PROFESSOR: Ladies and gentlemen, welcome to this course on nonlinear finite element analysis of solids and structures. A few years ago, we produced at MIT a course on the linear analysis of solids and structures. That course was quite well-received. And we obtained a number of requests to also produce a course on the nonlinear analysis of solids and structures. The present course is the answer to those requests.

In this course, we will be using my book as a textbook. You might be already a bit familiar with this book. We will be referring to some of the sections in quite some detail. In addition, of course, you also have the study guide.

Let me now share with you, briefly, some thoughts that I had while designing this course. The field of nonlinear, finite element analysis is a very large field. And, in fact, four large fields come to mind as feeding into nonlinear, finite element methods, continuum mechanics, finite element discretizations, numerical algorithms, software considerations.

Because the field of nonlinear, finite element analysis is such a large field, I had to select certain topics as the topics of this finite series of lectures. I believe that the lectures provide a good introduction and foundation to a nonlinear, finite element analysis. Of course, the lectures cannot answer all the questions that you might have regarding nonlinear, finite element analysis. I believe, though, that they will be answering some of the questions and, hopefully, stimulate discussions that you might have while watching these lectures with your colleagues.

We, at MIT, continue to work in nonlinear, finite element analysis. And we also offer, from time to time, short courses. It would be very nice to meet you at one of these short courses and to discuss any questions that you might have regarding these lectures, once you have seen them, and regarding your work on nonlinear, finite element analysis in general.

Let me now return to what I like to talk about in this course. In this course, we want to concentrate on methods that are generally applicable, modern techniques and practical procedures. I believe it is important that we discuss in this course practical and effective procedures. In particular, methods that are or are now becoming an integral part of computer-aided design, computer-aided engineering software.

In this course, I would like to discuss with you geometric and material nonlinear analysis, static and dynamic solutions, basic principles and their use, and share with you example solutions. I believe that the course will be of interest in many branches of engineering throughout the world. And in this spirit, we have designed a logo for this course.

The logo is shown here. It shows the whole world as an assemblage off finite elements. If you're watching me from Chicago, you're watching me from about here. If you're watching from Munich, you're watching about there. And if you are in Tokyo, you're watching me from about there. Welcome to this course.

I'd like to now start with the first lecture, which has the title Introduction to Nonlinear Analysis. And let me take off my jacket, because we have a lot of work ahead, and summarize to you what I would like to present to you in this first lecture. We discuss, first, some introductory view graphs and show some short movies.

We then classify nonlinear analyses. We then discuss the basic approach of an incremental solution. And we share some example solutions. Let me walk over to my view graphs which I've prepared for this lecture so that we can start with a discussion of this material.

Finite element nonlinear analysis in engineering mechanics can be an art, but it can also be a frustration. For those of you who have been doing some nonlinear analysis already, I think you will value that it can be an art and it can be a frustration because it can be a very difficult matter. But it's always provides a great challenge.

And that, of course, is the exciting part of working in nonlinear, finite element analysis.

Some important engineering phenomena can only be assessed using nonlinear analysis techniques. For example, the collapse or buckling of structures due to sudden overloads-- I'm thinking here, typically of shell structures, say-- the progressive damage behavior due to long-lasting severe loads such as, for example, high-temperature loads in nuclear reactor components and, for certain structures such as cables and transmission towers, nonlinear phenomena need be included in the analysis, even for service load calculations.

The need for nonlinear analysis has certainly increased in recent years due to the need for the use of optimized structures, the use of new materials that are being introduced or have been introduced over the recent years, the addressing of safety-related issues of structures. There's more rigorous attention being given to such safety-related issues now and certain the corresponding benefits can be most important.

Problems that are addressed by nonlinear, finite element analysis can be found in many branches of engineering. And I've listed here some such branches, nuclear engineering, earthquake engineering, the automobile industries, defense industries, in aeronautical engineering, mining industries, offshore engineering and so on.

