ANDREW SARTORELLI: All right, so my name is Andrew Sartorelli. I'm a technical support specialist here at AutoDesk. I work out of our Munich office. And with me today is David Truyens.

DAVID TRUYENS: Yeah, I'm David. I work in the simulation business development team from Vince Adams. And we try to enable everybody to be able to work with the software and to see how we can help you solve your challenges.

ANDREW SARTORELLI: So we're talking about different types of connectors in all of our products. So the workflows that we discuss today will be applicable Simulation Mechanical, Nastran In-CAD, as was the Fusion Ultimate Simulation. And again you can use-- so hopefully by the end of today, you'll understand when and how to use different connectors and different types of idealizations, as well as the limitations of all of these approaches, as well as which packages that you can use the different connector types in.

So some people that aren't familiar with FEA might be more familiar with the term fasteners. And in FEA, we generally talk about connectors. If you open any of our FEA packages, you're going to see that term. And they're synonymous with one another. So that's going to be bolts, springs, cables, rigid elements. It's going to be welds. You won't necessarily have weld-type connectors. But there's ways to use these idealizations.

So with fasteners, the big thing to think about is global versus local. So obviously if you have something like the Eiffel Tower, it's got 2.5 million rivets in it. If you're creating an FEA model, it doesn't make sense, if you're doing an upfront engineering approach, to model 2.5 million rabbits. You're going to end up with a lot of elements there that aren't really adding too much benefit.

Whereas maybe you're just concerned with a single fastener. In that case, you can go ahead and model that locally to examine the failure of that specific component. And oftentimes you can do the global model to figure out what specific fasteners in the assembly are actually going to be the ones you're going to look at locally.

So the first type of fastener we're going to talk about are bolts. This is probably the most common fastener that people are going to be familiar with and using in FEA. So we're going to run through a few different topics-- pre-processing, idealization methods, applying pre-loads, and then some applications of bolts.

So we got a quick video here going over using the bolt calculator in Inventor. So one thing to keep in mind when using bolts in FEA is oftentimes there's hand calculations that you can do, or Excel sheets that you can use to actually figure out the correct sizing of bolts before you bring it into FEA. You don't want to end up with a giant bolt in there. And you have no idea if this is the right size. Take the easy approach. And use the tools that you have available before going into FEA and doing the analysis.

So here we've been able to set up a number of sketch points in the model. We then use the bolt connector utility. And we can specify a bolt diameter as well as the loading conditions. And from those loading conditions, it's actually going to tell us the number of bolts that we need in this model.

So here it tells us we need six. So it's not automatically going to go ahead and change the number of bolts there. We're going to have to go back and make some changes to the Inventor model before we can see that change. So we'll just go through those steps here as well. So we're going to use some of the functions built into Inventor. Just change that number from eight to six.

So when we're talking about bolts in FEA, there's three main approaches to idealization. So we can think about contact. We can think about solid modeling bolts. And then we can use beam and bar elements to idealize bolts.

So the first approach using contact is typically when you have continuous behavior. So if we look at this component here, we don't have a single point of failure, I would say, with these bolts. The two components that are joined together with these fasteners are going to act as one. So we can use contact here, instead of going in and modeling all of these fasteners and rivets.

If we go with the solid approach, one thing I often see with customers is they're going to go in with all of the threads, the bevels, the strength marks, and the hexagonal head. These are all things that don't really matter in FEA. You're not concerned with your threads. If you add those in, you're going to have to add in a significant amount of contact, which is going to significantly grow the time it takes to run your analysis. Things like strength marks have no bearing in FEA. Get rid of those. It's just needless details that you're going to have in there.

So whether we're in Inventor or Fusion, we can go through and simply idealize this. So I took

the first model you see here on your left and I meshed that. And I ended up with over a million elements to mesh this with a high quality mesh.

And if we go all the way down to the right hand side there, after removing all these things like the bevel, the hexagonal head, the threads, we end up with approximately 1,200 elements. So it's a significant difference there in the element count. And we'll actually see, with some further idealization methods, we can get that number down significantly.

