as q dot f and it depends on the shear stress at the contacting surface and also, the relative tangential velocity V s ok.

So, absolute value, it depend on the absolute value; it does not depend on which direction there is a slip. If there is a slip, there will be heat generated and then, this row ok is taken as a function of the effusively of the two materials in contact ok. So, this is not rho; I mean let us say this factor ok is e upon e 1 ok e 1 upon e 1 plus e 2 ok for first body.

And e 2 equal to I mean rho 2 equal to e 2 upon e 1 plus e 2 and e 1 is root over K by rho C K 1 rho 1 by rho C and e 2 is root over K 2 rho 2 by C 2 ok. So, 1 denotes body 1; 2 denotes body 2 ok. So, part of the heat which is generated at the contact surface will go inside the one body; the part of the heat which is generated if assume will completely go inside the other body ok. There is no heat which is getting dissipated to the surrounding environment ok, that is what we assume ok.

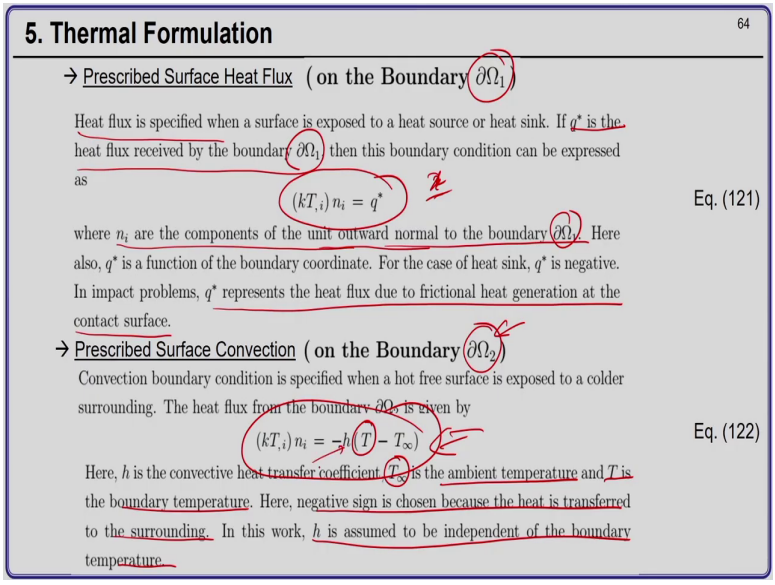
(Refer Slide Time: 07:08)



Now, to complete the given differential equation ok, we need what is called the initial condition and the boundary condition ok. So, the initial condition is specified by specifying the initial temperature T 0 ok of the body ok as a function of spatial coordinates.

So, at time T equal to T 0; the initial temperature field is T0 and then, on the boundary ok, we have some temperature boundary condition specified ok. So, certain part of the boundary may have temperature which is fixed ok. So, let that boundary be denoted by del omega 0 and at those boundary's temperature will always remain T s ok; where, T s stands for the prescribed temperature and it has the function of the boundary coordinate ok.

(Refer Slide Time: 08:03)



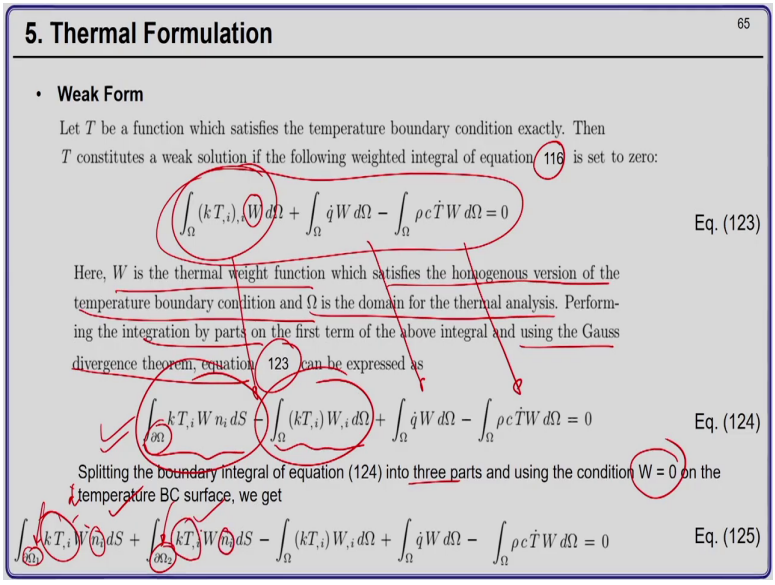
Now, at the certain part of the boundary del omega 1 there will be heat which is coming inside the body ok. So, heat flux is q star is received on the boundary del omega 1 and this is specified by following condition ok. k del t by del x ok into n i equal to q star. Here x, I mean spatial coordinate x ok and then, n i is the component of the unit outward normal vector on the boundary del omega 1 ok.

So, this q star represent the heat flux due to frictional heat generation at the contact surface. So, that is how we will take into account the frictional heat ok. And also, if your problem is such that convection also becomes necessary to take, then we also have the convection boundary condition given by equation 122 ok at boundary del omega 2 ok.

Here, we assume that h is a convective heat transfer coefficient and it is independent of the temperature and T infinity is the ambient temperature which always remains constant and T is

the boundary temperature ok and this negative sign is chosen because heat is transferred to the surrounding. We assume that T will be greater than T infinity therefore, the heat goes outside the boundary ok.

(Refer Slide Time: 09:43)



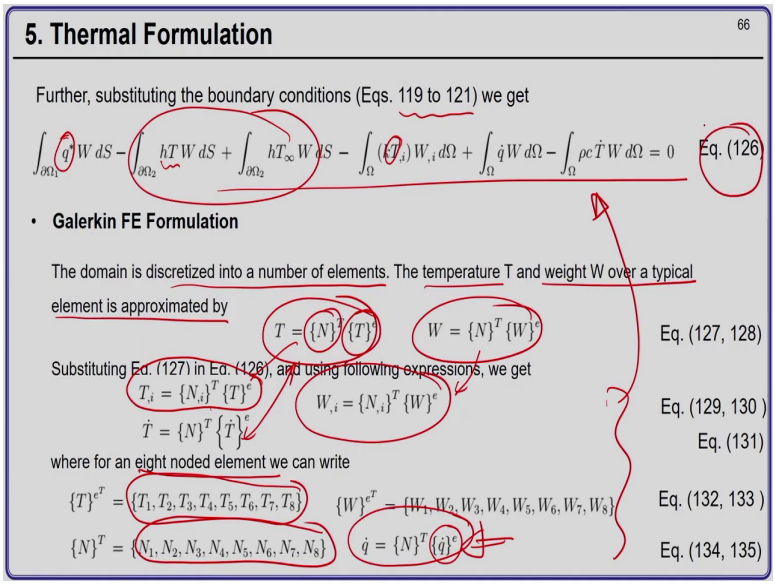
Now, we develop the weak form first and for weak form, what we do? We multiply equation 116 by a weight W ok. So, there is multiply by weight W and integrate over the entire domain of the problem omega ok. So, W is the thermal weight function and it satisfies the homogeneous version of the temperature boundary condition and omega is the domain for the thermal analysis ok.

And now, you do; here you do integration by parts ok and then, you use the gauss divergence theorem. So, equation 123 can now be written in the following form and then, and this is so it will this is component to two terms; one is here and the other is here ok and these two terms

remain as it is ok. Now, you can split this boundary integral into three parts; one which is the boundary del omega 1; other is del omega 2.

So, this is where your heat flux is there, this is where your convection boundary condition is there ok and one where your temperatures are specified; so, where temperature are specified, you will have W equal to 0 at the temperature boundary condition. So, instead of 3, you will finally have 2 surface integrals.

(Refer Slide Time: 11:08)

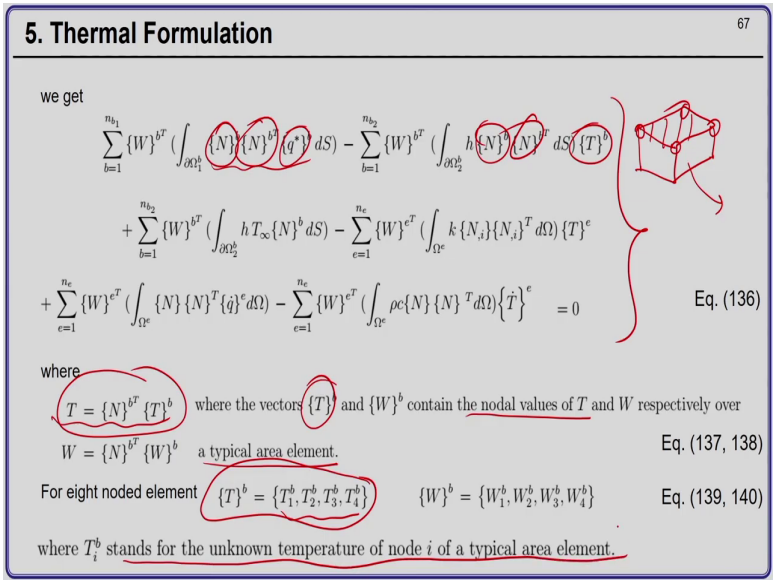


And then, if you substitute from equation 119 and 121, we will get on del omega 1, we will have this term equal to k T comma i; n i will be q star and k T comma i ni will be h T minus T infinity ok. So, this is the second term ok; and then, this is your weak form ok.

So, you have reduced the continuity requirement on the approximation of T by one order and then, what we can now do is we can discretize the domain into a number of finite element and the temperature T and the weight W over a typical element can be approximated using equation 127 and 128 ok; where, n is the shape functions corresponding to the element that you have chosen and T corresponds to the temperature at the elemental nodes ok.

Now, if we choose an eight noded brick element, then you will have 8 temperatures at 8 nodes and you have 8 shape functions ok. Therefore, you can write the derivative of temperature with respect to spatial coordinate x like this and the time derivative of T can be written like equation131 and the gradient of W is obtained like this ok. Now, the heat generation ok per unit volume per unit time can be written in terms of the heat generation at the nodes ok, by usual this kind of discretization ok.

(Refer Slide Time: 13:06)

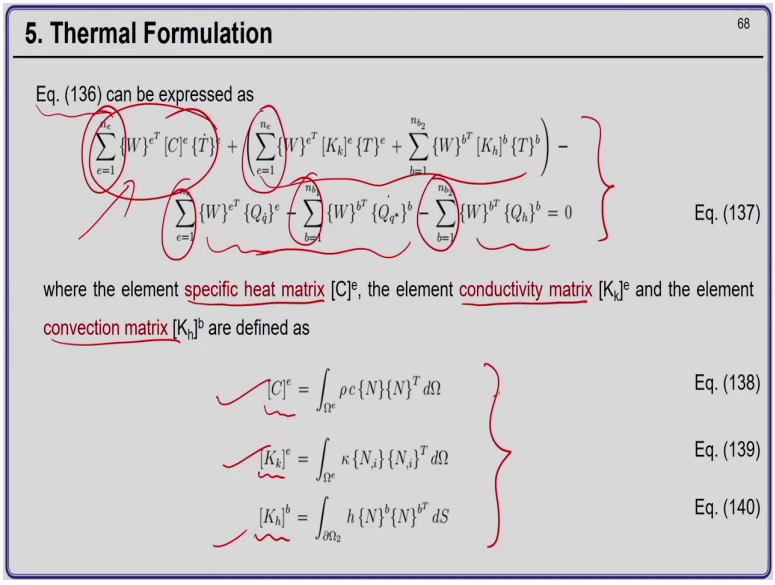


And once you substitute all these approximations ok; so, if we substitute all these approximation here ok, so equation 127 to 135 if you substitute in equation 126, then you will get following form ok. So, once you have this following form, here you have we have used matrix vector notation ok.