I'd like to now show you some movies that we have assembled regarding some finite element analysis in certain fields. And let us start with a movie related to the defense industries, then go on to a movie related to some work in the automobile industries, and then look at a movie related to some work in earthquake engineering. And finally, I'd like to also show you an interesting movie regarding a structural engineering problem.

So let me now introduce you to the first movie, which is a movie in which a tank is modeled using a geometric model first and then setting up a finite element mesh. Here we see a tank in a maneuver. You surely have seen such structure before. And here you see, in the top part of the picture, the geometric model of a tank and, in the bottom part of the picture, a finite element discretization.

The top model was generated using a geometric modeler. And the bottom model was obtained using a finite element mesh generator. These models were constructed by Structural Dynamics Research Corporation.

The second movie relates to a problem that has obtained much attention in the automobile industry, namely the problem of what happens when a motor car crashes against another motor car. And tests have been performed of motor cars crashing against rigid walls. Test results have been assembled. And finite element methods may well be used in the analysis of such types of problems.

Here we see a test performed by the Ford Motor Company of a car crashing against a rigid wall. And here, a close-up of what happens to the passengers not wearing seat belts, and another such test. Here now, in more detail, what happens to the front part of the car. There is since some time much interest in modeling these phenomena on the computer. And finite element methods can be applied and are very valuable here.

This certain movie relates to the use of finite element methods in earthquake engineering. Here we look at the finite element model of a tank and the dynamic motions of that model. We are surely all aware that earthquake motions can cause severe structural vibrations and possibly collapse of structures.

Here we see a transformer and the tank to be modeled on it. Here a close-up view of the tank. The finite element model of the tank was constructed by Asea, a Swedish company. And here we see some vibratory motions of the model.

The fourth movie shows the dynamic response and collapse of the Tacoma Narrows bridge in 1940. The Tacoma Narrows bridge collapsed on November 7, 1940, about four months after its opening in winds of 40 to 45 miles per hour. Here you see the dynamic motions of the bridge prior to its collapse, a side view, and now a view along its center line.

Notice the high torsion of vibrations of the bridge. The bridge went through large dynamic motions for hours until its collapse. We only show a very small segment of that time. And here you see how the bridge collapsed.

Of course, these movies only indicate, to some extent, where finite element methods might be applied in engineering practice, but I thought you might like, you might enjoy, seeing the movies. Let me now continue with the material that I have documented on the view graphs.

For an effective nonlinear analysis, a good physical and theoretical understanding is most important. You want to have some good physical insight in the problem, setup, and mathematical formulation of finite element model. Solve that model, and that will enrich your physical insight. It is this interaction and mutual enrichment between the physical insight and mathematical formulation that can be most valuable.

The best approach for a nonlinear finite element analysis is to use reliable and generally applicable finite elements. With such methods, we can establish models that we can understand, that we have confidence in. We start with simple models of nature. And we find these as need arises.

In fact, I like to think of an engineer as developing a first model on the back of an envelope using, of course, simple equations. And these equations will give some insight into the structural response. If necessary, then the finite element model is set up. The first finite element model is set up. And this finite element model is then refined as need arises.

To perform a nonlinear analysis, we want to, altogether, stay then with simple, relatively small and reliable models. We always want to perform a linear analysis first. I will show examples in this course particular related to this item here.

It is very important, in my view, to first do a linear analysis and then only go on to a nonlinear analysis. As I mentioned already, we want to refine the model by introducing nonlinearities as desired and, once again, use reliable and well-understood models, obtain accurate solutions of the models. This is very important for possible proper interpretation of the results.

Here we show schematically the finite element modeling process. We have a problem in nature. We model that problem, the kinematic conditions, the constitutive relations, the boundary conditions, the loads, and so on, using finite elements. We solve the model and interpret the results.

Now this model surely can only approximate the actual problem in nature. And on interpretation of the results, we may find that we really should refine our model. And we do so, set up a new model, solve again. And like this, we may cycle a number of times through this process.

Of course, traditionally, test results have been sought for very complex problems, laboratory test results have been sought. And this may still be necessary now. However, the finite element modeling process will certainly compliment these laboratory test results.

