**ALLEN FOWKES:** Go over a little bit about myself. Again I'm going to mention the title-- "Shaking All Over-- Using simulation to understand vibrating products." This is a course really about looking at all the dynamic capability within the simulation products of Autodesk. And I'll cover primarily Simulation Mechanical and Nastran In-CAD. But regardless of that, I think you'll get a good glimpse into just what all the dynamic capabilities are.

So this is Allen Fowkes. I came with an acquisition that Autodesk did back in 2009. And it was formally known as the product ALGOR. And I worked there for a little over 20 years, 22 years, and saw a lot of vibration applications. And I really got into it pretty deep.

Before that, I worked at McDonnell Douglas in St. Louis. And that's really where I got indoctrinated, if you will, into the world dynamic side. I always tell my boys I was essentially a lab rat back in the day. We had all the fun toys-- big aircraft, random vibration shakers, acoustic rooms, wind tunnels-- you name it.

So I tried to touch on all the different types of analyses that aerospace companies do but not limited at. Every single product that really people have has some kind of a vibration environment. And we're going to look at how to set that up and analyze that.

Worked with a buddy of mine, Vince Sharma, at McDonnell Douglas. He always called it, find the killer mode, Al. He used the mode shapes that I acquired. And then we tweaked the FEA model. And then he'd do his flutter analyses and determine instabilities of the aircraft. It was really about looking at what's the killer mode. And I didn't quite I understand that initially. And every time I worked with him, he was after a specific problem.

And one of the projects we did was an F-15 vertical tail was cracking. And we got the project, of course, because it's obviously a vibration problem. And that really was about getting the killer mode. We determined what that input was to that vertical tail. It was flexing in a certain way and cracking. So if it was a fatigue problem, but it really was a vibration problem.

And [INAUDIBLE] got the response spectrum, plugged that into the actual aircraft, and started looking what modes were being excited. And lo and behold, we found it and looked at what mode was causing the bulk of the stress, the flexing, near that crack. And then knowing the mode shape, we could stiffen that structure up with some additional plates that we could put on there. So really finding a killer mode kind of settled in with me. And that's really a practice

I've kind of preached throughout my whole career.

This is from my buddy James Herzing-- shout out to him. He set up a scavenger hunt. So if you guys are interested, go to that website. Like I said, you've got an email from me. And take a look at it. If you got time, why not?

Before we get into the dynamics, I want to look at simplifying the CAD model. I find that is just absolutely key to getting all your vibration analyses done. And we'll look at SimStudio tools as well. But really, it's going to be about understanding what all the different loads and inputs are to the dynamic processors. Differences between the Nastran and the Simulation Mechanical certainly. And what it's really about is understanding what the results mean to make basically your designed changes.

So your real goal is to improve your product, whether that's a grain harvester, pressure vessel-- whatever it is. It's in a certain vibration environment. And you need to make design changes that make sense to improve that product.

Like I said, it's everything from pressure vessels, grain harvesters. You know, people sit in grain harvesters for 8 to 10 hours. Vibration becomes a very important aspect. Not doing vibration is certainly an option. But usually what that ends up being is something where you got to add a lot of expensive dampening systems and try to control things from there, and that's just expensive.

I was watching one speaker. He's talking about composites the other day. And everything should be made out of composites, according to this guy. Anybody not doing composites, he's just like, that's just sad. I thought that was funny, because in a way, vibration is kind of like that for me. It's simple enough to where, again, why not do it?

We're going to look at the simplifying the model a little bit. We're not going to go crazy here. But as you can see, there's quite a room for simplifying things, not including the nuts and the bolts and the washers. Going with simple beam elements. Both Nastran and In-CAD and Simulation Mechanical have-- I don't have a pointer, but maybe I do. Yeah, I do. Centroid Creator.

These are the Simulation Mechanical menus, by the way. But the same functionality exists in Nastran and In-CAD. You have beam elements where-- I can even have beam offsets, distributed loads, remote forces, and then we have joints and bolts. And that's what you end up with a lot of times. No reason to model every single one of these in. As a matter of fact, the bolt-- call that the bolt wizard-- can even have preloads on your bolt. So we can take those into account certainly in the dynamics and in the dynamics world as well.

As I mentioned, we'll talk about SimStudio tools. And just real briefly, this is really I want to call it an analysis tool. I don't want to call it a CAD tool. It's really something that you get this if you have-- how many people in here have Simulation Mechanical, by the way? OK, quite a few. Nastran? I see other people.

So what you end up getting a lot of times in your CAD model are things like real small surfaces, real slivers. Those kill me. They just basically add up to a lot of elements. Why is that? Well, in the FEA world, we really deem it illegal to change these features.

So these curves, these edges of surfaces, we preserve. So you as the engineer or the analyst, you've made that. We don't change it. So SimStudio tools can actually come in and take your Inventor model and actually start getting it ready for FEA, getting it ready for dynamics. That's one small part.

Simplifying things again. There's actually tools in here where I basically, with a slider, I can get rid of fillet radiuses, fillet radii, right there. Some on this. Yeah, all that simplified. Kept the bolts in there, but maybe you can simplify that too. Threads probably aren't a good idea for dynamics. You get the idea.

But we also could come in and actually just delete parts and actually suppress parts. So very easy tool to use. If anybody's used CAD, you should be able to figure it out. There's really no training. Really no learning curve for it.

So I'm going to try to simplify things as much as possible. And I've got a real simple model I'll show you-- a little video on the SimStudio tools. Has anybody used SimStudio tools in here? OK. A couple. A lot of people, they don't even realize it exists. So you get it with any simulation package now, any simulation product.

So what it does is getting rid of features that-- I'm really saying, this is really ridiculous in the FEA. So the real power of this tool really comes from the fact that, as customers, you might have a lot of CAD models from even different customers of yours. And you need to get your model ready for FEA.

This can read in any geometry. And its real claim to fame is the fact that I can actually manipulate features, as you're seeing here, on the fly with a slider. I boxed the whole model in here. And what I'm really trying to get rid of is a lot of these washers and bolts. But these leads-- from the dynamic standpoint, I just want to model the mass of these parts. That's the important part. And really, with a click of a button there, they're gone.

So pretty powerful tool. That's what I'm talking about as far simplifying your model to get it ready for a finite element analysis. So that's SimStudios tools. If you're new to it, get into it. Give me a call. We can get on a net meeting back home and go through things.