So with 1D approximations using beams and bar elements, in all of our packages you'll see a 3D representation of these. But actually in the back end, there's simply one dimensional elements that are generated there. So in In-CAD and Fusion Ultimate, you don't see the actual elements generated. But you see the idealization.

So here we end up with approximately 25 elements, compared to if you went with a full solid modeled bolt with threads and strength marks and bevels and hexagonal head, like I said a million elements. So this allows you to quickly iterate your analysis, rather than running significant run times with fully modeled solid bolts.

So one of the big things with bolts is you want to make sure you've got that pre-load in there, because it significantly impacts the stiffness of your model as well as the load transfer. So that's the big thing. You want to make sure you're correctly modeling those low transfer paths when you're using different types of idealizations. So with bolts, adding that pre-load in there allows you to correctly model that behavior. And we'll see in some examples later on, when you actually misjudge that pre-load, it can significantly impact the results that you're getting.

So here's just a quick overview of the screens that you would see when using both idealizations in all of our products. So all of them allow you to select geometric entities. And then with Simulation Mechanical on then, you'll actually see the elements generated rather than the 3-D representation. Simulation Mechanical uses more of a drawing environment versus the CAD inventive approach with Fusion and Nastran In-CAD.

So one of the big concerns if you're using solid model bolts-- so say you've taken it down. You removed the screws, removed all the unnecessary details. You want to have that pre-load in there in the model. But there's really no good way to add an axial pre-load with solid elements, or so some people say. But we actually have a good way to do it. We can use thermal loading to add pre-loads to solid elements.

So if we know our force, we know our pre-loading there, we can actually back out using some engineering equations. We end up with-- on the right here, we can come up with a thermal coefficient of expansion. And we just assume that our change in temperature is one degree.

When creating a specific orthotropic material to take advantage of this coefficient of thermal expansion, you want to make sure that you have your coefficient of thermal expansion, the axial direction, to be the same one that you calculate here. But when you're defining your orthotropic material properties, you want a coefficient of thermal expansion of approximately zero in the other directions. Otherwise simply, you have convergence problems. And some solvers me won't accept a zero entry there.

So I actually went ahead and just created a simple model here to show that compression that's going on. I simply calculated, based on a 700 pound pre-load in a steel bolt. They I was able to come up with a coefficient of thermal expansion there.

So we've got our first case study here. I want to--

AUDIENCE: Do you guys have just another way to pre-load solids? Or are you going to go on? Do you have another way you could show?

ANDREW SARTORELLI: Go ahead, Vince. I don't have another way.

AUDIENCE: You don't? I have one. What you can actually do, using bolt connectors is you can-- here's your bolt, bolt to [INAUDIBLE]. Put a small cut in the middle and connect the two ends of the bolt to the bolt connector. Put the pre-load on the bolt connector. And that way, you're guaranteed that load. Whereas with thermal, it's what used to happen in it. But it's hit or miss. If you can use a bolt connector between two solid bolts, it can imagine contact and a little more response. [INAUDIBLE].

ANDREW SARTORELLI: So we've got our first case study here. I pulled this out of my Machine Design textbook. So we've got eccentric loading on two plates that are joined together with four bolts. And what I've done is I've run a number of-- it's essentially parametric study, varying bolt diameter given the conditions here.

So I backed out the forces that each bolt will experience using some hand calculations based on the instantaneous center theory. So we actually can come up and figure out exactly what forces in the y and z direction that we're going to see each bolt. So my study, I was just

focusing on bolt number one here. And I went through and examined the forces on that bolt.

So I varied the bolt diameter from a 1/4 inch, all the way up to 3/4 of an inch, as well as the mesh density along the bolt hole itself, and running linear and non-linear solutions. So here we can actually see the different shear forces that we calculated in the y direction. And we can see the analytical solution of 200 pounds as this blue line here.

For some of the larger bolt sizes, we're actually under-estimating that shear force due to the way, with a 1D idealization, the bar and beam elements we generate sometimes can have larger cross-sections. So this is why it's important to make sure that you're sizing your bolts correctly before you go into FEA. If you're adding in larger bolts than are actually necessary, you're overestimating that stiffness.