So, where we have used the boundary ok, so at the boundary we have to use T equal to N b transpose T b; where, T b contains the nodal values of T over a typical area element ok. So, remember say for a 8 noded element which lies on the boundary, so at the surface; so, if this is the bulk on the surface, you will have only 4 nodes ok.

So, therefore, the boundary temperature will be only consisting of these 4 nodes ok. So, that what you have to see here ok; so, all these here represent the boundary ok. Here, T i b stands for the unknown temperature of node i on a typical area element ok.

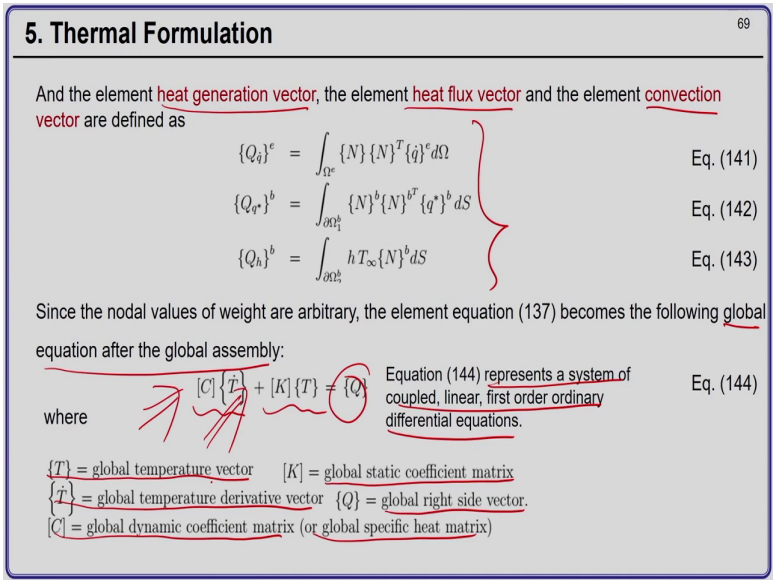
(Refer Slide Time: 14:32)



So, now I can further simplify equation 136 by following these notations given in equation 138 to 140 ok. So, here C, K k and K h are called the specific heat matrix, elemental conductivity matrix and elemental convection matrix and if you use this in equation 136, you will get equation 137 ok.

So, you will get summation over all the elements, the specific heat matrix term ok, you have the (Refer Time: 15:14) matrix term and you will have the convection matrix ok, you have the heat generation matrix ok like this.

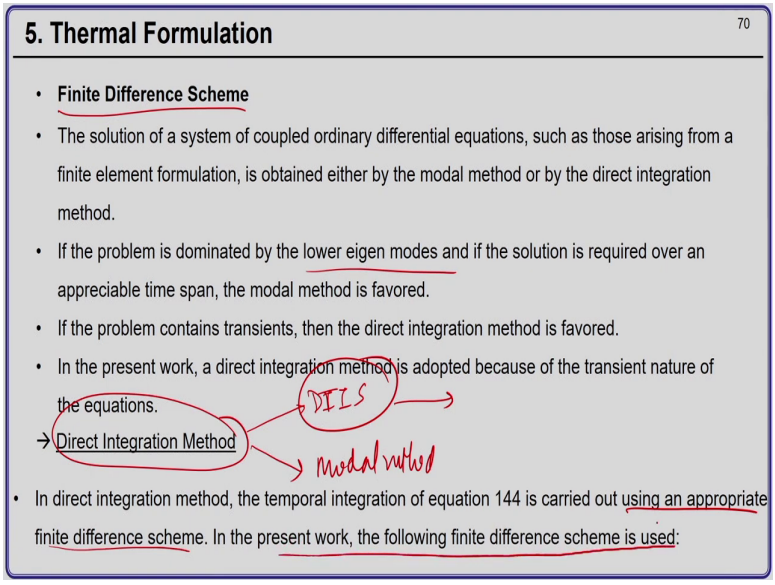
(Refer Slide Time: 15:26)



And then finally, the heat generation vector, the heat flux vector and the elemental convection vector are defined using equation 141 to 143 ok. Now, the only job is you assemble these elemental quantities over all the surface and the bulk elements and you get what is called the global equation after the assembly as C T dot plus KT equal to Q ok. So, equation 144 represent a system of coupled linear first order ordinary differential equation ok.

So, T here is a global temperature vector, T dot is the global temperature derivative vector, C is a global dynamic coefficient matrix or the global specific heat matrix, K is the global static coefficient matrix and Q is the global right hand side vector ok. Now, this is a first order linear ordinary differential equation and to solve this, what we have to do is we have to discretize this time derivative of temperature ok.

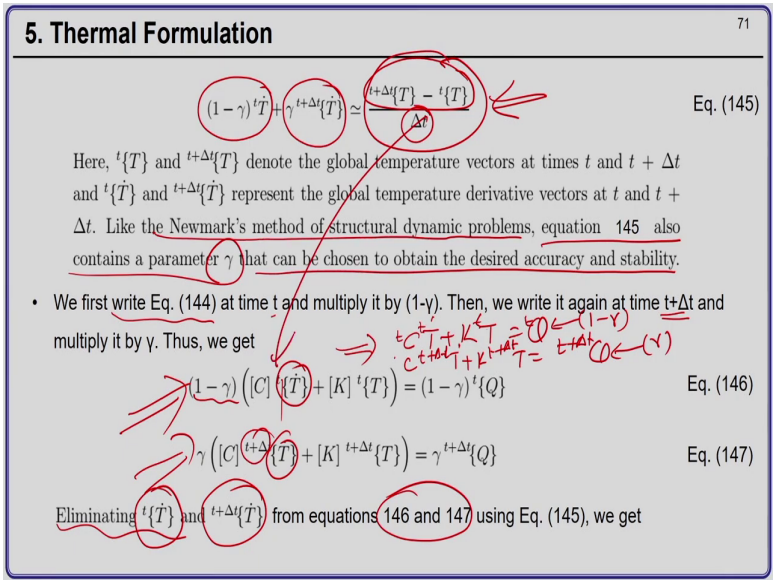
(Refer Slide Time: 16:41)



For this, you can use again the finite difference scheme ok and here, what we do we use what is called the direct integration method ok. So, remember time integration are of two types.

So, there are direct time integration, direct time integration schemes and there are modal methods ok. So, modal methods are usually favoured when you have lower eigen modes and direct time integration scheme works for all cases. So, we use direct time integration and for this, we have to use an appropriate finite difference scheme.

(Refer Slide Time: 17:28)



So, in this particular discussion we use the following finite difference scheme. So, 1 minus gamma times the rate of temperature at time T plus gamma times the rate of temperature at time t plus delta t is roughly equal to the difference of the temperature at t plus delta t and time T divided by delta t ok.

So, that is the approximation that we use ok. And then like the Newmark's method for structural dynamics equation 145 also contains a parameter gamma that can be chosen to obtain a desired accuracy and stability ok; and at the end of this particular section we will see, what are different values of gamma which are commonly chosen ok.

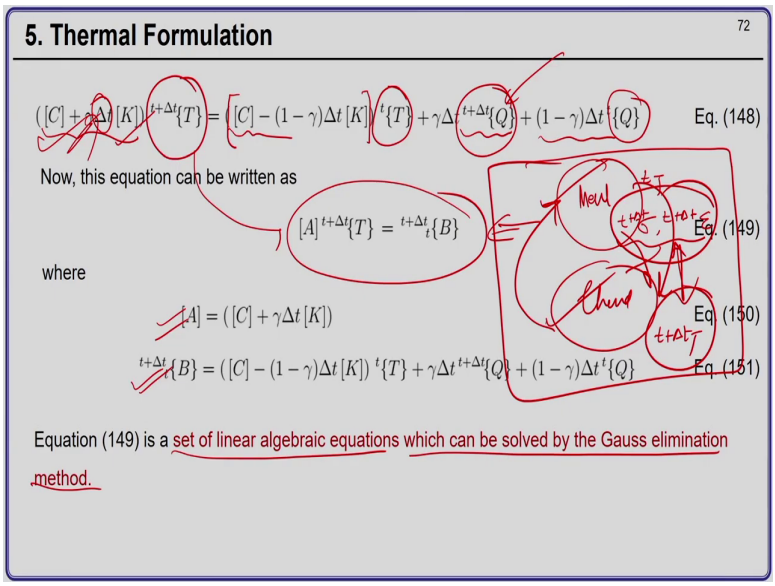
So, now what we do is we write equation 144 at time t ok. So, our equation 144 over C dot plus K T equal to Q. So, we first write this equation at time t ok; I can write this as time t and

then, I can also write this as time t plus delta t ok; I can write t plus delta t T dot plus K t plus delta t T equal to t plus delta t Q.

Now, I have two equations; I multiply this equation by 1 minus gamma. I multiply this by gamma and I simply add these two. So, this is the first equation, this is the second equation and I just then eliminate the rate of temperature T dot and t plus delta t T dot from equation 146 and 147 using this equation over here ok.

So, if I add these 2 terms over here, you see I will have 1 minus gamma into t plus delta t t T dot and plus gamma times t delta t plus delta t T dot ok and this is nothing but this term over here, then that I can substitute.

(Refer Slide Time: 19:28)



If I do this finally, I will get c plus gamma delta t K into the vector of temperature at time t plus delta t equal to C minus 1 minus gamma delta t K ok into the temperature at time T plus gamma delta t into the vector Q at t plus delta t plus 1 minus gamma into delta t the vector Q at time t ok.

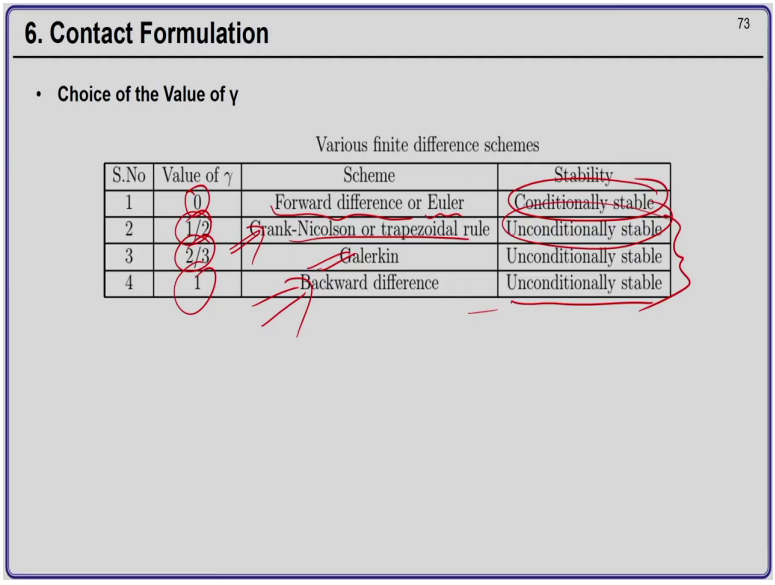
So, I basically will know my heat generation at t plus delta t because at that point, I would have computed my displacements, I would have computed my stresses in the plastic strain and using those updated values, I can compute t plus delta t Q and then, I can find out C, K.

I know my delta t, I can set my gamma and then, I can and I also know the temperature at the previous time step. I can directly compute my t plus delta t using following equation 149 ok; where, A is given by this and B is given by this. So, this is a set of linear algebraic equation, which can be solved by Gauss elimination method ok.

Now, here what we have assumed? We have first solved the mechanical problem using a particular temperature once and at the end we have got our stresses at time t plus delta t and plastic strains ok at time t plus delta t and using this, I can now solve my thermal problem ok and then, I can get my temperature t plus delta t; and then, I have to iterate between these two till I get a final convergence when both of these are not changing.