Let's look at a typical nonlinear problem. Here we have a bracket of mild steel subjected to the load shown. And the possible questions that we might ask are, what is the yield load of this bracket? In other words, at what load do we see first plastic deformations?

What is the limit load? What is the maximum load that this bracket can take? What are the plastic zones? What are the residual stresses when the load is removed? Is there adhering where the loads are applied right around here?

What is the creep response of the bracket when the bracket is subjected to high temperature conditions and these loads? And what are the permanent deflections of the bracket, et cetera. There are many more questions that could be asked. And certainly, these questions here can only be answered by a nonlinear analysis.

Possible analyses that we might consider are the following. First we, of course, would always perform, as I mentioned earlier, a linear elastic analysis. The linear elastic analysis would determine the total stiffness and the yield load of the bracket. Notice here we show the displacements in red. Actually, these displacements would be very small. And they are magnified in this view graph.

Another analysis now would be to perform a plastic analysis, but assuming still small deformations. This analysis would determine sizes and shapes of the plastic zones. The displacements here are still small. We have not shown them at all actually here. They are very small, however.

Then we might want to go on to a large deformation plastic analysis. This analysis would determine the ultimate load capacity of the bracket. And notice the displacements are now large, they are actually large. Here, we have had small displacement assumptions. Here, we have included the large deformation effects as also shown on the view graph.

For analysis, it is very good to actually classify all the types of analysis that one might want to perform. And the first category of nonlinear analysis is the one that we call Materially-nonlinear-only analysis, MNO analysis. In this analysis, we assume that the displacements are infinitesimal, the strains are infinitesimal.

In other words, both of these quantities are very, very small. And the stress-strain relationship is nonlinear. So all nonlinearities lie, really, in here, in the stress-strain relationship.

Here, I'm showing a schematic example. You're looking at a four-node element that is subjected to the loads shown, and delta is the displacement. Notice that delta/l is very small.

In fact, even this four is already relatively large. We should really have here, possibly, a two. So nonlinearity lies in the material description. The material is an elasto-plastic material, Young's modulus, E, yield stress, sigma y and strain-hardening modulus, Et.

As long as the stresses do not exceed sigma y, we really have a linear analysis. So if you use a computer program with an MNO formulation and you subject your model to forces such that the stresses are below sigma yield everywhere in the model, then you should really obtain the linear analysis results.

The next category of problems is the one of large displacements, large rotations, but small strains. Here, in other words, we still keep the assumption of small strain, but we allow large displacements and large rotations. The stress-strain relations can be linear or nonlinear.

Schematically here, once again, our four-node element. This four-node element now would move, as shown, going through large displacements and large rotations. But the strains in the element, expressed by delta prime over l are still small. Once again, you may actually want to make this 0.02. As long as the displacements are very small, we have now, really, an MNO analysis.

The third category of problems is the one of large displacements, large rotations and large strains. Here we have included all kinematic nonlinearities. And the stress-strain relation is probably also nonlinear because we are dealing with large strains.

As an example, we have, again, our four-node element here moving as shown. Notice material fibers here, displaced by a large amount, rotate and are also stretched by a large amount.

This, of course, is the most general formulation of a problem. However, still not considering any nonlinearities in the boundary conditions, nonlinearities in the boundary conditions provide contact problems. And here, we look schematically at a simple contact problem.

The four-node element, once again, subjected to loads, there's a gap here between this element and the spring. And as soon as this gap is closed, the spring, of course, provides stiffness into the system, the system being now the four-node element. And, well, that difference has to be taken into account in the analysis.

And there are procedures to solve contact problems. This is, of course, a very simple contact problem. But there are procedures to solve contact problems.

Contact problems are very difficult problems to handle, particularly if you're dealing with frictional effects. We will, in this course, in this set of lectures, really not address contact problems, except that, in the last lecture, we actually look at one particular contact problem. We use a computer program for the analysis of a contact problem there.