The other important thing to see here is that for the quarter inch bolt here, which is the solution that we generate with the Inventor bolt calculator, we're relatively constant regardless of mesh size down here for the shear force in the y direction.

So that's important to keep in mind. I often see customers when they're using these bolts connectors, they're going in, and they're refining the mesh significantly in that region. But when you correctly size your boat, you don't actually need to go in and refine that, because you have a singularity in that area where you're connecting all of these bar elements. So when you keep adding more and more bar elements, you're increasing the stiffness in that region and affecting your results.

And again we can see the same thing when we take a look at the shear forces for the bolt in the z direction as well. And then again I did a mesh sensitivity study comparing the max von Mises stress in the region around the bolt. And we can see that with a decreasing mesh size, the stress continues to increase. So we know that when we're using idealizations, we're doing this, again, for a global approach rather than a local approach. So if you're concerned with localized stresses in the region of a bolt, you should be using solid elements there to actually look at that on a local level rather than a global level.

So you can also use-- the example I did used solid elements for the plates. But you can also use shell elements. But some important things to keep in mind when doing shell elements in this example is checking your normal direction. So your contact between the plates is going to be determined based on that normal direction using contact offsets. So if you're David

Cordova's presentation, I'm sure he talked about using offsets with shell elements. And then using the edges for the bolt connector rather than surface typically generates the elements in a better manner.

So what we found was there's a small variance in the shear forces compared to the solid elements. But we're seeing displacements. And obviously reaction forces match up quite closely. So some takeaways from this examination is you want to make sure that you're sizing your bolts and determining your pre-loads before you go into FEA. Use those hand calculations. Use those that Excel sheets that you're familiar with. And then add that level of detail into your analysis.

And again don't refine your mesh around your bolt hole It doesn't add any value there. It just increases run time. And on this point of determining pre-load before you go into FEA, I actually made the mistake, when I was preparing for this class, of having an excessive pre-load on some of my bolts. So you can see here, for the 1/4 inch diameter bolt, I had a 20,000 pound pre-load on this boat, which was significantly more than was what was required based on the Inventor bolt connector calculator. And you can see where we're off by 25% on our shear force there.

So we'll skip over now and talk a bit about welds. So Vince did a great job last year of covering quite a lot about welds and weld failure specifically. So we're just going to focus on-- if my clicker will work here. So Vince covered weld failure. We're just going to talk about approaching welds for use in load transfer.

So we're looking, again, at the global level rather than the local level. So the local level is where you are concerned about weld failure. But if you're working with a larger model, you're more concerned about correctly transferring that load in your assembly.

So we've got a lot of different approaches that we can use between solids, shells, using beams and plates. But all of these methods are a little bit of black magic from time to time. And they require a lot of pre-processing. And none of them really improve the accuracy of the results in the region that you're concerned with.

So here we have a Nastran In-CAD. And we can actually correctly size our welds based on the results of doing a shell-to-shell analysis. So we can figure out our normal forces, our shear forces, and our moments. And then we can take those and do some post-processing of those results to correctly size your weld. So you'll want to align your element direction and all of your

elements in a single direction, so you can easily post-process those results to figure out your membrane forces and your moments.

So we can use xy plots to extract the elements on the bottom face there. So we can see the elements along this face to figure out those forces, and then extract those to use in our weld calculator. So David is now going to talk a bit about some of our other connector types.

DAVID TRUYENS: So another way we can work with connectors is here you see a winch. This was the [? ReMake ?] demo. You've probably seen this quite a lot. I also used it in the presentation with Wasim.

So if we're going to calculate something like that, you're probably not interested in the winch, because you probably buy it, unless you design winches, of course. But you need to mount it on your machine you make. And in this case, it's the red plate you see behind the winch. So we don't actually want to have the winch model in our analysis. So we could replace it using, in this case, a rigid connector, which you see on the right hand side.

Now there are a couple of options. And this can be a bit confusing the first time you work with that, because there is, if you go to-- if you make a connector, you can choose rigid body. And then on the other side, you can choose rigid or interpolation.