If my time step is very small, then I can do within 1 Newton-Raphson iteration, I can one solve this and one solve this and that should be enough. But if the time step is very large, then I have to do the iteration between these two mechanical and thermal problems ok.

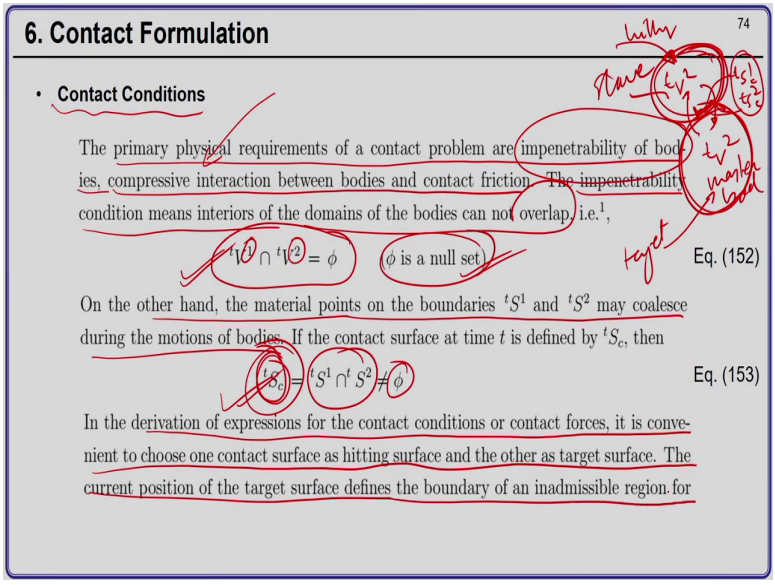
(Refer Slide Time: 21:37)



So, now there are different values of gamma that one can choose. If you choose gamma equal to 0, you get what is called the Forward difference or the Euler method and this is an explicit method which is only conditionally stable. For gamma 1 by 2, you get what is called the Crank-Nicolson or the trapezoidal rule and this is unconditionally stable.

Gamma 2 by 3 gives you the Galerkin method and gamma 1 gives you the Backward difference method. The last three are all unconditionally stable, while the first one gamma equal to 0 is the conditionally stable; it depends on delta t that you choose ok.

(Refer Slide Time: 22:13)



So, once we have completed our thermal formulation, our final job is to do the contact formulation. Now, the all the derivation that you have done till now, it is for two body contact problem and what happens at the contact surface, we neither know the displacement and we neither know the forces.

So, what happens? We have more number of unknowns than the number of equation. So, we have to generate some extra set of equations and these extra set of equations will be generated using the contact formulation ok. So, in contact rather than discussing in detail about the contact formulation, what we will do is we will just look into some of the essential features of contact ok; and we will discuss the Lagrange multiplier method in a brief sense ok.

Because contact itself is a very big subject which requires itself a whole course on contact formulation which I am not doing here. So, but I will give you the basic idea ok. So, now the

first we have to write what is the contact conditions ok. So, the primary physical requirements of a contact problem are impenetrability of bodies, compressive interaction between the bodies and contact friction ok.

So, if you are at the macro scale and you are not at a very small scale where the adhesion comes into picture, in that case if you are solving the contact problem, non-adhesive contact problem; then, the interaction between the bodies can only happen if the tractions between the bodies are compressive in nature ok.

If you go to adhesive contact problem, then there can be tensile tractions also ok. Now, this is one important thing. The other is impenetrability, which means that during the entire course of simulation or the contact, one body cannot penetrate the other body that is physically not possible ok. So, the impenetrability condition means that the interiors of the domains of the two bodies cannot overlap ok.

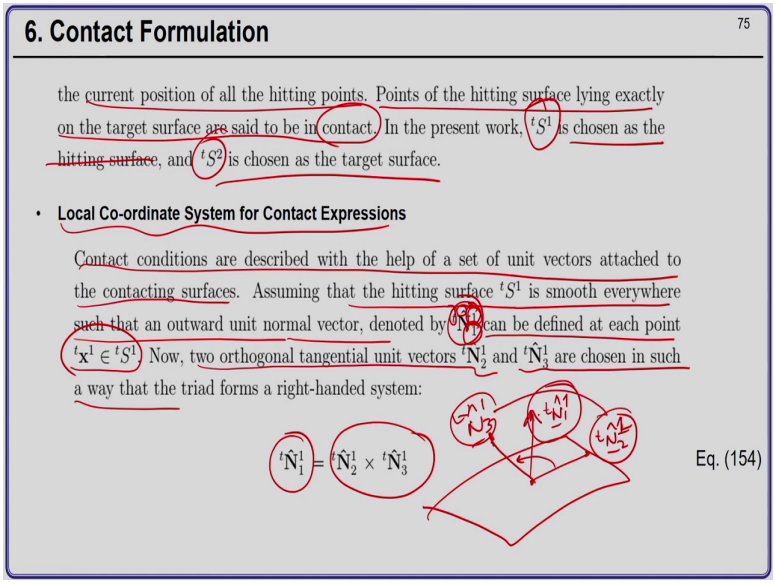
So, if you have one body over here and you have the another body over here ok and this is volume 1 and this is body 2 ok. So, this 1 denotes body 1; 2 denotes body 2; so, this body cannot penetrate this body and this body cannot penetrate this body. So, mathematically, you can write the intersection of the volume for the two body is a null set ok.

So, this is the condition that you have to satisfy. Also, on the other hand, the material points on the boundaries may coalesce during the motion of the bodies ok. So, during contact this part of the boundary say ok, so, the top is t S 1 and this is t S 2 ok. So, now the two bodies can have a common contact surface called S c ok. So, the intersection of the surface of two bodies ok; so, this is the ok; so, the cc.

So, this is the entire surface of body 1; this is the entire surface of body 2; and if you take the intersection of these two, you will get what is called the contact surface and this is not null set. So, the volumes intersection of the volume gives you the null set; while the intersection of the surface gives you the contact surface ok.

Now, in the derivation of the expression for the contact condition or contact forces, it is convenient to choose one contact body as hitting surface, while the other at the target surface ok. When you do contact analysis, one body is always chosen as the hitting body and the other body is chosen as the target body ok. So, this is also called the master body and this is the slave body ok.

(Refer Slide Time: 26:32)



And the current position of the target surface defines the boundary of an in admissible region for the current position of all hitting points ok. So, the points on the hitting surface lying exactly on the target surface are said to be in contact ok. So, we can choose body 1 as the hitting surface and body 2 as the target surface ok.

Now, before we proceed with the contact simulation, we have to define what is called the local coordinate system ok, for the derivation of the contact expression ok. So, contact

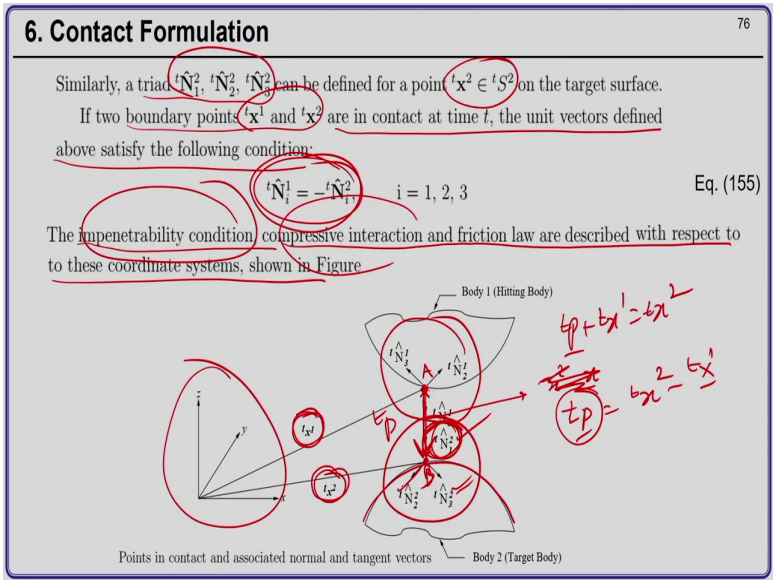
conditions are described with the help of set of unit vectors attached to the contacting surface ok. So, at each contacting point, you have to define a set of unit vectors ok which are normal and tangential to that point ok.

So, assuming that the hitting surface is smooth everywhere such an such that an outward normal denoted by N 1 1 ok. So, this N hat 1 1; so, this 1 shows body 1 and the so superscript 1 shows body 1 and super subscript 1 shows the direction. So, the one direction is always taken as the normal direction N hat; so, hat shows it is a unit vector and t denotes here that it is at time t.

It is calculated based on the configuration of the body S 1 at time t ok and this is defined at each point x 1 which belongs to body surface body 1 ok. Now, two in orthogonal tangential vectors N 2 and N3 are chosen such that the cross product of N 2 and N 3 ok gives you the normal ok.

So, at the contact surface ok, you can define ok what is called the normal like this N2 tangential vectors like this ok; so, N 2 cross N 3 gives you the normal vector N 1 ok. This is said by the right hand rule ok. So, if you would curl your fingers from this to this, then the direction of the thumb will give you the normal ok. So, this is the convention that we follow.

(Refer Slide Time: 29:01)



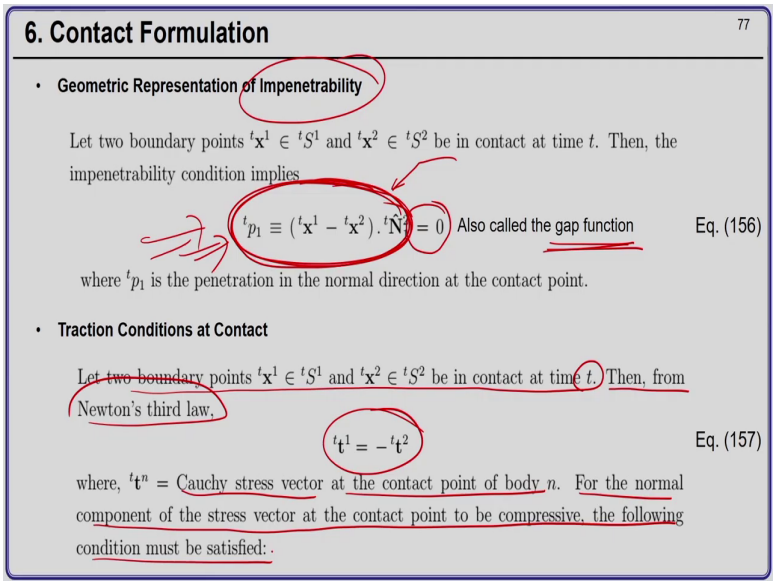
Therefore, similarly a triad of vectors normal vectors N 1 2, N 2 2 and N 3 2 can be defined for the corresponding point x 2 on target surface ok. Now, if the two boundary points x 1 and x 2 are in contact, then in unit vectors ok satisfy the following condition. So, they are equal, but opposite in sign ok.

Now, the impenetrability condition and the compressive interaction and the Friction law are described with respect to these coordinate systems ok. Remember all these impenetrability condition that mathematically will derive the compressive traction ok, everything have to be described with respect to these coordinate system ok.

So, normally you will have those traction with respect to the global coordinate system, but then you have to transform those vectors to the local coordinate system defined by these normal and tangential vectors ok. So, you can use the concept, where they discuss in our

discussion on tensors where how to transform one tensor from one coordinate system to another coordinate system ok.

(Refer Slide Time: 30:25)



Now, mathematically you can express the impenetrability condition using what is called the gap function ok. So, now, this is your target surface ok, target body and this is the target surface and let the point which comes in contact on the target surface have the coordinate x two and the point on the hitting body which comes in contact with the target point, let its coordinate be x 1 ok.