A little story to tell you about the model I'm going to use. This was actually a colleague of mine. Chris Cripps over in sales has a boat, and the boat deck failed. Boat deck hanging off the back of the boat. Ended up big storm, wiped it out. Wanted to redesign it. Wanted to make it stronger, lighter.

But what he did was he actually ended up using ForceEffect. And we started with a ForceEffect model just to get the rough idea, the geometry down. So we kind of scrapped the whole initial design. It had rails on the side, and they were actually stiffening up the part. We want to get rid of that. Those are just things that are just in the way. People can trip over them, whatever. They're of no use. This is a much simpler design.

So what we did was run a lot of static stress analyses, get an idea of the displacement stresses. But we want to do more. People are going to jump on this. People are going to run on it. There's going to be vibration from the motor, from the shaft, the propeller shaft. So we wanted to take a look at all the various types of analyses. So I thought it was great model just to keep it simple, not jump from model to model. I really only have one model here. I'll show you.

But before we get going, I always keep this picture in mind when I'm thinking about doing a dynamic analysis. And these are the various analysis types we'll cover for transient stress, frequency response, response spectrum, random vibration.

Keep in mind that modal's really the prerequisite to all these analysis types. So modes get fed into these processors. So where do you have your control? You don't have much control about your input. If it's a certain random vibration, that's it. What you have control about is redesign your part to change the mode shapes, and that's the key.

The key also is understand what are the outputs of these. And I just have a couple more things to pull down. They're generally speaking square root, sum of the squares output, and time-dependent output. So what are we talking about? We're talking about determining what you're after.

If you're after initial transience in analysis, that's a time-dependent result. So engines start up. Frames get shaken. You're looking at the initial transience, and those die down. That's where you want to go-- something where maybe a steady-state response and a frequency input. Looking at a square root sum of the squares response, steady state.

So it's going to get you out to steady state much faster obviously in a time-dependent analysis. We'll get into these, but more of the same-- square root, sum of the squares. Those are the mode shapes that we're squaring. We'll get into that.

Let's talk about modal analysis first. Why do we do modal analysis? I was talking to someone earlier about a-- got a customer a couple weeks ago. They're designing a plastic welding device. And what this device does-- basically, it's a tuning fork.

And they're trying to design it so it's an extension mode of that tuning fork. And then they run it along the weld seam, where the two plastic parts come in. Basically pounding that plastic with 40,000 hertz at a very small amplitude. You're welding this thing. Bend your toothbrush back and forth, eventually it gets hot. Metal, you know, plastic melts, welds together.

The key is really, for example, understanding what your frequency input here is. And this is an exhaust muffler. And brought this in from Inventor and ran it in. You're looking at the mode shapes and trying to determine, do I have any mode shapes near my input frequency?

That could be the engine. That could be a pressure wave. So this is an exhaust. This is the intake to the exhaust intake, the outtake's on the other side there. But maybe you have pressure waves inside of that tank-- that vessel, whatever it is.

We have some people in here working with pressure vessels. Maybe there's something pulsing inside of that pressure vessel. We can input it into that vessel and get an idea of what particular modes are we exciting.

We'll jump into the software here. And I told you I was going to run Simulation Mechanical in Nastran. And just a little bit about this model here-- this is a beam plate model. Pull my plate in here as well. A pretty simple model. So beams are running. Whoops. Beams are running this way, that way. And ForceEffect kind of got us through a certain point. ForceEffect-- I don't know if you've heard of that. That's a quick force reaction kind of program up on the Autodesk Labs. It's free.

The idea here is, we're in modal analysis. And I might as well show you the setup a little bit. Get into the analysis parameters, as we call them. And what you're looking for is, how many modes am I going to solve for? Maybe you don't know that. That's really something you just have to eventually get to. Or if you know the frequency range, you can go ahead and put that in. Maybe you don't know how many modes that are in there, but you know what the frequency range is. That's the idea.

Within Simulation Mechanical, you can get into some convergence problems with rigid body modes are there. So maybe you have your part, your structure, on some dampening isolators. You're getting rigid body translations and rotations. This basically speed up your analysis and avoid some confusion for the solver.

There's Chris, that's the, actually, culprit of the boat deck.

**CHRIS CRIPPS:** [INAUDIBLE].

**ALLEN FOWKES:** How did that thing fail, by the way?

**CHRIS CRIPPS:** How did it fail?

**ALLEN FOWKES:** Yeah.

**CHRIS CRIPPS:** A windstorm on the lake, and a boat got drifted onto it [INAUDIBLE] hit the back [INAUDIBLE].

**ALLEN FOWKES:** OK. Lovely. We were talking a little bit about ForceEffects, Chris. Did you anything you wanted to mention about ForceEffect on that?

**CHRIS CRIPPS:** Yeah. I mean, ForceEffects thing, as far as figuring out how big and how powerful the cantilever [INAUDIBLE] was going to be. I mean, pop that thing up and I sketched quick little sketches and try to figure out the best way to load it. I was trying to figure out the stress on a bolt. and how could I actually make it 5 feet long? Or would my bolt be so big I couldn't do it? They use that really quickly to optimize the most efficient ways, size-wise, to start the design.

**ALLEN FOWKES:** And that's when we turned to FEA, right? I mean, we got to a point where it's like, OK, now let's put some plating on here. Let's start playing around with cross sections. And really efficient model to work with. These are the boundary conditions back here. Forget to mention that.

So I tell it how many modes to solve for and jump into the results. And what we have are the 20 frequencies. I'll go back to mode 1. And what's important here? Well, what we really didn't want was a springboard.

We can design something within ForceEffect for linear static stress. But how does it respond to someone jumping on it? As I'm driving over waves, and this thing's bouncing in the back, what are some of the modes that I'm really exciting in?

The main key for me was, we didn't want this frequency to be like 20 hertz. That's when you're starting to get into a springboard. People are going to be jumping off this, standing on it, walking on it, et cetera. So very quickly, I can tell just how good my design is from just the frequency.

And if I go up in frequency, I can animate some of these. Whoops. There we go. So the results-- you know, tremendous-- really easy-- what am I trying to say? Really easy just to look at what the modes are. The displacements are relative. So we can turn the actual displacement magnitudes on. But what you're really getting out of this, as I mentioned, are the frequencies and then what the shape is.

And if I [? look ?] up at certain frequencies, I start getting higher. I start to notice that I've got some frequencies that are starting to be near my engine operating speed, right? And that's starting to be a concern to me. So what I'm really after here is, what's my first bending frequency? And then certainly, what are all the other frequencies that I need to be concerned about because they're going to be excited to some point? That's going to be-- we'll see that that's actually one that's going to be excited later on with a different type of input.