So what it actually does is the one is rigid. And the other one is applying and distributing those force. And I'll show it to you later. There's actually another way which you can work with that. And that's using beam elements. So it's somewhere in between.

So if we take a look at that, here is, on the left hand side, there's the rigid version. So you see here you have these holes. And in between those holes, it's completely straight, because it's actually a rigid block. It's a massive block. And if you look at the winch, you could consider it as a massive block. While on the right hand side, this is the rigid connector. But this is the interpolation. You'll see it will distribute the force. But it will not add extra stiffness. so we'll see the plate bending.

Now there's the other type of approach I told you about, is with connecting them with beam elements. And with beam elements, we can play around with our stiffness. So we can choose something in between very stiff or very, very weak.

So if we look at those results for displacement, we have the interpolation. so the interpolation

is adding the least amount of stiffness to the plate. So you could expect that your displacement is the biggest. And because it's a thin plate, I used just a normal linear analysis and a non-linear analysis. Because if the plate starts bending, you add geometrical stiffness as well.

So here are the results. And as you can expect, the rod is somewhere in the middle. And the rigid is the most stiff. So this is quite logic, as this is something I would expect for sure. And if we take a look at the stresses, it's a bit different. Of course, the rigid one, it's very stiff. So there where you're-- if you have a stiffener in your model, so if you have two beams connected, then your highest stress will be in the corner. So if you add a stiffener, you will just move the stress concentration from this point to where the stiffener stops.

Actually what you're doing is moving around the stress concentration. And you hope that the stress will be lower. But that's not always guaranteed. So here you have a big part, which is very stiff. So there's no stress going on there. And then suddenly you have a big jump from something very stiff to something more flexible. So if there's a big jump, you end up with high stresses.

So that makes sense. And then the rod, because it's more flexible, it will distribute it more evenly. And then you have the interpolations, so the rigid connector with interpolation, it will do a very similar job. And if you run them non-linear, all of them, the stress will become a lot less, because you take in account the geometrical stiffening of the plate. Because if it's straight or if it's bended, it adds a lot of geometrical stiffening.

And then we have a last one of ways to connect things. And that's cables. And I'm a big fan of sailing. And this one is now sailing around the world. This is a boat sailing solo, one guy around the world. And I think there are about 30 boats competing. It's really interesting to see these designs changing over and over again, because in boats, you have a lot of cables. And here they use big spreaders. So they can distribute the force even better. So they use cables.

And yesterday, who has been on the high roller? That's a really amazing example of cables, of course. So if you work with cables, they are always non-linear, because it doesn't know. First you have pre-stress. So you first need to apply the pre-stress. And then it needs to know if it's in tension or if it's in compression, because it cannot be in compression. So if it's in compression, the solver says, well, it's actually not doing anything. So we'd always have to assess what's happening.

So you can fill in those values. So if you hover over those boxes, you will see this is the cross-

section. And this is the pre-load you want to apply. And there's also, the last one is the failure mode.

Now there's a very important rule in Nastran In-CAD. Everybody knows in simulation that properties are very important. So we have a tendency to fill in as much as we know. Now in Nastran in general, I think I may say, that the rule is if you're not sure, don't fill in a box if it's not needed.

Some boxes are crucial. Of course, you need to have a cross section in these kind of things. But the last one is your failure criteria. If you're in the beginning, you always build up your analysis. So in the beginning, don't fill in a failure criteria, because you're not sure what's going to happen. So if you fill in a failure criteria and you hit that failure criteria, it could be that your solution doesn't converge because your cable has snapped. We can do that.

So the first analysis, you want to make sure that your cable is within its working area. Because normally, you don't want a cable to snap. So if the forces in the cable are higher than allowed, you should redesign your cable. You should probably use a bigger cross-section. And that would be a conclusion of your analysis. And you change the cross-section.

So only fill in the value if you want to do a snap analysis. If you want to see what happens if I go above a certain load and my cable snaps, that's something we can do. So be careful. And there's a lot of other boxes in Nastran. If you don't fill in anything, it will find it out automatically. One of them is material properties for the G value. Probably you have seen-- who works with Nastran In-CAD? Any Nastran?