So, then this vector over here will be nothing but x 2 minus ok. This from A to B ok, let us say this is p. So, t p plus t x 1 will be t x 2 ok. So, tp equal to t x 2 minus t x 1 and then, if you take a dot product with the normal in this particular direction, you will get the penetration ok.

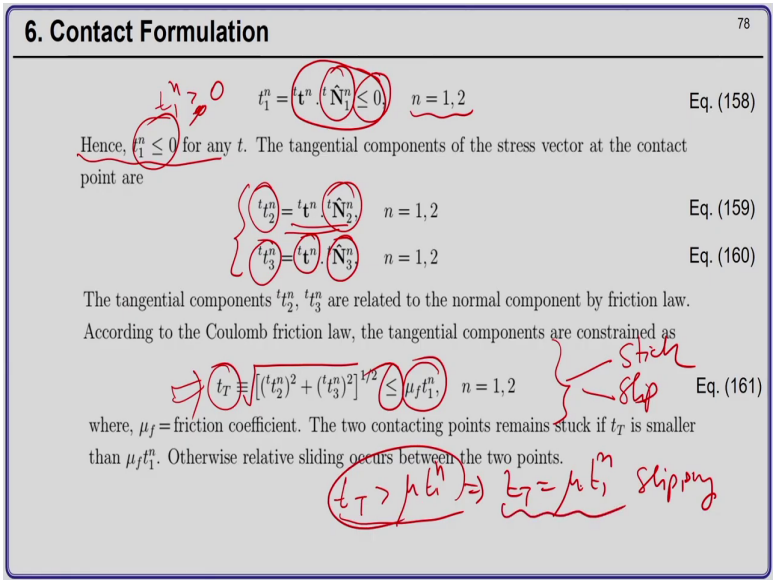
So, this is your normal and if you take the dot product of this penetration with respect to that normal, you will get the component along the normal direction and this is called the gap function ok. So, this is the penetration that can happen and from the impenetrability criteria, this penetration always has to be equal to 0 ok.

So, when you are writing the code, you always have to ensure that this gap function turns out to be 0 and this gap function for every point A, it is used to find out the corresponding point B on the target surface on which point A will come in contact with ok. So, that is where the gap function is usually used. That dot problem itself is a non-linear problem which is solved locally at each contacting point using Newton-Raphson iteration ok.

Now, in the tangential direction; so, this was along the normal direction, now what happens in the let us say in the tangential direction ok. So, if this point is contact within this point, then depending on the tangential traction, the body may slip or the body may stick ok.

So, if the two boundary points are in contact at time t, then from the Newton's third law you know that the traction ok, the contact traction at the contact point are equal and opposite ok. So, for normal component of the stress vector at the contact point to be comprehensive, the following conditions have to be satisfied ok.

(Refer Slide Time: 33:13)



So, if the contact traction have to be compressive, then the normal tangential vector dotted with the in the normal vector on body n ok should be less than equal to 0 and hence, for all time t, if the traction is less than equal to 0, the two points will remain in contact.

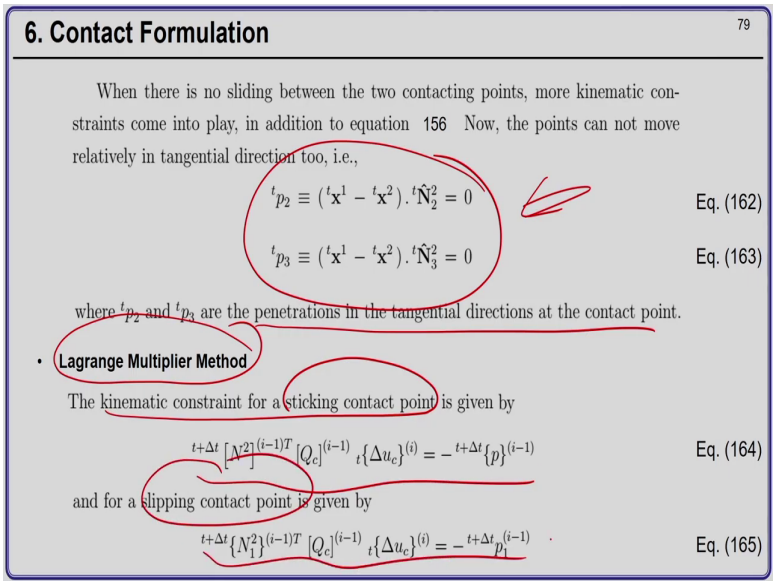
If this traction t 1 n becomes greater than 0, then we say that the body has gone out of contact and in the tangential direction you can find out respectively the tangential tractions t 2 and t 3 the by taking the dot product of the traction t n with the tangential vectors N 2 and N 3 ok.

And then, if the resulting traction ok; so, this is the resulting traction root over t 2 square plus t 3 square and that gives you the tangential traction and if this tangential traction is less than mu times t mu f times t normal traction, then you have what is called the sticking condition, then the two bodies cannot move relative to each other. However, if t T becomes greater than

mu times t 1 n, then t T should be equal to mu t n and the body is set to be in the slipping condition ok.

So, the in the tangential traction, you again have two ways, I mean two conditions either the body may stick, if this is less than equal less than equal to mu times t or it may slip. In that case, t T is greater than this. Then, t T has to be set equal to mu t and during the numerical simulation ok.

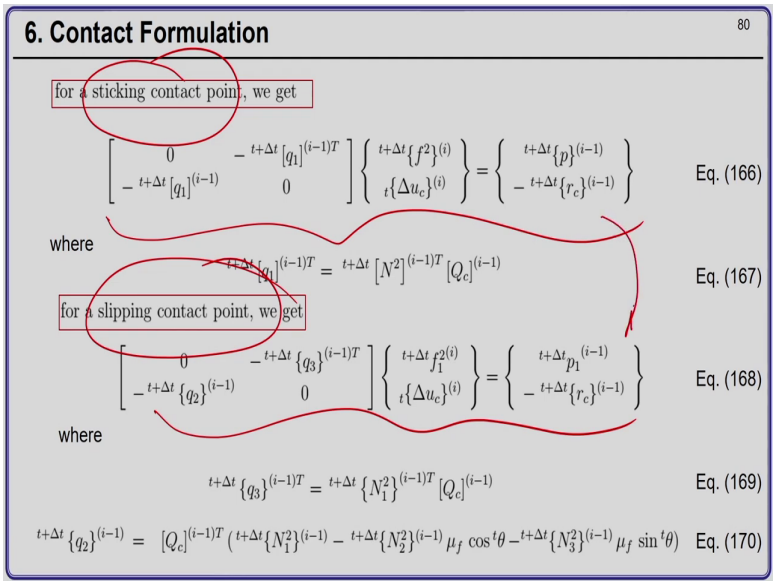
(Refer Slide Time: 34:59)



So, these two conditions can be written in the following penetrations in the tangential direction in the following equation form and then, once we have set up the penetrations and the impenetrability criteria, then we can use one of the constraint handling techniques available in contact mechanics ok, computational contact mechanics.

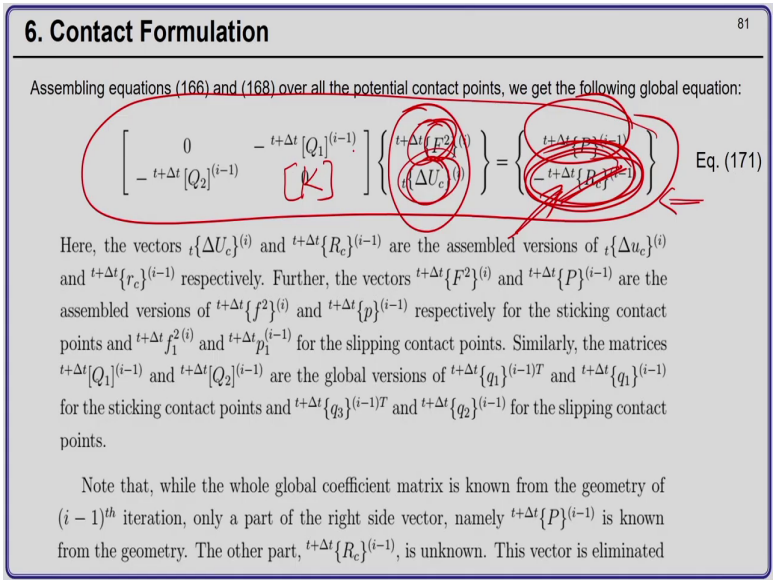
There are Penalty method, Augmented Lagrangian multiplier method, Lagrange multiplier method. So, we use what is called the Lagrange multiplier method and for that, the kinematic constraint for a contact point is given by following equation and for a slipping point will have the following equation ok.

(Refer Slide Time: 35:45)



And these when we assemble ok for all these sticking contact point, we will get this final form ok. So, this is a very long derivation. So, I am just giving you the final expression ok and for a slipping contact point will have this particular form ok.

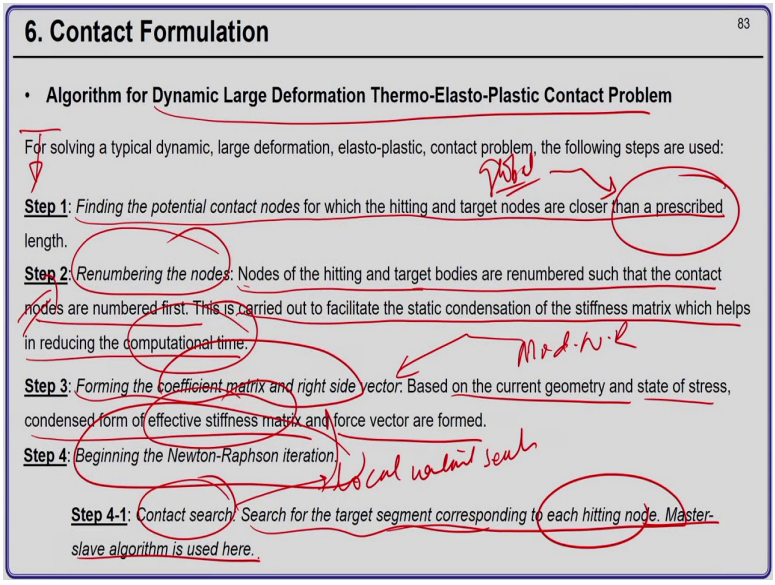
(Refer Slide Time: 36:08)



So, you can assemble these globally and you can get the following global form ok and here, you notice that we do not know part of the right hand side vector and also, we do not know the part of the I mean this is obviously the contact displacement and the contact reactions are not known also; we just know p from the geometric consideration ok.

So, what we do is we add this equation with the equation that we get from the Newton-Raphson iteration to eliminate the reactions corresponding to the contact node. So, these are global unbalanced force vector corresponding to the contact nodes and this when we add with the Newton-Raphson iteration, we can eliminate this over here and then, you will get the tangent matrix K of two bodies here. In that case, you can solve for the unknown contact displacement and the contact forces ok.

(Refer Slide Time: 37:18)



So, what is the algorithm for dynamic large deformation thermal elastoplastic contact problem? Ok. So, the first step is, find the potential contact nodes for which the hitting and target nodes are closer than a prescribed length ok.

So, when you start a simulation, one body is already a master's body; the other is a target body and then within a prescribed length from the master's body, you check master surface you check whether there are some nodes of the hitting body that are coming in contact. And then, what you do?

You can and this step is optional; but you can do the renumbering of the nodes, nodes of the hitting and target bodies are renumbered such that the contact nodes are numbered first and

this is carried out to facilitate the static condensation of the stiffness matrix, that we discuss at the end of the finite element discretization to in reducing the computational time ok.