For Nastran In-CAD, go back here. Within Nastran In-CAD, it's an environment that sits within Inventor. So I just click the Nastran In-CAD button here. And if you saw real quick, there was another stress analysis that was in that particular first ribbon. And if people don't know, if you have Simulation Professional, you get a linear static stress analysis and a modal analysis within that particular product. That I didn't cover today. But certainly, if you had questions, we can definitely talk about that.

So I'm in Nastran In-CAD now. And let's take a look at the analysis types that we can run.

Actually, that's a good thing. One of the projectures I had wasn't showing this. But this is quite an extensive addition to what the Inventor Simulation program can have.

So these are all your dynamic processors. Notice that there's normal modes. There's buckling. There's a-- called pre-stress normal modes. That's the same as-- we call it modal analysis with load stiffening typically. And then we get into some non-linear analyses.

And then we're going to into the transient analysis response. Notice just real quickly that there's two types. And then there's impact analysis and non-linear. And then there's frequency response, et cetera, random vibration.

What we're going to do-- I've run this with Simulation Mechanical. Stop that. Let's just go back to our PowerPoint for a second. Like I warned you, I'll jump around in here for the whole class. What's important here?

So we're going to input a force over time somewhere on the part. So in other words, what I simulated was someone jumping off that deck-- a 200-pound person jumping on the edge of the corner of the deck. What you get out of it is displacements and stresses over time.

We're going to see that we can plot things. We can animate the displacements over time. And again, in Simulation Mechanical, just like in Nastran In-CAD, there's two types of solvers. There's a direct integration solver, and this doesn't use the mode shapes. So we go back to that first initial slide where we saw the modal results are an input to this, not for this particular solver. I'll go ahead and show you where those are.

The key difference for me is really the damping and how those get input. So if I go to my transient stress analysis-- didn't mention it, but any of you guys that have Simulation Mechanical-- let's go ahead. Yeah, we want to look at results.

Notice that I have these all in different design scenarios. So I can easily go back and look at my modal results if I want to. Let's go ahead. I've got a lot of curves here. Let's get rid of those for a second. And just like I did in my modal analysis, I can animate it.

These stresses and displacements now mean something. These are tracking this information over time. And we'll go back and take a look at how we set this up real quick. A little hard to look at that. What I like to do is typically go with the-- let's just pick a node here. And I'll just right-click and graph that value. And this is my response. I could take it and get rid of all these data points in here, make a line. But you get the idea. Let's go ahead and unzoom that.

What I was really after is how much displacement really occurred on that initial jump. 0.04. Linear static, by the way, was 0.02. So not bad. Not bad. Like I said, it's a real-life situation. It's going to happen. Someone's going to jump off of it, drop something on it. How much displacements over time on that?

We go back to the setup on this. I'll go into the analysis type. Let's talk about this menu for just a little bit. I tell it where to input my particular load over time-- a load curve, a direction. I'm putting in the negative z direction. And I'll talk about activation time in a second.

We've got load curves. You dictate just what your time step size is. And typically, that's something that you're probably going to want to run this thing a few times. Again, that's why we want to simplify the model as much as possible. We don't want to run 20,000 steps if we don't have to.

Damping is the key here for me. This is why I picked this particular processor over the direct integration solver. This is a much more known value to me-- percent of critical damping. So I get that readily from the test data-- test world.

What mode shapes are we going to use? We're going to get them from Design Scenario 5. Let's take a look at the load curves here. And I'll make that a little bit easier to see. Let's say we go to 0.03, something like that.

So it's an impulse. Impulse over time-- 200-pound load. I defined my load curve out to two seconds just to give me some additional data out to two seconds. That's it. I just want to be sure I captured whatever the peak response was due to that, and that ought to do it. But again, that's a decision you make.

We can see that we can put in a piecewise linear curve or switch over to sinusoidal. If you're doing a sine wave and you're really after steady state response, there's no separate processors that are definitely going to be more appropriate.

And as with most of these processors, we can import the data as well. So if you've got some random vibration signal, you can break that down. Now, back to the activation time. This parameter. This is kind of handy, because what I can do is actually simulate someone walking on that deck.

We've got ladder manufactures as customers. They want to actually simulate the person stepping up a ladder, looking at the dynamics of the ladder, how unstable it is, those kind of things. A lot of bridges. Virtually every bridge that I've looked and been involved in, simulate a truck rolling across the bridge.

What's the activation time? It's OK, at time 0, I'm going to have that particular load curve going into my part. And then I could have a different time, different activation time, a different location-- the truck actually hitting another spar. So these spans within the bridge-- I can actually simulate a truck 50,000 pound, 100,000, whatever, rolling across this bridge and simulating that.

One other feature of the time history analysis-- something called ground motion. And that's what this additional optional tab here is for. Take a look at the setup. Basically acceleration over time. And we could have a constant, turn it off. So you could have something shooting down a railroad track. Cars accelerate. Even robotic arms. That's a ground rotation, like pitcher's arm. So I can simulate that and look at the dynamics of my part due to that.

Cancel out of there. And we're going to switch over to-- you guys just holler if you have questions too. Not a problem. No real order to this other than I feel like this is probably one of the most common dynamic analyses that I run across-- shock or respond spectrum analysis.

This is not an accident over here. This is actually a staged event. We're literally separating these two rocket components. And how we're doing that is with an explosion. So earthquakes. Here's a tip earthquake analysis. Shock pulses. So if I have a 100,000-pound stamping machine in my shop floor, I can determine the response somewhere else in that building. That's a shock/response spectrum analysis.

We're going to see that the results are a square root sum of the square of the modal contributions. And one of the reasons I put this particular curve in here, guys, is because it really tells me what's going on here. This is period versus acceleration. And acceleration can be either acceleration, velocity, displacements. We can convert those pretty easily. But if you take the inverse of this period, that's a frequency. So think of this as frequency.

What I'm doing is I'm actually scaling my mode shapes to this particular response. So this is really the most important curve for shock/response spectrum analysis. This actually looks so familiar. I didn't quite research this enough, where I got this. But that's probably the most common earthquake spectrum that you'll see. I believe it's called the El Centro earthquake. So it's something that was just heavily measured.