AUDIENCE: Actually if you put it in, it'll give you a warning.

DAVID TRUYENS: Exactly. If you don't put in the correct value, it says, well, there's a warning. The material properties are not-- how does it say again-- are not reasonable, I think. So if you just delete that, it will take care of it. And this is the general rule, I may say, in Nastran. If you don't know, just put it on automatic.

AUDIENCE: --is to leave it blank unless you know exactly what you want it to be. If you're not sure what blank means, then [INAUDIBLE].

DAVID TRUYENS: So I will [INAUDIBLE]. If you don't need the value in the first step, leave it blank. And later on, you can always play around with those values. But start automatically is the best way. So here

is-- yeah?

AUDIENCE: Something on that slide that I think is important, because it caused a lot of confusion with us, is you got to make sure your units are consistent. When you're typing in data in that field there, usually people are going to get the area and the moment of inertia right if they specify it. But the other things, the stress units and the initial displacements units, the pre-load, force units-- and the one thing that is important to understand is that the cable itself, this thing assumes it's a solid cable. No cables are usually solid. I don't know. That would be solid rod.

That's part of the i there. You really do want something in there that's small. And it should be-- I guess, would you mind if I explain this?

DAVID TRUYENS: No, that's OK.

AUDIENCE: So essentially what it is is that the cable itself was a solid rod. The default of that i is going to be a solid. You're probably saying, why does it assume that? Well, the reason why is because it doesn't know what else to assume. You haven't told it what the cable looks like, the strands in it.

So I looked this up a while ago. and there's ways of calculating what their actual moment of inertia is. You can refer to that and then put in that value of what that i really is. So I guess in that case, leaving it blank might not be good, because the default says it's going to take the area. And it's going to calculate a radius and then use that to calculate the moment of inertia of a solid.

So if you want it to be a cable that has very little bending capability, you can put a small number in there, like 1e to the minus 5 in this case, down to 10,000 times less than your area. If you want to factor in some bending stiffness, then you have to [INAUDIBLE]

DAVID TRUYENS: So first example-- well actually 2D model might not be the right one. Actually I should have called this a 1D model, because it's a 1D model just using beams. And this is an interesting case. This is a customer of mine. And they made this mass. It's for emergency situations or military applications. They're in the field. And it's aluminum package. So it's not very heavy. You just can carry it on your back. And you can move it up. So it could also be an emergency situation where you needs a light to shine around.

So it's very light. But this is a very nonlinear thing, because on the top you have your device, a light, and then a light, an antenna, cameras whatever. So if you would run this linear, it will

just-- and you put on a load or something-- you push it in this direction, that's the only thing that happens. And there's actually no bending, because it assumes it starts everything as vertical.

But if you run this non-linear, which you should, because there are cables. But there's also another reason, because once it starts moving, the biggest weight in this model on top will do a displacement. And it will create a moment. So it will create a bending moment on these poles. So in this case, it's a really nice example, I think, of cables combined with non-linear analysis.

So set up, so there's a-- in the session together with Wasim, I showed that we have a really nice connection. Thanks, Mitch, for the frame analysis combined. So who has worked with the frame analysis within Inventor? Yeah. So you just build up your model. You use a skeleton model. And then you put in your frames. And then they'll transfer in beam elements.

Well Nastran In-CAD can do exactly the same thing. But then you can continue, because you can combine and can use. You can add plates. And you can add cables, for example, these kind of things. But one of the things, if you want to try working with this, you have your skeleton model in Inventor. You need to suppress that in your analysis.

But usually you also have your reference point. So if you're going to suppress that, and you have to use those working points to put on your constraints, they're gone. So you need to put your working points in your assembly environment. And then it will run perfectly. So there's another thing. But that's coming later.

So other considerations, if you have something like this, a construction like this, cables pulling it down like on the mast of a sailboat, there's a lot of tension just pulling the mast down. So there's a lot of compression. So you'd think about buckling.

And vibration, well what I think is the best way to work with-- but you can discuss about that-- usually you have the mast. And you have the cables. And they have very different densities. They're very different stiffnesses, of course. So I think the best way is to separate them and to look at the mass itself, the vibrations, and the cables.