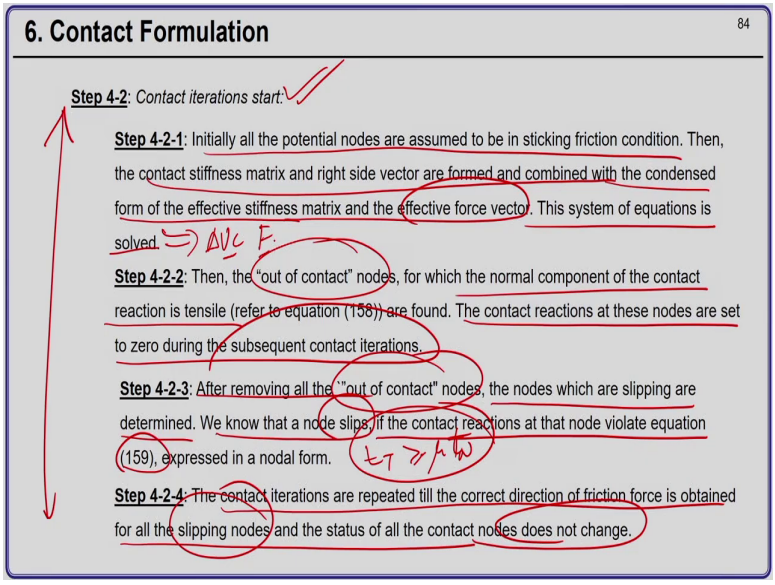
Now, we formed the coefficient matrix and the right hand side force vector based on the current geometry state of stress and the condensed form of the effective stiffness matrix and force vector are formed and then, we form the Newton-Raphson iteration. So, this here corresponds to the modified Newton-Raphson scheme because the effective stiffness matrix is formed outside the Newton-Raphson iteration ok.

First, what we have to do once the Newton-Raphson iteration start? we do contact search ok. So, this step is called the global contact search ok. So, here we are more interested in whether a node is coming in contact or not. We are not interested where it comes in contact, we are just interested whether it can come in contact or not in the current load step ok.

Remember this is for one particular load step ok; one particular delta t, from t to t plus delta t this is what we are discussing. So, now, you do what is called the local contact search ok.

You search for the target segment corresponding to each hitting node. Now, I am more interested in where does my hitting node exactly come in contact with the target segment, which target segment and which location ok. So, we can use what is called the master slave algorithm and then, we start the contact iterations ok.

(Refer Slide Time: 39:34)



So, we initially we assume that all contact nodes are in a sticking friction condition; then, the contact stiffness matrix in the right hand side vector are formed and combined with the condensed form of the effective stiffness matrix and effective force vector and this system of equation is then solved.

Then, we find out if there are some out of contact node for which the normal component of the contact reaction is tensile. This may happen as you update the contact reaction some nodes may go out of contact and some nodes may come in contact ok. So, the contact reaction that this node are set to 0 during the subsequent contact iteration.

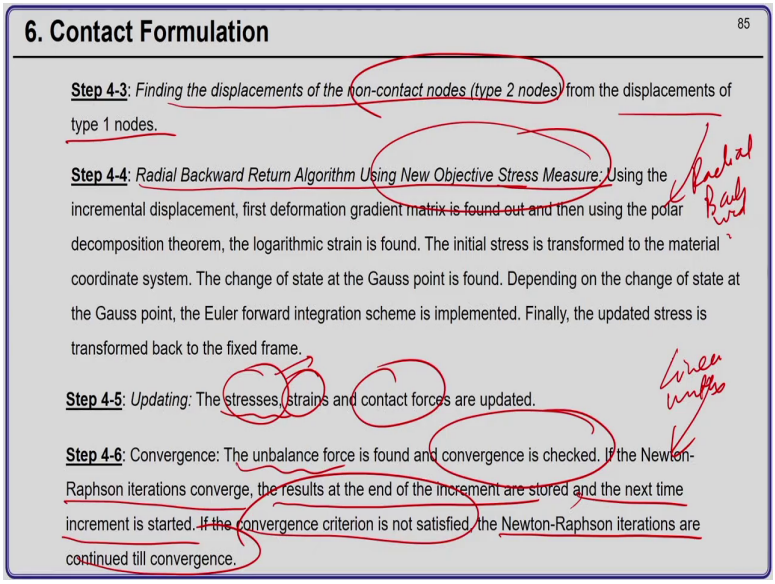
Then, we remove all the out of contact nodes the nodes which are slipping are determined ok, after we have done this and we know that if the node slips, if the contact reaction at that node

violate equation159 which is the tangential traction is greater than equal to mu times the normal traction t N, then I know that my node is slipping.

So, I determine which are the nodes which are slipping, there I have to set my tangential traction equal to mu times normal traction t N. The contact iterations are repeated till the correct direction of the friction force is obtained for all slipping nodes and the status of all contact nodes does not change.

So, we have to do these iterations till there are no nodes which are going out of contact are not coming in contact and the status of all the sticking nodes or the slipping nodes remain as it is; once this happens, we will say that our contact iterations have converged ok.

(Refer Slide Time: 41:12)



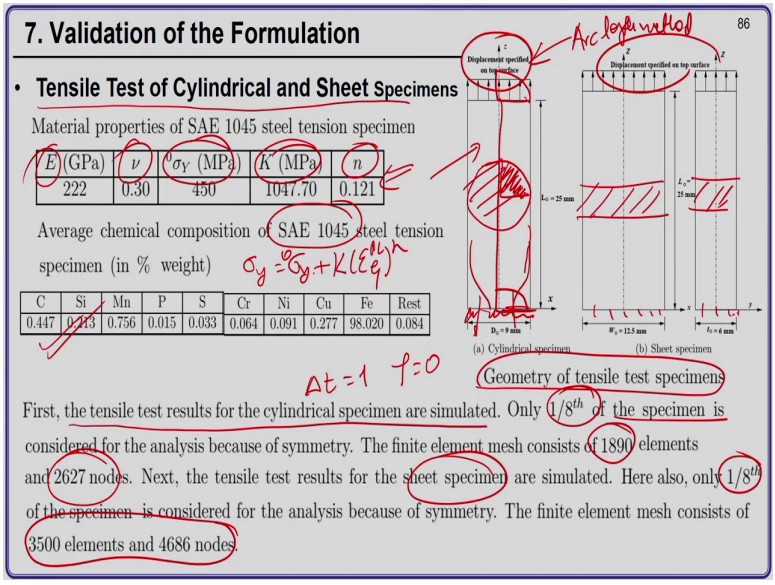
Once we have found out the contact displacement, this is where we have found out the contact displacement and we have found out the contact forces. Once we have found that, we can find out the displacement of non-contact nodes from the displacement of contact nodes which are called the type 2, type 1 node also.

This is done using static condensation scheme and then, we apply what is called the radial backward return algorithm using new objective stress measure ok. So, this we have already discussed. We find out the stresses and from these stresses, we find out the what is called the internal forces ok and then, I can find the unbalance ok.

So, I can do the first updation of stresses strain in the contact forces, I can find out the internal forces corresponding to the stresses and I can find out the unbalanced force vector and I can check for the convergence. If the Newton-Raphson iteration has converged, then we print the results at the end of the increment and we move to the next time increment ok.

And if the convergence criteria is not fulfilled, then the Newton-Raphson iterations are continued till the convergence that is we go back to step 4 and we restart the next Newton-Raphson iteration ok. So, here if your Newton-Raphson method is found to be diverging ok, then we use what is called the line search method ok. Here is what we use radial backward; radial backward radial backward return algorithm for integrating the incremental stress strain relation ok.

(Refer Slide Time: 43:02)



So, now we come to the validation of the formulation that we have discussed. Why is validation needed? Any complex numerical model that you develop ok has to be first tested for any form of error in the modelling or in the implementation step. So, how do you test whether your formulation is working fine?

So, what you do? You take some problems for the literature either experimental or numerical and you run those problems using your code and try to see whether the results obtained from your code and those reported in the literature that is experimental or numerical matches well ok. They may not match exactly, but they have to match pretty well ok; say up to 10 percent error ok.

So, here we have validated a lot of problem we just take two of them; one is the static problem. It is a tensile test of cylindrical and sheet specimen. So, you see here, the geometry

of the tensile stress. So, this is the cylindrical specimen. So, this has a cylindrical cross section ok and this is a sheet specimen.

So, it has a rectangular cross section ok. So, you can see here, this is the side view and this is a cylindrical specimen and what we do? We apply displacement at the top surface and at the bottom, we keep this fixed in the normal direction, it can move in the tangential direction; here also we keep fixed in the normal direction, it cannot go in the Z direction ok.

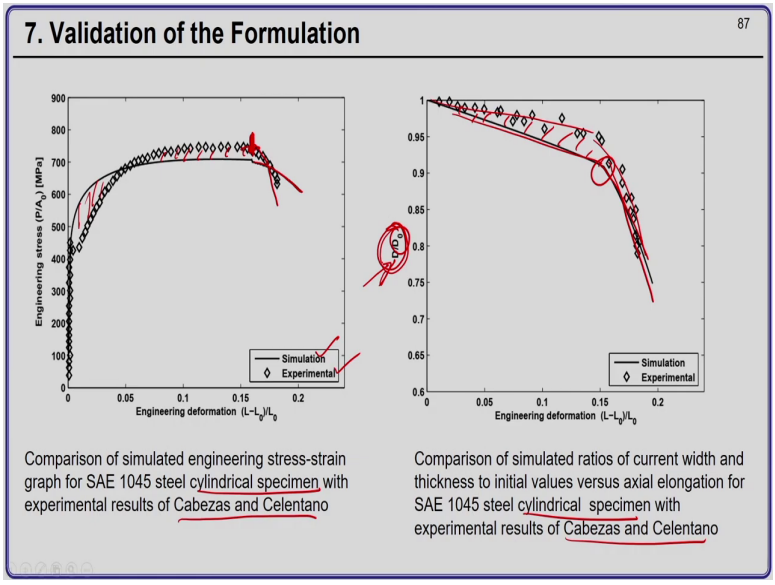
And then, because the there are no external force; therefore, I have to go for what is I have to apply what is called the arc length method. So, here I will use arc length method because I expect that after some time, there will be making which will happen here ok. So, because of the necking the load displacement curve will have a negative tangent and then, I have to be careful because usual Newton-Raphson will not converge ok.

So, these are the material properties ok; so, the Youngs modulus, Poissons ratio, the initial value of the yield stress and the hardening coefficient and hardening exponent ok. So, remember our hardening law is sigma equal to sigma y 0 plus K epsilon equivalent plastic raise to power n ok.

So, that is how my yield value is changing and so, this is the chemical composition of this SAE 1045 steel which we have taken from the literature and we first the tensile test results of cylindrical specimen are simulated. We only take the one-eighth of the specimen ok. So, we just take this. So, this is the half of the specimen. So, we just take the one-eighth ok. So, we just take this much ok.

So, if you can see, it is only this much of specimen is what we take in each case and the number of finite elements are 1890 and these are the number of nodes and then, we do the simulation of the sheet specimen; again, we will use only the one-eighth of the specimen with these many number of elements and nodes and this is of course, a static problem. So, here delta t is always chosen to be 1; rho is taken to be 0; so that there is no mass matrix and the problem turns out to be a static problem.