And like I said, if I have a mode here, I have a certain amplification of that particular mode. I have a mode over here. I have less amplification of it, right? So it's starting to look like, if you really did a single degree of freedom system, that's essentially what you get. So there's an amplification at that particular frequency. So we're really scaling these frequencies to this curve and coming up with the overall response. Let's take a look at that real quick.

Go back to my Simulation Mechanical. Go down to Response Spectrum. And let's take a look at the results first. A couple different results. I mentioned we have the individual components and the result.

And so right now, what I'm looking at is actually the resultant of all my 20 modes. So I have modes from 70-- whatever it was-- 79 hertz out to a few hundred hertz. And what I'm doing is I'm combining these all to get my maximum response of my model. That's the idea.

What I'm looking for is the killer mode again. That's the important thing. That's what's always in the back of my mind. I'm looking for what does that resultant really look like, and then go back and see what mode is really causing that. So if I turn that off, I'm now looking at the individual modal contributions. And I can go back to my first one-- simple blending.

And these displacements now actually mean something. Like I mentioned a second ago, we've now scaled our mode shapes to actually mean something according to that response spectrum input curve. And if we go up the frequency list, go to maybe 7, 8, I can see that 7 looks pretty much just like that particular resultant.

So what I'm doing is, with my input, I'm actually exciting my seventh bending mode. And if I'm getting cracking or maybe large displacement, I now know what mode shape to go back and look at to stiffen up. And I can actually put maybe an additional beam element along here. Drive that frequency up a little bit.

If we go back to the FEA editor and go into the analysis-- parameters. Don't do much with the output. Those are to log files. I'll make this a little more readable. Let's say 0.03-- something like that. So what I have here-- I've got my input spectrum. And usually, you don't have a lot of data points for this. So there's not a lot of input. But certainly, you can import it again, like I mentioned.

You're picking a particular type of input-- displacement, acceleration, or G level. Again, as I mentioned, you can very easily convert those, direction. And there's a combination method. And if you're in the nuclear business, you pretty much know what NRC Reg Guide 1.92 is. Well, these are all pretty similar.

If you're not in the nuclear business, then use the original procedure. It's a square root sum of the squares-- pretty typical. This just has slight variations in it for more conservative analyses, if you will, but pretty much more of the same.

A cluster factor is something that I get a lot of questions on. What is that? That's something where, if I have mode shapes that are very close together-- let's say 10% together-- and these modes shapes start influencing each other, feeding energy into each other.

So what happens is we actually, in that particular case, we sum those two modes, and then we square it. So the idea is 2 squared plus 2 squared would be 8. If I sum 2 and 2, that's 4, and square it, that's 16. Just basically make sure that you're aware that, when you see your results double, that's probably why. And certainly a square [INAUDIBLE].

This one I don't use too much-- the generate response spectrum. But if I do click that, it does activate this middle section of data here. And what is that? This is really for a single degree of freedom system. So very simple amplification curve.

So you're going to have a curve that comes up and comes down, really indicating just how much amplification is over that frequency range. Very simple formula. We don't use that too much. So maybe if you don't have a curve and you want to run something very quickly, you could do that. Again, we're going to get the modes from Design Scenario 5.

Let's go back to my PowerPoint here.

**AUDIENCE:** Hey, Allen?

**ALLEN FOWKES:** Yep.

**AUDIENCE:** Before you get off that topic, I'd love to clarify. Because this is often a really hard concept [INAUDIBLE] to grasp. So when you apply the PSD or that shock/response curve, you're not applying all those modes sequentially. The assumption is they're all hitting at the exact same time. And those waves are [INAUDIBLE] of the actual wave.

But you have to remember that, because it's not a transient input, time-based input, which

[INAUDIBLE] from, is that you lose phase information [INAUDIBLE]. So if there was a situation where shaking in one frequency up or the other frequency down [INAUDIBLE], you're giving that up for the convenience of [INAUDIBLE] problem because of the transient dynamic [INAUDIBLE].

**ALLEN FOWKES:** Yeah. Yeah.

**AUDIENCE:** [INAUDIBLE] there's a little bit of giveaway, but it's the [INAUDIBLE].

**ALLEN FOWKES:** Yeah, that's a good point. I mean, look at it as, like Vince said, you're taking all these mode shapes, and you're squaring them. So these all become positive, right? Think of it as the worst case scenario, if you will. All my modes are acting in the same direction, as opposed to offsetting each other. Maybe bending mode here is a different direction than second bending and subtracting those displacements. So it's a procedure that I'm going to stick with and look at the maximum displacement possible in my model, forgetting about time. Yeah. Yeah.

Next up on the list is random vibration. And included a curve here. This is from the test world, and that's the world I lived in for a little bit. Basically, I'm going to take that curve, and I'm going to input it into my model with that menu.

Where's this curve come from? This comes from the test world. You're not going to be able to come up with this on your own. This is typically from a random vibration accelerator or some piece of data acquisition system. You're acquiring data. You're filtering it.

So you're pulling down frequencies a little bit. You're basically trying to simulate the real-world environment the part's on-- something that's driving across the road. Boats traveling across the water could be a little rough. We can mimic that particular roughness with this particular curve.

And what this is is frequency versus amplitude. And this amplitude is in the form of-- we'll see it in a second in the menu, a little clearer. But it's g squared per hertz. Like I said, you're not going to come up with this. So very similar to the response spectrum that we just took a look at. And we're going to combine all these results into, again, a square root sum of the squares. So again, you're losing that real-time dependent potentially lower displacement result. Think of it as, again, more of a maximum result that's going to occur in this particular model.

There's a lot of filtering that goes on in this. This concept of g squared per hertz, or acceleration per hertz, the "per hertz" is really something specific to that test setup. So you're filtering things. There's a certain bandwidth for filtering, et cetera. And we're normalizing the acceleration with that bandwidth.

So again, just to make matters simple, you're generally going to have this. We were talking to a couple customers earlier, do some electronic board testing. And that's exactly what they're going to put into their board. So they're going to mount their electronic equipment onto the random vibration shaker, plug that in, and get a response. Real-life environment. Let's take a look at the Simulation Mechanical for that.

I can get rid of this menu too. We'll go into the analysis parameters again. And adjust this just a little bit. Let's say 0.05. Yeah, something like that. That's fine. And maybe a little lower. Yeah. Oh, I pulled up a response spectrum, guys. I screwed up. Hold on. I meant to get this one. And back to my analysis parameters. Analysis Data. There we go.