Let's turn to an example analysis. And I'd like to consider with you, briefly, the analysis of a bracket. Here is a bracket of the kind that we looked earlier at already. Notice the dimensions are as shown. The thickness of the bracket is one inch. And this bracket is going to be loaded to large loads.

We make a material assumption, namely the one of an elasto-plastic material with isotopic hardening as shown here. The yield stress is 26,000 psi. Here you have the Young's modulus. And here you have the strain-hardening modulus.

We use symmetry conditions to analyze the bracket the same way as we would do it, of course, in a linear analysis. And with the symmetry conditions, we consider on this view graph just the lower part of the bracket. You see a pin here, rollers there. And we're using eight-node, isopolymetric elements to model this part of the bracket.

We apply the load as shown here. Notice no special considerations to the hole which the load is applied. Our interest really lies in predicting the stresses and strains in this region and to also predict the overall collapse of the bracket.

We will use three kinematic formulations for the analysis. First of all, we use the Material-nonlinear-only analysis assumption. In other words, small displacements, small rotations and small strains are assumed in the analysis of this bracket.

Then we will perform an analysis using the total Lagrangian formulation. We will discuss this formulation quite extensively in later lectures of the course. This formulation assumes large displacements, large rotations and large strains kinematically. Kinematically, these are the assumptions.

However, we will point out that the material law that we are using, or the material law description that we are using, is only applicable with this formulation to small strains. So the overall analysis, using this total Lagrangian formulation, is then only applicable to model large displacements, large rotations, but only small strains. I get back to that just now.

We will also use an updated Lagrangian formulation which kinetically includes large displacements, large rotations and large strains. And on the material model level, we also include there large strain effects. So once again here, as summarized on this view graph, the material used in conjunction with the total Lagrangian formulation is actually not applicable to large-strain situations, but only to large displacements, rotation in small-strain conditions. So once again, our total analysis is only applicable to this kind of situation. Whereas, the updated Lagrangian formulation does model, kinematically and on the stress-strain level, large displacements, large rotations and large strains.

The analysis results obtained are shown here. Notice we are plotting here the force applied and the total deflection between points of load application. The three analyses give us three distinct curves at large displacements. But for small displacements, of course, these curves are indistinguishable.

Notice here we have about 10% strain at point A. I will show you just now where point A is. The important point to notice is that these are three distinct curves. They are distinct because we have made different assumptions, kinematically and on the stress-strain load level. We will talk about such assumptions in the later lectures of this course.

On this view graph, now you see this point A, which I mentioned earlier. And you see also the original mesh shown in dashed, black lines, and the deformed mesh, shown in red, corresponding to a level of load of 12,000 pounds. I'd like to now look with you at two animations regarding problems that show nonlinearities typical of the nonlinearities that we are talking about in this course.

The first animation shows a plate with a hole that is subjected to high tensile forces. The plate undergoes plastic deformations. And, in fact, the load becomes so large that the plate basically ruptures. Let's look at this animation now.

Here we see one quarter of the plate. The plate is subjected to a uniformly distributed tensile load along its upper edge. For the quarter model of the plate, symmetry boundary conditions are imposed along the left vertical and lower horizontal edges.

The time code given above the plate gives the time of the load step. Also given is the load applied at that time. However, note that we perform a static analysis. Therefore, the time code merely denotes a load level, as we will discuss later in detail.

Each time increment of one milli corresponds to a load increment of 12.5 megapascal. We will increase the load monotonically. The plate will become plastic at the hole. And this plasticity will then spread until, in essence, the plate ruptures.

We used 288 eight-node plane stress finite elements for the quarter of the plate. And we show to spread of plasticity in the plate by darkening the areas that are plastic. Here now you see the time and load increasing. In the first load steps, the plate remains elastic and the deformations are small.

Now you see the first plasticity developing near the hole. The plastic zone increases rapidly as the higher load levels are reached until a large portion of the plate is plastic and, in essence, the maximum load-carrying capacity has been reached. We will consider the analysis of this plate in more detail later in the course.