(Refer Slide Time: 46:36)

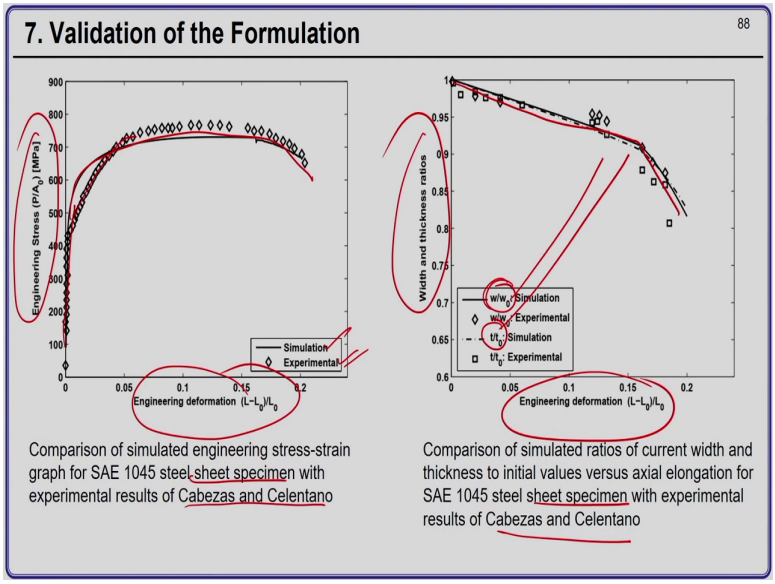


Now, if we have static problem, then the first thing we do is we compare the engineering stress strain curve ok and the ratio of the current diameter of the cylindrical specimen. So, this is for the cylindrical specimen ok; diameter of the cylindrical specimen at current value divided by the its initial diameter and we compare our simulation result with the experimental results of Cabezas and Celentano ok. So, if we do this, you see this is the experimental result and this is what the simulation result that we get ok.

So, this is the experimental result and this is the simulation result that we get and although, there are some errors at certain positions; but then, what we can say that at least our numerical implementation is pretty much ok and you can see there is a, if you can see here there is a necking.

At this step, you will achieve the ultimate stress and the load drops off ok. So, this is where you will observe necking and then, you will have a rapid degradation of the value of the diameter ok.

(Refer Slide Time: 47:54)

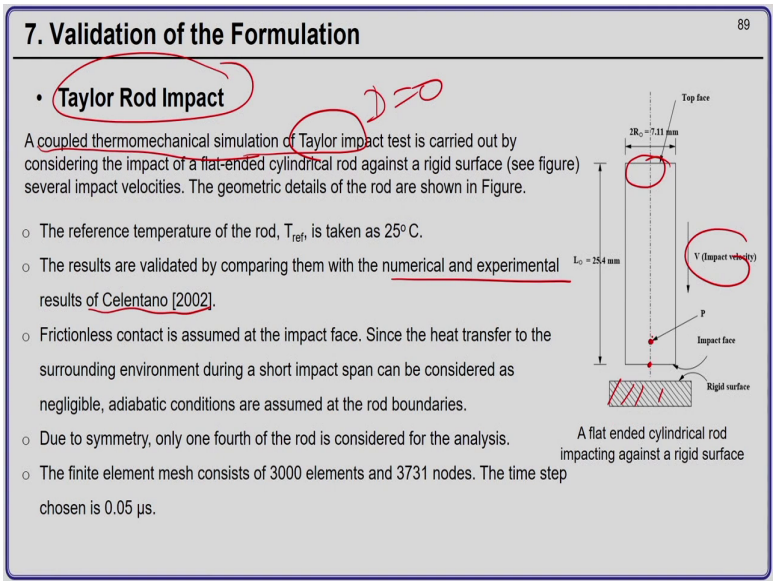


So, this is the result for the sheet specimen. We compare the engineering stress versus engineering deformation and the width and thickness ratios with the engineering deformation with those from simulation and the experimental results of Cabezas and Celentano and again, we see we are very well able to capture the trend which has observed in the numerical simulation ok.

So, here the width and the thickness ok, the current width by original width and current thickness by original thickness matches well with the those results reported by Cabezas and

Celentano ok; so, with this static result, we are sure that at least the static part of the code is working fine.

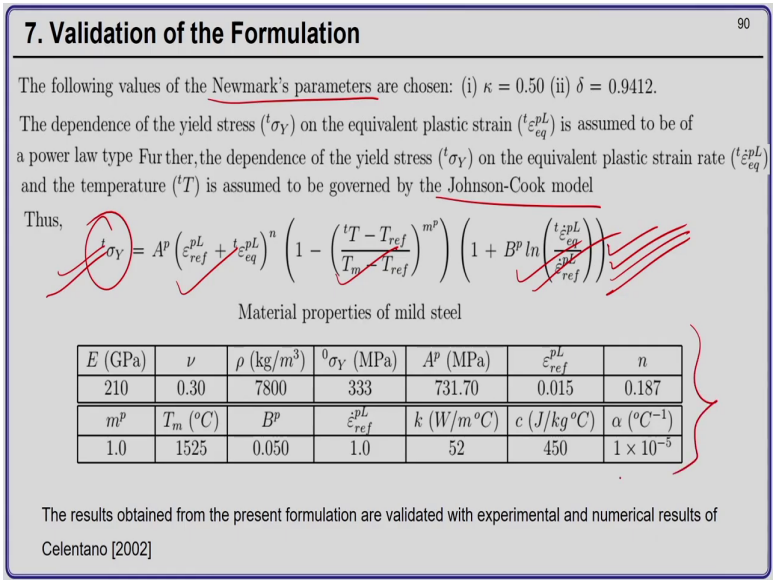
(Refer Slide Time: 48:46)



So, next what we do we try to validate the dynamic part and the dynamic part, we do using the Taylor rod impact tests for which results are already available numerically and we do a coupled thermo mechanical analysis. There is no damage here; damage is equal to 0 ok and we take the Celentano's numerical and experimental results.

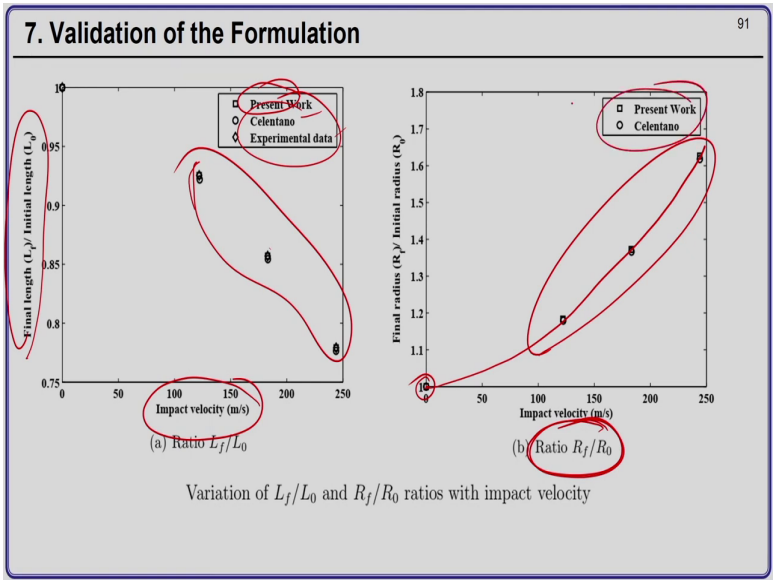
So, this Taylor rod impact problem is you have a cylindrical rod which impacts a rigid surface at a very high impact velocity and we observe the growth of equivalent plastic strain temperature at this impact phase at point p at different time step ok.

(Refer Slide Time: 49:30)



And we use the Newmark's time integration algorithm and the Johnson-Cook model is used to take into account the effect of plastic strain temperature and strain rate on the yield value ok.

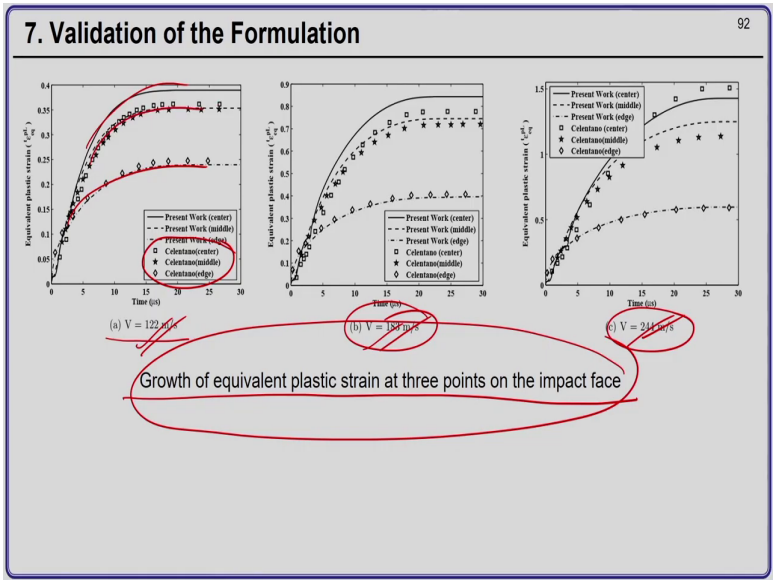
(Refer Slide Time: 49:50)



And these are the material properties and we compare the ratio of the final length to initial length of the rod with the different impact velocity and we compare our work with the experimental and numerical work of Celentano, we see that for 122, 163 and 244 metre per second the our results matches well with the experimental results and the numerical results of Celentano.

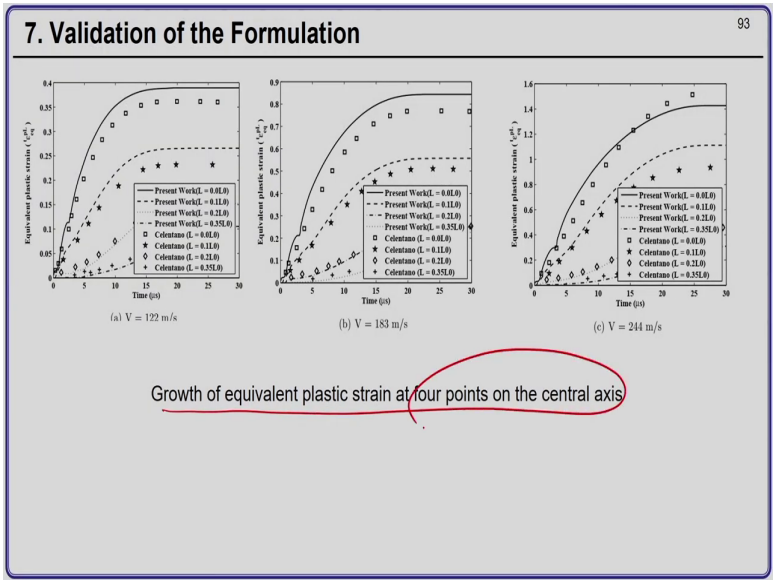
So, we have a very nice result that we have and this is a ratio of the radius at of the rod at the impact phase divided by the initial radius ok. So, at the rod impacts, it will expand and the initial value which is 1 ok. So, initially R f equal to R0. So, you have one and it will expand and then, you will see we have the results matching with the results of Celentano ok.