So yeah, we've got same kind of concept. We're going to have a combination method of all the different mode shapes. Well, first, we're going to scale those mode shapes according to this curve. This could be random vibration from the engine. This could be random vibration of the boat going over the water, whatever it is. I give it a certain direction.

This is the power spectral density curve. And I'll probably leave this for a separate discussion. But basically, there's two types of combination methods. And if I select this one, it will remind me what it is. I can neglect something called cross-mode effects.

So if I have something that is-- what's the application for that? Maybe something where there's a similar frequency. Maybe I've got some frames that are exactly bending at the same frequency but in different directions. It's up to you whether you want to include that effect. Normally, you're doubling the amount of amplitude, if you're combining all those modes. So we want to be a little bit careful there.

Again, this has a cluster factor. So if two modes are relatively close together-- maybe we got a couple modes right in here-- then we're going to sum those. And then we'll do our square root sum of the squares. If you do have a part in maybe a 45-degree angle-- maybe you've put your electronic piece of equipment on the shaker table at a 45-degree angle to the input-- that's what this is for. So we can put it in through the x and y direction.

Go over to the results. Very similar to the response spectrum. But here, you're going to see that a different mode is actually being excited. So we take a look at the resultant. It now looks just like mode 1-- a little bit different.

So in this particular case, I can see that I actually am exciting a different mode. Again, I'm going after the killer mode. What mode is that? Take a look at the resultant. Undo the resultant. Take a look at what mode that is. And that's your killer mode-- 79 hertz. So in response spectrum and random vibration, I can easily go back and take a look at what's really causing my problem.

Now, switch gears here a little bit. Go back to my PowerPoint slide. And something called DDAM analysis. And this is only available in the Simulation Mechanical in the interface. You could certainly run this out in Nastran In-CAD, but you will have to edit the input file for that.

But what it really is it's the Navy shock spectrum analysis. So we've got some boat manufacturers in here. If there's an explosion out here, what does that do to my parts that are in that boat? Let's take a look at a simple example, where we had a particular enclosure around this whole part.

And what we were after was really just we had this large mass over here. We didn't stimulate these vertical bars. We didn't care about those. Those were pretty beefy, and they go into some other beefy equipment there. But we [? really ?] didn't model the motors but just that vertical plate/shell model.

Basically, this method is a little different. This spectrum is calculated internally. So if you are doing DDAM analyses, that's the benefit of them. There's equations built into the software that go ahead and compute that input spectrum that I showed in shock/response spectrum analysis.

A little bit difference in the combination methods-- not really significant. We just are sure that we use the one that is spelled out in that particular analysis type. That's short for Dynamic Design Analysis Method, guys. Let's take a look at that input real quick.

Go over here. Come down to my DDAM. And you see similar information here. If I can enclose my model. There we go. And again, I can turn off my resultant. There's our killer mode again. And I certainly can step through them, but look at my resultant. Not much difference there, guys.

We'll go back and just show you the input. Just get you familiar with it real quick. Basically you're telling it surface, submarine, type of ship. Mounting location. Attach to the deck, attach to the hull. Attach to some plate, some steel plating. And there's some additional options if you want to include plasticity as well. So this is all-- everything particular to that particular DDAM analysis.

For people that have their own input spectrum, you can go ahead and just go ahead and tell it to read that. It'll grab those from a file. Secured run. There are certain people that don't need to have their input spectrum written out to log files.

So that's what we call a secure run. We can just read that spectrum in, but we don't have any memory of it. We toss that out afterwards. So it only resides in that file. But basically, I tell people, make sure you don't zip everything up and send it to me. That's the key there.

For a couple different solution options-- never quite understood this one entirely. But I think I finally figured it out. There's really a couple ways that they calculate the input spectrum. And one is based on velocity only or acceleration. We're picking by default velocity or acceleration. The idea there is that'll grab the most conservative curve-- in other words, the highest amplitude curve for you and put it into your structure.

There are some additional options in here. And this is true of Nastran, In-CAD as well. I can actually go in and tell it, you know what, don't include modes that are less than 10% from a modal participation factor. So in this particular case, in this analysis, you start to get concerned about, have I solved for enough modes in my structure?

Why is that important? Well, if I only solve them for, let's say, four or five nodes, I'm really exciting my structure with some input random vibe or could be even response spectrum. The idea is that you may not have enough modes to really see that accurate maximum displacement in the model.

What DDAM does have is a requirement for 80%. So that's our default for here. So that's always happening by default. And we can certainly tell it, in that particular case again, don't include modes that are less than 10% modal participation factor.

And lastly but not least, this is really more for people like myself that don't know what aft forward ship means and fore aft, I can put it into a certain direction-- x, y, or z. So you can use either one. And we looked at the results, OK.

So back to my PowerPoint. The next dynamic analysis we want take a look at is called a frequency response analysis.

**AUDIENCE:** [INAUDIBLE]?

**ALLEN FOWKES:** Yes. Yeah. It's otherwise known as harmonic analysis or frequency response analysis. And the output is in SRSS a little different there. This is an SRSS of our in-phase and out-of-phase components.

We're going to see that we can use this particular analysis for-- if I've got a piping system, might have pumps and compressors in it. Could have a typical skid that goes onto an offshore oil rig, put some equipment on there. These things get turned on, left on for quite a long time. This is giving us our steady state maximum response. So that's what you'd want to think of it as.

The other processor we looked at earlier was a transient stress analysis. So over time, this will get you at a steady state much faster. Of course, there's definitely advantage for looking at initial transience again. As I mentioned, transient stress is the way go.

One of the other things we can actually do for this or use this for is sign sweeps. I see that quite often with people if they're trying to match-- if you hook a shaker up to a particular part and then you increase the frequency-- let's say from 0 hertz up to 1,000 hertz or 2,000 hertz-- you're going to get a transfer function.

Simplifying your model, again, is always important. In this particular case, we used our beam elements to locate the center mass of our motor that's mounted onto this particular assembly. I'll talk about this curve. This is really coming from fluid flow. We'll take a look at that. So this is a very common analysis in pressure vessels, piping systems. And as I mentioned, really things like a motor sitting on here and it's just turned on for infinity. And we want to take a look at the response of that.

I had a slide in here because I was having troubles earlier displaying the menus for Nastran In-CAD. But these are the differences. And the main difference between Nastran and In-CAD in Simulation Mechanical is more the setup. I'm going to get into that in a second. But really most, if not all, of your information is in the analysis parameters within Simulation Mechanical. We'll get into what this is in a second, once I pull the software up.