The second animation shows a frame that is subjected to forces such that the frame undergoes very large displacements. Here we see the frame model, using beam elements that we discuss later in the course. The frame is loaded, as shown, by the force arrows. There are pin connections at the points of load application.

The frame is assumed to be of an elastic material, hence, no plasticity is assumed to develop or that the frame will be subjected to very large displacements. The indicated loads will push the top of the frame down and the bottom up to such an extent that the points of load application cross over. You see above the frame a time and a corresponding load level. This is, again, a static analysis. And each time increment of one corresponds merely to a load increment of 250 pounds.

Here now, you see the deformations of the frame develop as the load is increased. The displacements are very large at the higher load levels. Of course, this is only a numerical experiment at the high load levels, but an interesting one that indicates in a simple manner what can all be done in finite element analysis. We will consider this problem solution also in more detail later in the course.

Let us now look at the basic approach of an incremental solution. We consider a body, a structure or solid subjected to force and displacement boundary conditions that are changing, and we describe the externally applied forces and the displacement boundary conditions as a function of time. Schematically, here we show a body, of course, supported as shown and subjected to a force varying as a function of time and a prescribed displacement varying as a function of time.

The time variation of the force is shown down here. And the time variation of the prescribed displacement is shown here. Notice that here we have a particular time point, T. And here we have a time point T plus delta t. Here we have, similarly, the time point T and here, also, the time point T plus delta t.

Since we anticipate nonlinearities, we use an incremental approach measured in load steps or time steps. And this means that the loads, the prescribed loads, the imposed loads and the prescribed displacements, are discretized as a function of time, as shown on this view graph. Notice here we have, at time T, the impulse load TRI.

This upper, superscript T means at time T then delta t, in advance, time T plus delta t. We would have, in other words, T plus delta t ri. In other words this T would now be replaced by T plus delta t and so on. This is how we are inputting or prescribing the impulse loads and also the prescribed displacements.

When they apply forces and displacements very slowly, meaning that the frequencies of the loads are much smaller than the natural frequencies of the structure, we have a static analysis. This, of course, means that the periods of the loads are very long when measured on the natural periods of the structure. In other words, you have a spring and you apply a load that varies very slowly. And this means we have a static analysis. If the load's very fast, and by that we mean that the frequencies of the loads are in the range of the natural frequency of the structure, then we have a dynamic analysis.

Let us look a bit closer at the meaning of the time variable. Time is a pseudo-variable, only denoting the load level in nonlinear static analysis with time-independent material properties. As an example. here we have a cantilever subjected to a load, R, a tip load, R. And if we were to perform a run, 1, in which we prescribe the loads as shown here-- and notice, at time 1, a load of 100, at time 2, a load of 200-- delta t is equal to 1.

If we were to perform this analysis and, in addition, this analysis where delta t is equal to 2 and the load at time 2 is equal to 100, the load at time 4 is equal to 200, then we would obtain identically the same results in run 1 and run 2 because it's a static analysis and the material properties are time-independent.

So in this particular case, certainly, time is a pseudo-variable. And we can design a time-stepping. We can design many different time-stepping schemes using different time steps and always obtain the same results, provided, at the end of the first time step, we have the load 100 and at the end of the second time step we have the load 200. However, time is an actual variable in dynamic analysis and in nonlinear static analysis with time-dependent material properties, for example, when the material contains grid conditions.

Now delta t must be chosen very carefully with respect to the physics of the problem, with respect to the numerical techniques used and the costs involved. If delta t is not chosen appropriately, you may have a very high cost for the analysis. And on the other hand, you may, with improper choice of delta t, also obtain very bad results. So it is very important to choose delta t judiciously for an accurate and cost-effective analysis.

At the end of each load or time step, we need to satisfy the three basic requirements of mechanics, equilibrium, compatibility and the stress-strain law. These are the three fundamental requirements to be satisfied in mechanics. This is achieved in finite element analysis in an approximate manner using finite elements by the application of the principle of virtual work.