(Refer Slide Time: 50:46)



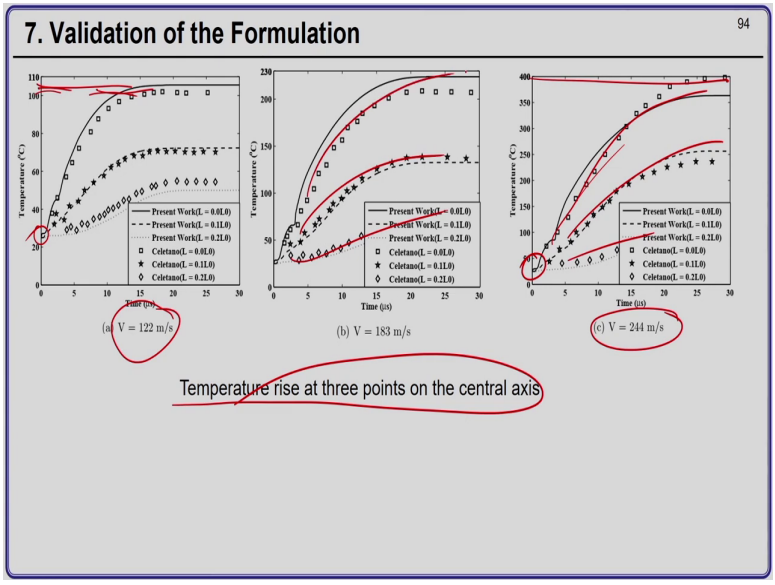
So, these are some of the results for the equivalent plastic strain growth at three points on the impact surface and for three different impact velocities 122, 183 and 244and we see our results which are in solid line and the results of Celentano and we see that our results also match well with the results reported in the literature ok.

(Refer Slide Time: 51:12)



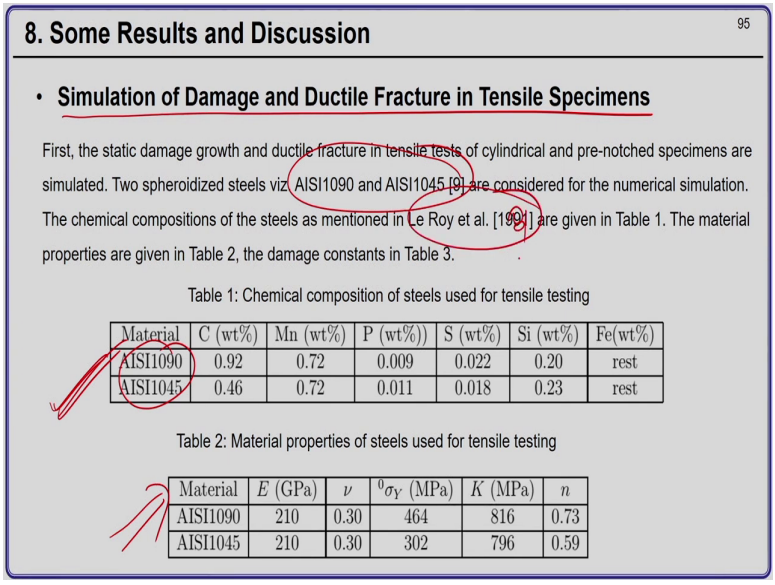
So, these are the growth of plastic strain at four points along the impacts axis and these are temperature rise ok.

(Refer Slide Time: 51:16)



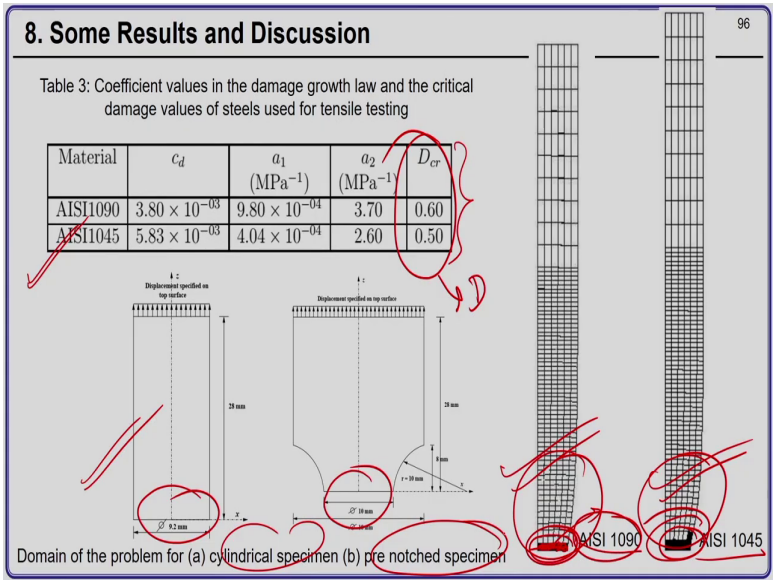
So, the initial temperature is 25 degree and you see that for lower velocities at the raise of the temperature is only around 108 degrees; while for 244, it can go up to 400 degree centigrade ok. From 25 it goes up to that level and for three impact points our results match well with the results reported by Celentano ok.

(Refer Slide Time: 51:45)



So, with this validation, what we can do now is we can discuss some of the results. So, the first thing is we simulate the damage and ductile fracture in tensile specimen. So, again the geometry remains as we use for the validation problem. However, the material now is chosen as AISI1090 and 1045 for which we have the experimental results reported in the literature. So, this is the chemical composition of the two steels and these are the material properties which you have taken from Le Roy's 1981. So, this is 1981 results.

(Refer Slide Time: 52:20)

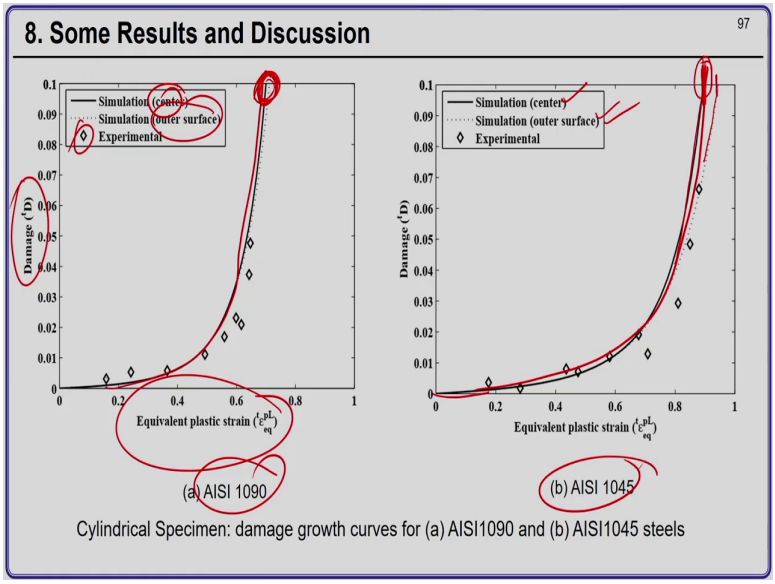


And the damage constants in our damage growth law which we discussed in the initial slides are obtained as follows ok. So, these are used during curve fitting and these again, we use this cylindrical specimen and the pre notched specimen. So, this is the pre notched specimens cylindrical in nature and this is completely cylindrical and for cylindrical specimen, this is the result for 1090 and 1045.

You can clearly see there is a necking involved and these black coloured elements here show the fracture. So, these are the elements, where the damage has reached this critical value ok. So, this critical value or damage means when value of D reaches 0.6 for 1090 and 0.5 for 1045, I say that that particular element has lost its stress carrying capacity and then, the stiffness is set to a very small value and that is blackened.

So, this is the fracture. So, clearly for 1045 which is more ductile, you can see the cup cone kind of fracture; while 1090 is a much more brittle kind of thing, you have a flat surface ok.

(Refer Slide Time: 53:25)

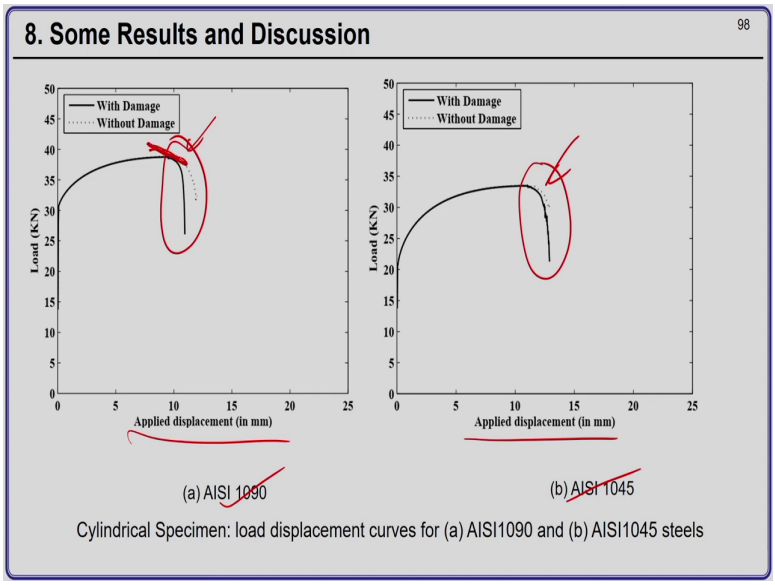


And from the experimental results reported by LeRoy, I compute the numerical result for damage and versus the equivalent plastic strain at the centre and at the outer surface and match with the experimental result and you can see the our numerical results match well with the trend for the numerical results of experimental results of Celentano for both 1090 and 1045 ok.

So, you see that at the outer surface, the damage growth is very slow; while at the centre, it is much more faster. So, this is also observed in the literature that the crack will initially start at the centre and propagate towards the outer surfaces and you can see for 1090, its more or less the crack will occur all throughout the in centre surface; but, for 1045 which is much more

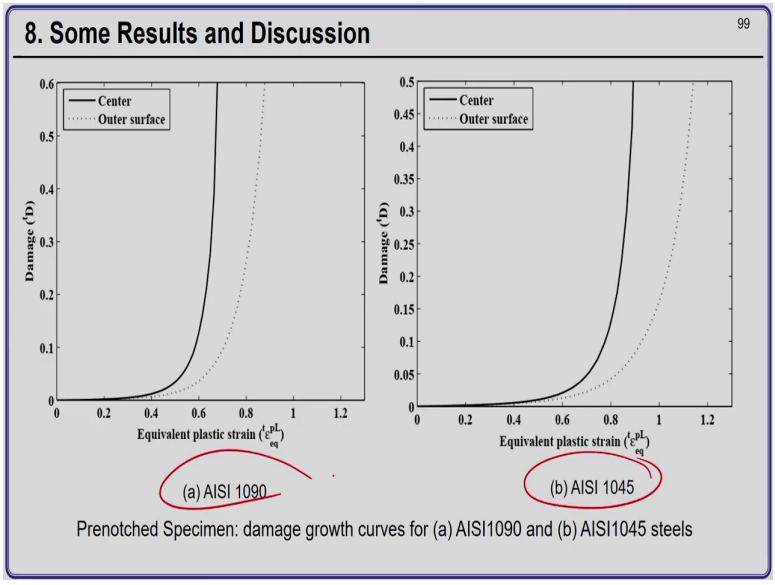
ducktail, it will first it will occur at the centre much more earlier than at the outer surface, that is why it leads to the cup cone fracture ok.

(Refer Slide Time: 54:26)



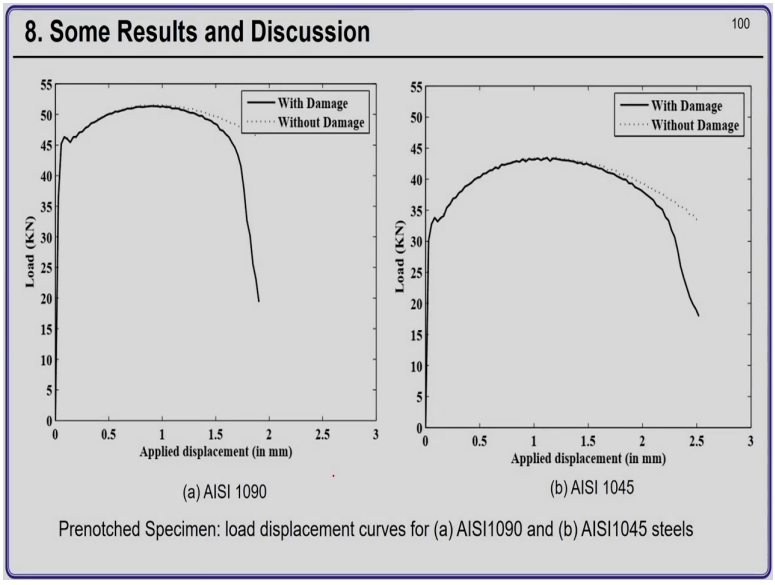
Again, this is a load displacement graph for the 1090 and 1045 and you can see you have a sharp drop in the load, once the necking starts and the damage reaches a very high value there. Therefore, you need arc length method. Here, to solve these kind of load displacement curve here because your tangent ok; if the tangent matrix will become singular ok. So, you need some arc length method ok.