But over in the Nastran In-CAD side, generally speaking, you're putting in a little bit less information-- the main time-dependent information. But things like load curves are going to go into your load menus. We'll take a look at that.

One note-- you can put in a range of exciting frequencies. So I can put a range in as well as a range-- I'm talking about a range from F1 and F2, and I can put that in. The nice thing about Nastran and In-CAD will actually go ahead and compute and an input additional frequencies in near your mode shapes, which is kind of a nice thing.

So if I'm really trying to really get a good handle on what that peak is, if I have more frequencies that I'm inputting, that's going to help me out-- get a little more resolution in the mode shapes and the natural frequencies. You really can't do that in Sim Mac-- Simulation Mechanical-- unless you generally import the load curve. Let's take a look at that.

So I'll come back here to the setup. And under my analysis type, again, here's our menu. And what I can do is put this particular frequency in at a certain location. So I just pick this in my model. In this particular case in the boat, I put in the back end. This is where the shaft is probably going to be, somewhere in the middle.

So what I'm going to input I'm going to input at an exciting frequency into that particular location as a force versus an acceleration. As a frequency index, we're going to look at that in a second in the Exciting Frequencies tab. And we also define our damping ratios and amplitudes.

Go over to the damping, or the exciting frequencies. I'm putting in my 450 hertz, which is what my motor's operating at. Or that's maybe what the shaft is spinning at, the propeller shaft. This is actually a newly designed menu for 2016, guys. So if you're back on 2015, it's going to look a little different.

Same thing. You've got a cluster factor for it, again, if a couple modes are very close. We sum them and then we square them. Same thing with damping ratios. And what I'm doing is I'm putting in a couple points, and the software is going to linearly interpolate between here and determine what the damping ratio is as well as the amplitude for that input. So 350 pounds. I'm just bracketing my excitation frequency.

So I go back to the exciting frequency. You'll see I can put in a whole range of frequencies. But at some point, I probably am going to import it if there's that many. Go into Excel. Generate a big list of frequencies and import them.

So that being said, let's take a look at the results on that, guys. A little different now. Now, I've only input one frequency, so I'm only getting one load case out of this now, as opposed to the 20 for my response spectrum. What am I looking at? I'm looking at the in-phase component or the out-of-phase. And we'll take a look at either one of those by going up to the response type and clicking in-phase or out-of-phase, or I could pick the square root sum of the squares.

Now, what's the benefit of this? I don't get a lot of benefit from the in-phase and out-of-phase. But it lets me do is put in a frequency range, for example. And then I can plot-- let's say I have 200 frequencies. Then I'm going to get 200 load cases. I can actually plot that over time-- not over time, but over frequency. So I can have amplitude versus frequency and get my curve.

I want to take a look at the frequency response in Nastran In-CAD for just a second, guys. Let me locate that direct frequency. Yeah. So what we have in Nastran In-CAD are two different types of the frequency response analysis. One's a direct integration solver, and the other's a modal superposition type of solver.

Direct frequently is not going to require you to run mode shapes-- solve for mode shapes. Modal will. What's the difference? You'd have to run them. You're probably not going to see much difference. My opinion is you're going to pick one and stick with it. If you use modal frequency response for 10 years, you'll probably stick with that.

**AUDIENCE:** Allen?

**ALLEN FOWKES:** Mm-hmm?

**AUDIENCE:** The biggest difference is in number of frequencies contributing to your response. If you've got hundreds of frequencies, the modal method is going to be faster. If you've got--

**ALLEN FOWKES:** It's a speed issue.

**AUDIENCE:** --dozens of frequencies [INAUDIBLE]. That's really the breakdown [INAUDIBLE].

**ALLEN FOWKES:** Yeah. Yeah. Again, keeping a simple model. You're not going to see the difference. But take a look at that from a speed issue. Absolutely. I'll just leave that there for a sec.

And one other thing about the frequency response, or just a slightly different application-- we've got an exhaust muffler. And what we actually run is a CFD analysis. So we can compute the pressure waves in here. So let's just say, for example, we have a pressure pulsating wave going into this particular vessel, or this exhaust. The CFD program will actually compute that.

And I don't know if it's built into CFD yet. I know it's a free app on the Autodesk apps page. But what I can do is take that time data-- so pressure over time-- and actually perform an FFT of it and get that frequency, and then go input that frequency into my structural analysis model.

That makes sense? Maybe another application is, I had a customer the other day call me. He's like, can you do this? This guy builds towers. And basically the wind's going around him. And he's looking for vortex shedding around that tower.

Another customer has the same thing with pressure vessel-- large, tall pressure vessel sitting on the wind. Could have vortex shedding. Run a CFD analysis. Determine that frequency. So I can take that. And what I do is I typically look at drag force or pressure difference.

So I'll actually see that sinusoidal pressure difference. And then I can bring it into this particular application, which, to my knowledge, is outside of CFD right now. But you get it and then do an FFT-- a Fast Fourier Transform-- and get the frequency that I'm interested in, then input that into my structural model.

So that's pretty common. I think if you didn't know every single light pole in parking lots or on the roads, that's what they're concerned about. So that's a very common analysis. It's called the Strouhal number. That's basically your vortex shedding frequency. You can hand-calc it pretty easily, but that's a nice app. I'll check to see if that's actually in the software yet. I'm not going to get into that. So that's frequency response analysis.

Take a look at buckling now. And there's a couple different types of buckling analyses. There's a linear buckling and nonlinear buckling. Linear is going to give you things like the mode shape. And I call it a mode shape because it's an eigenvalue solution. It's an iterative analysis, just like modal analysis is.

It's determining modes that this part will buckle in. And it'll tell you at what load multiplier this will buckle in. So if I have a 10-pound load on my part, and my buckling load multiplier is 2, my part will buckle at 20 pounds in this particular shape.

Now, we also have nonlinear buckling. We can look at the onset of buckling, post-buckling. We can include material plasticity in here and actually look at the whole response all the way through the buckling and beyond and, as I mentioned, include material nonlinearities as well.

Geometric stiffness is definitely updated. That's what's actually changing this curve. So what is buckling? Buckling is a geometric stiffening changing thing. Your stiffness is changing during that loading process. And we can capture that in two different ways that we'll take a look at.

Take a look at the linear buckling first. Jump into the results. There we go. And what I did in this particular case is I put a load out here. I don't know why I'm not seeing it. There it is. Got to turn them on.