Because I don't know if you have ever been in a harbor with sailboats, and all the tension on all the ropes, there's always one making an awful lot of noise. I once thought the engine was running in the boat. But it was just the noise of one of the ropes. So I would put them apart.

So there has been a lot of talk about generative design. I don't know if somebody has seen this model. This was one of the early tests from the Dreamcatcher project. And this is a connector exactly for something like this. There's a mast. And you need to attach cables.

So I didn't make it as fancy as that. But I just made something similar. So that's why these points are in different positions. And then we want to make a connection. So the first model was only 1D elements. And now I'm going to put in this connector part, where the cables attached.

So if you set up your analysis, this step you choose nonlinear static. And also make sure this, by default, the force is off. But of course, you're interested in the forces in the cable. So you can just turn it on. And once you turn it on, you will see in your results that you have the cable forces.

So this is the same model. It's a huge mast. So on the top if you zoom in, you see that one. So then it becomes obvious that you should use beam elements. You're not going to use a solid mesh for the whole mast. Or even a shell would have been quite huge. And here we can work with the cables.

So I'll show you this here. So here we can see we have our stresses. So you should see cable stress here on the top. If you don't see it, you're not running a nonlinear analysis. So a lot of people-- the general rule is start easy. And then add complexity. So usually the rule is, start linear static. Once that's running fine, add a non-linear analysis. But with cable, it's a bit different, because otherwise the cables won't do anything.

So if we go to, if you want to know the force, you go to Other. And then you'll see the cable force here. So that's also something which you can get out of the simulation if you want. So if we zoom in here--

AUDIENCE: David, we can't see the screen.

DAVID TRUYENS: I'm sorry. That's why is because it's-- how long have I been talking without it? What have I told you already? So here is, if you go to Stress, here you should see cable stress. So that was what I was telling. So if you don't see it, you're running linear and not non-linear. If you go to Other, you have cable force. If you don't see it, you didn't turn on the force in the output.

So if we're going to look at the details here, we can see at the stress. And then here we can go

to our von Mises. So you see the size of the model. If we zoom in, we'll see this part. And the thing we use to connect it, which is-- I was wondering if you said, I'm going to a cut a bolt and put in a rod, how are you going to connect them? Because if you connect a rod directly to a solid element, you're going to pull on a node.

AUDIENCE: You also can have a ball joint. You also have a ball joint, because the solid can take up rotations of the rod.

DAVID TRUYENS: Well but in this case, I have a line or a beam element. And I want to pull on a solid something. So I cannot just pull on a node. Because if you pull on a node, it doesn't make any sense. You're never going to pull on something infinitely small.

So to do this, you can also use the connectors. So if we show those, this is a technique everybody uses to connect, to go from 3D to 1D, or maybe from 2D, from a shell element to a beam element. You can just add those connectors. And then you can continue adding your beams. So in this case, here is a beam. And here are the cables connected. So it's cool that you can turn it off, because otherwise it becomes hard to see.

So here you can also see our cable stresses, to see if they're actually doing something or not. And this was very confusing for me in the beginning, because I have a force to the right. And the whole thing is moving to the left. So if you see something like this, that can't be true.

But actually, rule one, start simple. And then add complexity. So the thing I did with, I thought I just make it a bit fancy. And I put the hinges on different points. So one cable is longer than the other one. So it's a bigger spring. So it will pull more to one side. So that's actually what happens, because the force is quite low. But if you start, all the cables with the same pretension. But they are different in length. So they will pull more in a different direction. So that's also something you should consider when looking at these results.

So that's about it, I think we have. So this was an overview on how you're going to get together. Yes?

AUDIENCE: You have these cables, but it was loose forever. And most cables are straining. Can you predict first strand breakage? Because you can break several strands before you have failure.

AUDIENCE: Let me move on that. On top of this, it's a real simple cable. It just snaps. And that's it. And once it snaps, it doesn't come back. Actually do we have time? Because I wanted to say a few more things about this. Actually, I know a little bit about this. And that's really tough to do. And

the way the code works is with bolts and with cables with pre-load, it goes through a initial cycle where it actually does that calculation that Andy was talking about.