Now there is a lot of information on this view graph. And we don't have time in this lecture to go into any depth. There is, of course, quite a bit of mathematics that has to be introduced for the discussion of all of what we see here on the view graph. And that's what we do in the later lectures. I do not have time now to go into these mathematics, but let us just very briefly look at the basic procedure that we using.

We are saying that, at any time step, T plus delta t, the externally applied loads, the vector of externally applied loads and this vector, includes pressure loads. Concentrated loads and, in the dynamic analysis also inertial forces, this vector must at any time T plus delta t equal to the vector F at time T plus delta t where this vector F corresponds to the internal element stresses at time T plus delta t. We will talk in the later lectures about how we calculate this vector T plus delta tF. Of course, there are some very important considerations in the proper calculation of this vector F.

Let us now assume that the solution at time T is known. Hence, the stresses, at time T, are known. The volume surface area and so on, all of the information corresponding to the body at time T is known. And we now want to obtain the solution corresponding to time T plus delta t, that is for the loads applied at time T plus delta t.

Of course, this is a typical step of the incremental solution. Once we have the solution for time T plus delta t, we can use the same scheme to calculate the solution for time T plus 2 delta t and so on. For this purpose, we solve, in static analysis, this set of equations.

TK is a tangent stiffness matrix of the system. Delta u is a vector of increments in the nodal point displacements. On the right-hand side, we have the externally applied loads corresponding to time T plus delta t in this R vector. And here, TF corresponds to or is the vector of nodal point forces that correspond to the internal element stresses at time T.

Notice this is an out-of-balance load vector. And this out-of-balance load vector gives us an increment in displacements. This increment in displacements is added to the displacements at time T. And that gives us a displacement vector corresponding to time T plus delta t. Notice I have an approximation sign there. This approximation sign is a result of the fact that we obtained this set of equations by linearization.

We will talk about how these equations are obtained in the later lectures. We will start from the basic principle of virtual work and use continuum mechanics considerations to derive, in a very consistent manner, this set of equations. Of course, there are many details involved.

We will talk about the total Lagrangian formulations, the updated Lagrangian formulation, the Material-nonlinear-only analysis, the details that go into these formulations in terms of kinematic approximations, kinematic approximations related to large displacements, large rotations, large strains. We also will talk about the constitutive relations that enter in this tangent stiffness matrix and in this force vector and so on.

Anyway, this set of equations gives us an approximation to the displacements at time T plus delta t because of the linearization process used to arrive at these equations. And, more generally, we want to solve this set of equations, tangent stiffness matrix, still on the left-hand side, times a displacement increment corresponding to the iteration, I. And on the right-hand side, we have the load vector corresponding to the loads at time T plus delta t and the nodal point force vector corresponding to the internal element stresses at time T plus delta t and at the end of iteration I minus 1.

This is here the out-of-balance load vector corresponding to the start of iteration I. This out-of-balance load vector, and with that tangent stiffness matrix, gives us an increment in the nodal point displacement vector corresponding to iteration I. We take this one, add it to the previous displacements that we had already, namely those corresponding to time T plus delta t at the end of iteration I minus 1, and we obtain a better approximation to the displacements at time T plus delta t.

Notice, in this iteration, we have initial conditions that I've given down here in iteration I equal to 1. On the right-hand side we need, of this equation here, we need this vector. And that vector is given by the vector TF, the nodal point force vector corresponding to the internal element stresses at time T. We also need initial conditions on the displacement vector, and those are given right here.

Notice that, when I is equal to 1, this set of equations reduces to the equations that we had on the previous view graph. Once again, we will derive these equations from basic continuum mechanics principles in the later lectures. And we will also talk about iterative schemes to accelerate the convergence in the solution of these equations.

This then brings me to the end of what I wanted to discuss this you on the view graphs. I'd like to now turn to an example solution, an interesting example solution. But, moreover, it displays the kinds of nonlinearalities that we will be talking about in this set of lectures. This example solution is documented on slides. So let me walk over here and put on the first slide.