And we'll take a look at how my part buckles. And you see, I've got five modes here, but I'm really usually interested in the first mode. My buckling load is 350 pounds roughly times whatever that load is. So pretty high load in this particular case. We look at 500 pounds. This is if Chris is backing his boat up and bumps into another boat or into the dock. We can look at just how that part's going to buckle, what load will it buckle at.

Displacements don't mean anything. So don't believe those. It's the relative shape of this part, and it's that multiplier you're after. I could look at additional modes. Definitely a much higher buckling load multiplier. It can buckle a little differently at that load. I can just keep going up and up. You get the idea.

Take a look at the setup real quick. This one's an easy one. Let's go back to the analysis parameters. And basically, it's going to take all the same loads that your linear static analysis has. It's got some load case multipliers for them just to make it easy, if you wanted to double the pressure and take a look at that. But usually not necessary here. But maybe I want to double my pressure but keep my thermal load the same.

I can, again, bring in forces from my CFD model-- pressures, temperatures. Actually, I'd have to go through the Thermal tab to pull it in from CFD perhaps. Or I could set up my thermal load on my structural model right within Simulation Mechanical.

Same thing with fluid reactions. CFD file. Under Solution is really where I tell it how many modes it's going to buckle in. Like I said, a lot of times, I don't look at too many modes higher than mode one. But if you're doing towers-- big frames, cell towers, whatever-- perhaps you're going to have a few modes that are very close to each other. And we want to understand what those are.

In that particular case, it's just not one overall buckling, perhaps, of our tower. It might be individual members that are buckling, and we want to take a look at that. If you don't know how many modes to solve for, you can give it a range of the load.

Go back to our PowerPoint here. And we'll talk about nonlinear now. But let's talk about it a little bit before we get into the model. Now we can include contact. So now we have a dynamic analysis that can include contact.

This is an animation. This was an actual part we worked on where it's actual pipe cutting. You're basically just cutting your pipe. Involves some buckling, and it certainly involves some contact. And to impose that motion, we've just put prescribed displacements on it.

We can track these forces on that particular knife over time. And that was the goal of this analysis. We wanted to take a look at just how much load with this particular knife design-- is it going to bucket? Or not buckle, but what kind of force is going to be required to shear that pipe?

And we'll go over to the Sim Mech here and get into the nonlinear design scenario. In this particular case, I modeled this a little bit differently. I reoriented my part. And you can see here I have what's called an impact plane. Where's my impact plane? Right here. I'll just edit it just to give you a quick look at it.

There's a Drop Test Wizard in Sim Mech. You can do the same analysis in the Nastran In-CAD. Typically model that a little closer to the impact plane, define initial velocity, initial acceleration. Can do the same thing here, but I think Sim Mech can handle really the dropping of these parts, the free falling a little bit better.

Here we just tell it what's the plane. Again, I went back and reoriented my part. And now, if I go to the results-- well, let's take a look at the results first. Then we'll come back to take a look at the setup. And I'll just animate them here.

You're looking at a drop test. So we do drop tests of phones. We just had a customer do the same analysis with their cell phone design-- you know, their case for the cell phone. Kind of tends to zoom in on you. Just ignore that. It's like a self-automated zooming in based on the orientation of the part.

And again, I can take nodes. And actually, let's see what we're looking at here. Displacements. I can take a look at von Mises stresses. That would be just important in the plate model. And if I just pick a particular corner, I can go ahead and graph this. Where'd that go? Over time. Why didn't that-- let's take a look at that again.

So various graphs over time you can use to take a look. And maybe you want to go back in and maybe refine the analysis, more time steps in that particular zone. We'll take a look at the input for that. We go back to the FEA editor and the analysis parameters.

Similar to the transient stress that we looked at before, you're going to have a force somewhere on the model, pressure on the model, perhaps. We're going to have a load curve. But here, what I'm doing is I'm defining how many steps I'm going to run in the analysis.

Now, not to worry here, guys. There's automatic convergence. There's automatic time steps. Things start getting close to the impact planes, and things start happening. Stiffness is changing. Stresses are happening. It's going to go ahead and automatically drop your time step. So I gave this is more of a setting based on how many steps I want to see for the free fall. I don't want 1,000 steps just to look at a free-falling object is the idea. Not really useful.

But I go over to Load Curves. Maybe I should go to Gravity tab also real quick. I defined gravity in the minus z direction. And the load curve is used. Here's the load curve for the gravity loading.

And if I go back to the load curve, it's just a flat curve. So zero out to two seconds. And I just I just have a multiplier of 1. We can certainly import the load curves as well. Again, this [INAUDIBLE] analysis type will also read in temperatures and pressures from your CFD analysis.

One thing I typically always recommend you do-- this is kind of a last-minute thought, I know, for a lot of customers-- make sure you output all your time steps. So when it's actually dropping and impacting the plane and dropping a time step, you go ahead and output those as well.

All right, guys. Make sure that's the last one. That was the last dynamic analysis that we had. You guys OK? You have any questions so far? Yes, sir.

**AUDIENCE:** Around the DDAM analysis, there's no way to just manually input the scale and the weighting factors [INAUDIBLE].

**ALLEN FOWKES:** That are in the DDAM manual that are calculated?

**AUDIENCE:** Yes.

**ALLEN FOWKES:** So the question is, do you have to-- state that again.

**AUDIENCE:** So the DDAM analysis uses a number of coefficients, weighting and scaling factors. And other [? packages ?] will allow you to actually specify those coefficients if you want [INAUDIBLE] research document or whatever [INAUDIBLE]. Is there any way to do that in Sim Mech other than Import File?

**ALLEN FOWKES:** No, you've got to import the file for that.

**AUDIENCE:** What's that file format?

**ALLEN FOWKES:** That's an ASCII format. That's just a comma, space separated-- nothing exciting. I can always dig up some examples, if you get back home. Give me a call. Yes?

**AUDIENCE:** Actually, if you set one up, and you simply export the Nastran, you can see those coefficients being written up in Nastran [INAUDIBLE].

**AUDIENCE:** And it's the same order in terms of [INAUDIBLE] copy that [INAUDIBLE].

**AUDIENCE:** Yeah, I mean, Nastran file is a text file, right? So you can [INAUDIBLE]. Or if you go [? to our helpline, ?] [INAUDIBLE].

**ALLEN FOWKES:** Yeah, we'll show you the calculations that are done, certainly. But right now, we just bring in the coefficients.