(Refer Slide Time: 54:52)

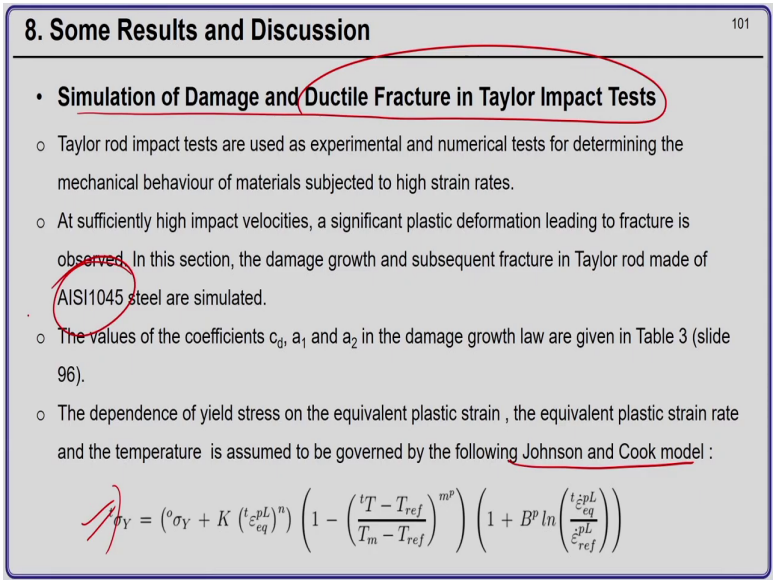


So, this is the growth of damage versus plastic strain at the centre and outer surface for different values of plastic strain ok.

(Refer Slide Time: 55:01)

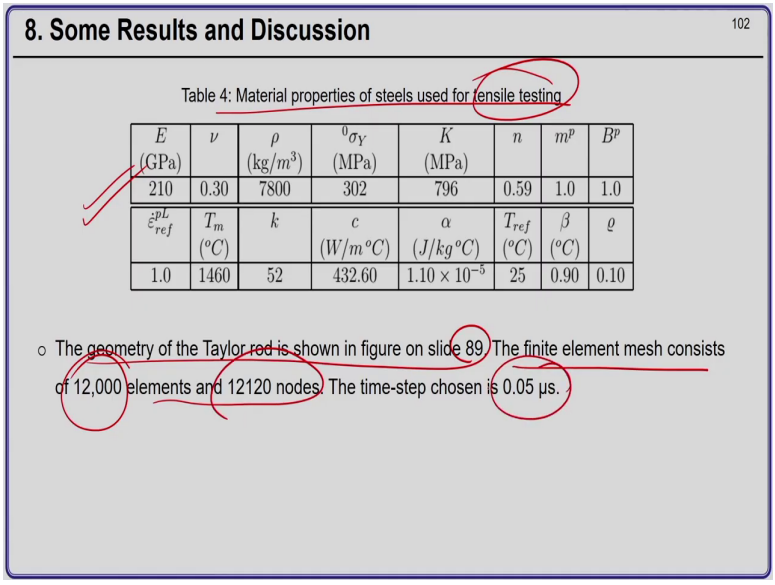


(Refer Slide Time: 55:04)



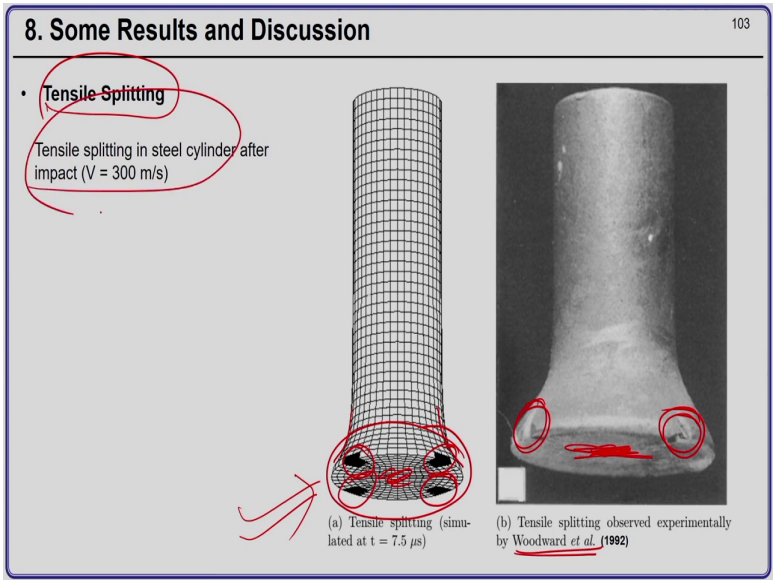
So, let me just skip. Now, we simulate the damage and ductile fracture in the Taylor rod impact tests ok. So, we take the geometry as we discussed in the validation; but now, we take a material which is 1045 and for that we use following Johnson-cook formula for which the material properties are given here.

(Refer Slide Time: 55:24)



And they are obtained using the tensile testing and then, this geometry of the Taylor rod is shown in slide 89 and the finite element mesh consists of 12,000 elements and 12,120 nodes and the time-step that we have chosen is 0.05 microsecond. So, this is a very high impact velocity problem. Therefore, we choose a very small time step.

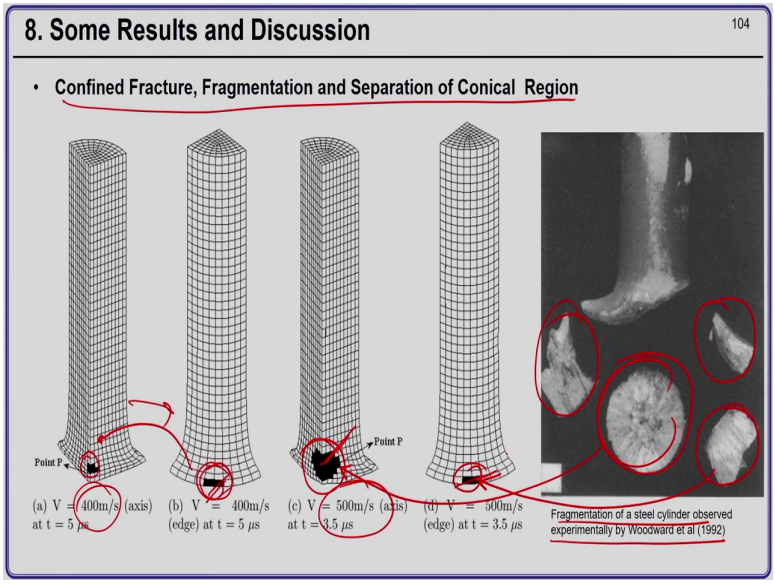
(Refer Slide Time: 55:52)



So, there are two kind of failures that are observed in Taylor impact tests; one is called the tensile splitting. So, at 300 metre per second impact, you can see that at the impact phase, you have a mushrooming and also you have fracture which occurs at the 4 corners ok.

And if you measure this kind of result with the experimental result, you will see in the experimental results of Woodward et al also, there is tensile splitting which occurs at the outer surface while the inner surface is more or less outside of that there is no visible fracture ok. So, we observe tensile splitting for lower impact velocities which is also observed in the literature.

(Refer Slide Time: 56:41)



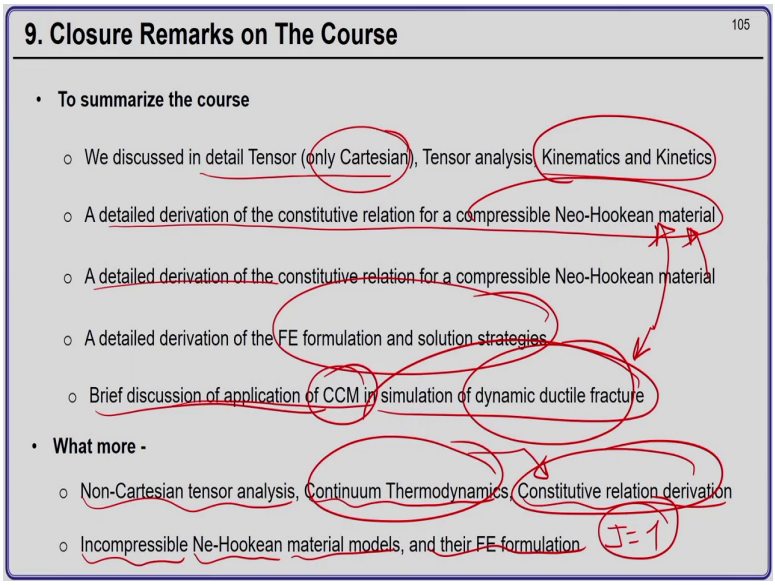
And for higher impact velocities say 400 or 500 metre per second, experimentally people are observed that you have fragmentation ok. So, you have some small parts at the outside surface which come out and at the middle, there is a huge chunk of a material that comes out ok.

So, here also for lower impact velocities, you can see at the centre there is some damage and also, at the outer surface there is some damage ok. So, they are same view. So, this is actually at the other side ok.

And for higher impact velocity, you can see there at the middle, there is a huge chunk like this here ok; there is a chunk like this here and in the outer surface, you will have chunk like this here ok. So, again the numerical simulation is able to capture what is called the confined

fracture, fragmentation and separation of conical zone ok. So, this conical zone and this is fragmentation which are coming out ok.

(Refer Slide Time: 57:39)



So, with these numerical results, we come to the end of this course and we have some closure remarks on this course. So, to summarise this course, we have discussed in detail tensor and tensor analysis. We did this only for the Cartesian ok; but for non-Cartesian, we did not do it because it was not possible.

But with kinematics and kinetics that we discussed was very general that you will find in a course on continuum mechanics. Also, we did a detailed derivation of the constitutive relation for compressible Neo-Hookean material ok. We did very detailed derivation so that you could understand.

We also did a very detailed derivation of the Finite Element formulation and various solution strategies ok. What we also discussed the application of computational continuum mechanics to the dynamic ductile fracture ok; although, we did very detail in new this hyper elastic material.

But for the sake of completeness, we also discussed in detail the a little detail about the dynamic ductile fracture, where elastoplastic effect, thermal effects, strain rate effect and the damage everything comes into one particular simulation. So, we did one kind of multi physics problem.

So, what more you can do? Non-cartesian tensor analysis, you can study by yourself if you want, we can refer you can refer to some books. I can give you reference to some books, if you are interested. Continuum thermodynamics is not what we discussed. This is discussed mostly in the course on continuum mechanics, not in this applied course and also, the constitutive relation derivations ok.

For example, the from the given Helmholtz free energy or the Gibb's free energy, potential how to derive the constitutive relations that we did not discuss. We directly took a material model which is Neo-Hookean material model. So, this also can be done in a course on continuum mechanics ok; and additionally, we can do incompressible Ne-Hookean material modelsand their finite element formulation.

So, incompressible means there will be no volume change and there is a condition J should be equal to 1 and so, all the deformation should be such that J should come out to be 1. This kind of constraint, we did not study in this course. But obviously, if you are interested, you can go into the references and look into this ok. So, with this, we will come to end of this course.

Thank you very much.