And essentially what it's going to do is it's going to go through and try to figure out exactly how much initial displacement is going to be created pre-loading the cable. Well if that base down there is not stable, if it's a pin, it can't do it. And what ends up happening is it has a huge cable nodes that are unrealistic. And the thing never converged. So that's the first thing. If your model's not converging, then the pre-loads are probably not calculating correctly, because there's some type of softness in the model that shouldn't be there.

And the other thing is that you asked about the failure. If you want to do that level of detail, you'd have to actually make these out of solids and have all the different-- we actually model cables that way, too. And in the beginning, you do the contact and everything. It's going to be a big model. But it's possible to do.

DAVID TRUYENS: So I'm just trying to get the problem. Is this something that actually happens, where you have one strand breaking and then the rest? Because then it becomes more elastic.

AUDIENCE: It becomes more elastic and has less strength. But see the issue is that you may have operational failure. Or you can have safety failures. So if you go into that, you may allow the first couple strands to break. But it won't give enough to completely fail to harm somebody.

[INTERPOSING VOICES]

AUDIENCE: So one way he might approach that that just throwing something out here, is you could, theoretically, put a bunch of these cables together. And as long as you don't worry about the fact that they're bound together, you could have it do that.

So I would imagine that you'd have a bunch of these cable elements that look just like lines. And maybe they have an [? RB-- ?] that connects to one end, another [? RB ?] at the other end. And you start pulling on it. And one of them fails. And all of sudden, the load's redistributed. And you can do some kind of progressive failure like that.

[INTERPOSING VOICES]

AUDIENCE: It's really simpler than it. Mine is you're not analyzing the cable. You've got to know in advance what the failure potential of the cable is. And you [INAUDIBLE]. If you've got the test data to show first string failure happens at x thousand pounds, then that's your model. So you're not

simulating the actual failure of the cable. It's just you know that this force--

AUDIENCE: You're [INAUDIBLE]. So with these connectors, you're really just providing a load [INAUDIBLE].

[INTERPOSING VOICES]

AUDIENCE: There was a way to di this. In the old days, for me, [INAUDIBLE] had that in Nastran, before I invented it in the Nastran, we used to just use a var element. And we would use a nonlinear stress training curve, the nonlinear elastic. And because you could do the first and the third quadrant. And you just say, hey, you have no stiffness in compression. You have stiffness in tension only. And you can model stuff.

And then you could even model, OK, it's break. It's going to break. It's starting to break. And you can model that whole progressive. Now it's lost all the stiffness. And you can do that with one element too, and just simulate that effect as well. So there's a ton of ways to skin this cat.

AUDIENCE: I was just curious. Thank you.

DAVID TRUYENS: And I think the more people-- probably are a couple of other people who have some other ideas. So the more people you talk to, the more ideas you get. But that's OK. usually you also have to find your own method, I guess. Any more questions? Yes?

AUDIENCE: On the winch example where you had a node of five pointing off the part [INAUDIBLE], how is that different than these where [INAUDIBLE] force?

DAVID TRUYENS: That's a good question. And I was thinking about that as well. But you have four holes. And I have a point here in the middle. And I'm going to, whether remote load this, it's going to take the distributive force and add a moment.

So if you look on the top, so the plate is like this. And then on my point here, I will just have this force distributed on the four holes. And I will also have a moment, which is not exactly true, because this one will pull. And this one will push. And even if you look at the top, and I have this point in the middle, it will also apply a moment.

And so these holes, they will be a torque on this direction, which doesn't make any sense. Because if you look at the winch or even a softer construction, this construction is actually going to prevent that. You have a moment in this direction. And that's what the interpolation actually does for you. It will distribute it. It will not add stiffness. But it will take care of these

kind of things. Does that answer your question?

It has been a long week for everybody. I think everybody's tired. So I would like to thank you for your attention. Nobody's fallen asleep, which is a challenge after the party and all that. Maybe Wasim did. But this is recorded. So we should start stopping now that conversation. And maybe get a beer and-- all right, thank you very much. And have a good--

[APPLAUSE]