Here we show the structure that we want to consider. It's a spherical shell subjected to pressure loading. The material data for the shell are given here. Notice we also include the mass density because we will be talking about dynamic analysis of this shell as well.

We will also introduce an imperfection of the shell. And that imperfection is given down here. Notice it is given as a function of the angle phi, which you see up here, as well as a thickness, H, of the shell, the Legendre polynomial given here and parameter data that we will be varying.

First, we will consider a static analysis of the shell and then the dynamic analysis of the shell. On the next slide, now you see the model that was used for the analysis of the shell, twenty 8-node elements, axisymmetric elements subjected to the pressure loading, and, we are modeling, the pressure loading in deformation-dependent loading. That means that, as the shell deforms, the pressure will remain perpendicular to the shell surface to which it is applied.

On the next slide now, we show the first analysis results. On this axis, we plot P/PCR where PCR is the buckling pressure of the shell, assuming elastic conditions, and calculated using analytical solution. On this axis, we plot the rate of displacement of the shell at phi equal to zero. The dotted line here shows the linear elastic solution to the shell problem. Notice that these dots, of course, could be continued here.

The total Lagrangian formulation, which includes large displacement and large rotation effects, gives us a buckling pressure of 0.98 PRC. A Material-nonlinear-only analysis, including the elasto-plastic material conditions, but not including, in other words, large displacement, large rotation effects, gives us this solution here. Notice that, of course, this MNO analysis does not give us a proper bucking prediction for the shell.

To obtain the elastic-plastic buckling load of the shell, we have to perform a total Lagrangian formulation solution. And we obtain this load level here as the bucking pressure. This analysis was performed with no imperfections on the shell.

The next slide now shows the results of the same kind of analyses. But at an imperfection of delta 0.1, notice here the linear analysis, denoted as E, the Material-nonlinear-only analysis results, denoted as EP, the large displacement, large rotation analysis results. But assuming an elastic condition for the shell, denoted as E, TL, TL standing for total Lagrangian formulation and the elasto-plastic large displacement solution given here, the results given here and denoted as EP, TL. Notice that, because of the imperfection, the load, the maximum load-carrying capacity of the shell, is decreased.

On the next slide now we show the maximum load-carrying capacities for delta equal to zero, delta equal to 0.1, delta equal to 0.2, delta equal to 0.4. In each case, we have used the total Lagrangian formulation including elasto-plastic conditions. In other words, of course, we had to include the large displacement effects in order to pick up the proper buckling load or load-carrying capacity of the shell, so these are the load-carrying capacities of the shell for different imperfection levels.

The next slide now shows the results obtained using a dynamic analysis. At this time, we apply the pressure instantaneously as a step load, constant in time. Notice we are plotting here now mean displacement data of the shell and time along this axis. The Etl solution results are very close to the E solution results. I think you know now what I mean by E and Etl.

The E Ptl solution results are quite close to the EP solution results. This means that the shell is stable under this load application. In other words, there is no buckling, no increase in deformations as time progresses.

The next slide now shows, as we increase the imperfection level from delta equal to zero to delta equal to 0.1, the displacements of the shell using the E Ptl solution or formulation increase with time. Therefore, the shell is unstable under this pressure application. Of course, dynamic pressure application, no. Notice the other solution results are given here.

Therefore, we now can play with the imperfection level and with the load level in order to estimate the load-carrying capacity of the shell for each of these cases. And on this slide, you're seeing for a constant imperfection level, we increase the load applied to the shell. And you can see that, at 0.4 PCR, the shell is still stable. At 0.5 PCR, the shell is unstable.

On the final slide now, we show all the information that we have obtained in the analysis. On this axis, we are plotting the buckling load, on this axis, the amplitude of imperfection. This curve here corresponds to the static backing of the shell. And this curve corresponds to the dynamic buckling of the shell.

Of course, we could not discuss all the details regarding these analyses in this short time. Please refer to the study guide in which you'll find a reference to the paper in which we have discussed the details of these analyses. This then brings me to the end of this lecture. And I hope you enjoy the course.