**AUDIENCE:** But the ones we have in Sim Mech are the declassified numbers. [INAUDIBLE], when Navy uses them, they have their own file. And they go into a [INAUDIBLE] that is not connected to the internet. And they run it only on that machine. You can't bring your phone. You can't bring anything [INAUDIBLE]. They have their own way of running things.

**ALLEN FOWKES:** Yeah. If that's something that you use all the time, I mean, give me a call. We can talk to developers. And that's the product manager, [INAUDIBLE]. We can look into adding that too.

**AUDIENCE:** Well, and the reason why I asked is because there was a client I was working with that wants that secure document [INAUDIBLE], and also would like to be able to tweak those coefficients.

**ALLEN FOWKES:** Yeah, in my opinion, if it's part of DDAM, we should have it in there. And if it's something that people are adjusting, that should be fairly easy to do, since we've got the equations built in. And we just have additional menu for those options. Yeah. I'll take a look at it. Give me a holler too. We can get that input.

One of the things that it's going to bring up, but kind of reminds me right now, is the IdeaStation. And I don't know if I recommended on that particular option. I think that's better than just a quick idea. But normally things you're running into that are real pains in the you know what. And you're like, I'm doing that a lot, and maybe it'd be better if we did it this way. You can fire off. They go right to him.

[CHUCKLING]

I think so, right? Yeah, all the products of Autodesk now have that. That's a great feature, because it's always tough. I come from the support world too. And I just never had enough time to really talk to customers on the phone and in person about, what would you really want in the software? So that's a great addition. Use that. That's on the web as well. OK. Any other questions on the dynamics that you might want to go back and take a look at? OK. Not a problem.

Mentioned the differences between the Nastran In-CAD and Simulation Mechanical. We can set these up and drop-test manually. But there is a menu for it, just to let you know. Basically, I'm inputting a impact plane, and I'm giving it a certain direction. All that's actually automated with this particular menu. This was put in a few versions ago.

So if you are doing drop tests all day, every day, that's the kind of stuff that we like from-- you know, ideas from customers. So those things that come to us we'll put in. So that's one of them.

**AUDIENCE:** Can I do part-to-part impact, or is it--

**ALLEN FOWKES:** Yes. Yeah, you could have part-to-part. The Simulation Mechanical has got, as well as in Nastran In-CAD, pretty robust contact. Yeah. So you're sliding contact. So you're outside of the linear realm real quick. It just can't handle that. But yeah, that is one thing that really makes the Nastran In-CAD and Simulation Mechanical much more robust than the linear stress. Put it that way.

Couple last slides for you guys. Feel free to jump in. What I would do is I would go to the-- and it's not quite on this. Yeah, I'd go to the Knowledge Network. I didn't put the actual website on here. But if you go to the Autodesk page, go to the Knowledge Network. And then you can get to things like the forums, the IdeaStation I just mentioned. So here, someone wanted to expand the hyperelastic material library.

Forums are going to include users, developers, technical support people, technical specialists, et cetera. We have a lot of lot of webinars, a lot of videos. So I just keep it that simple, guys. We try to keep these short and sweet. We used to do eight-hour ones. Those are pretty brutal. These are, you know, 20 minutes for this. Maybe some additional meshing help.

Maybe I'm interested in contact. There's a specific webinar on contact. I know that. YouTube as well. Again, go to the Knowledge Network and then search for help webinars, for example. A lot of these guys are the support engineers-- Andrew Sartorelli, he's here this week.

SimHub-- another great resource. This was really the initial video storage and forums and IdeaStations and everything with simulation. It's expanded beyond that, but we still have a lot of great content in there, like white papers.

You're going to get a lot more in-depth things. If you really wanted to get pretty deep and dirty into contact, you're going to have a white paper, for example, on that. Blogs. Again, all the AU stuff is going to be on here and lots of videos as well.

So just to summarize, guys, what we've looked at, what you're really getting out of this class is really understanding how to simplify the model-- using plates, beams, some of those other tools we looked at-- SimStudio Tools.

You can see how important it is once you look at all the different analysis types we could run just on this DAC model. We can look at really any kind of environment that our design is going to be exposed to and know what the inputs are, know what the outputs are-- key differences there-- and really use those interpretation results to make your design change that's going to make your part better and knowing why.

So with that, there's our happy customer.

[CHUCKLING]

And you can tell this is a pretty good looking design at the end. I mean, it actually looks like this thing really came straight from the OEM, the manufacturer of the boat. I mean, it ended up really looking nice. Tied in pretty well back here. These were pretty beefy, as I mentioned. And getting rid of the guard rails and everything-- really nice look and sleek design.

So with that, guys, thank you. And questions?

**AUDIENCE:** Yeah, could I ask you [INAUDIBLE]? Is anyone here using Dynamic Analysis molded body inside of Inventor? Does anyone know there's a rich body, molded body, tool inside Inventor [INAUDIBLE] [? Don't people ?] use that?

**ALLEN FOWKES:** What's the name?

**AUDIENCE:** It's called Dynamic Analysis.

**ALLEN FOWKES:** Dynamic Analysis, yeah. Yeah, motion-- yep. Right.

**AUDIENCE:** Potential is a little confusing, like this is not that, right? But where I was going with that is, it's important to remember that that's a rigid body solution. That things are moving relatively slow-- slow acceleration, slow rotational acceleration. Rigid body assumption comes into play. And as soon as you have increased speed or impacts-- [INAUDIBLE].

**ALLEN FOWKES:** Flexibility, yeah.

**AUDIENCE:** And if you find you're in those environments and you've got impacts in Dynamic Analysis, Inventor Dynamic Analysis. Or you're trying maybe [INAUDIBLE] increase speed or [INAUDIBLE] you may want to consider taking that mechanism-- at least portions of it-- into a nonlinear transient either in MES or inside of In-CAD [INAUDIBLE] transient and explore whether there are flexible body effects [INAUDIBLE]. [INAUDIBLE] see that [INAUDIBLE].

**ALLEN FOWKES:** Yeah, I have some good examples of that. But they're probably a little timely to pull up. But we do have kinematic elements within Simulation Mechanical, the nonlinear portion. So I could have some parts that I deem very rigid. Maybe I'm interested in this connecting rod, but I can give a crap about all the other parts. Maybe make them kinematic analysis elements, save time, and just focus on that one elastic body. Yeah. That's what Vince is alluding to. You can pick and choose what you're after there-- dynamics, rigid body dynamics, and/or elastic dynamics.

All right, guys. Well, that's a wrap. Any more questions? OK then.