WASIM YOUNIS: OK, morning everyone, I'm Wasim and I've got my colleague David here, we're going to go through this session. And I'd like to thank everybody, I know it's a first class, probably still awake, I imagine. OK, just before I begin, who is familiar with Inventor simulation? OK, got a handful of you here.

OK, so this is introduction class but we've got the Shape Generator in here, and we should go through the agenda just quickly give you an idea of what's happening. OK, so this is a plan. We'll try to keep to our timing but it never works, but we'll have some time. We'll wait for the Q&A at the end, that's probably the best thing to do. So basically, I'm going to go through introduction and then my colleague David is going to go through the innovation to show you some exciting technologies and applications. And then I'm going to go through-- we've got two examples we're gonna discuss Shape Generator. I mean, anyone familiar with Shape Generator, inside Inventor?

OK, and then basically, I have a couple of the customer examples who have implemented an Inventor simulation and how they used it. So maybe it might give you an idea of how you may want to use it as well. And then we've basically got some tips, just a handful of tips to go through, if you never used it before. And then we've got some more information, we have a series of classes after this one here, so you might want to just give you a little idea what's happening. And obviously, Q&A. So I'll get David to go through the innovation.

DAVID TRUYENS : All right, so the innovation we're going to talk about is mostly the things we'll see, the trends we'll see in the generative design area, or the tools we have at all to this now. I don't know if you've seen this example, they're building a bridge, or they're going to build a bridge in Amsterdam, and it's completely autonomous, by robots. So while they are start building the bridge they have to support themselves, right? So the links are in the PowerPoint, if you want to go there, there's a real nice movie.

But this is a very new way of 3D printing, actually. It's just welding on top of each other. So this is one example. And these are industrial applications, which are going to be built or are already in production.

So this next one is something I get very excited about. This is N-E-R-V-O-U-S design, of N-E-R-V-O-U-S Systems, and they have come up with a system where you can generally design a dress for fashion, right? I'm not interested in fashion, but suddenly I get really excited about fashion, with these kind of tools. And the crazy think is, you can-- they have a website, you can go to the website if you want to design something for your girlfriend, you can do that, you can just-- in the end, it's 3D printed.

But it's a quite big piece, right? So they invented a tool, like proteins, they fold. And that's exactly the same thing they do, they fold it. And then it comes out of the box like this. So if you're interested, I mean, you should go to the website. I really get excited about fashion these days, with these kind of things.

[VIDEO PLAYBACK]

We are committed to reducing greenhouse gas emissions by 50%--

[END PLAYBACK]

Of course, you may have seen the example of Airbus last year, on the main stage. So this is also some really exciting technology. Airbus is a very innovative company. And this year they launched their a-340 in carbon fiber, which is a huge step. So it always used to be aluminium and now they make it in carbon fiber. It's a breakthrough for the industry. But it's a very innovative company. They really know what they're doing, right?

And now they wonder like, what's the next thing? And then they started talking to Autodesk and they came up with this bionic panel. I mean, it's a really nice movie, I'm sure you will see it. But they started talking to Autodesk and they came up with this design. So it's a generative-designed bionic-- they called it bionic panel. And it's very interesting because we went from aluminium, to composites, and now it's back to some kind of aluminium, which is quite interesting.

But the approach is very different, because we always were taught in our-- when I was studying at school, we were taught the KISS principle, keep it stupid simple, right? That worked for an awful lot of time. And now, suddenly, with these kind of tools, you have to completely forget about it. Because if you keep on thinking this way, you can completely forget about 3D printing.

Because if you say, OK, we have made this piece for a long time, and everybody is talking about 3D printing, so maybe we should consider it as well. So you go to a company and say, how much would this cost, to print something like this in metal, right? You will be blown away by the price because with CNC milling you can do this quite cheap, and then suddenly it becomes incredibly expensive. So you have to completely change your design approach, and you have to go full ahead without optimization, and you also have to combine technologies.

Because maybe you can have it a functional model, maybe so you don't have to have an assembly anymore. Or I also compare with the story of Apple, where they had their unibody design, they were very proud of it. And they said well, we have this piece of aluminium and we grind away 90% of the material, right? And then we have left with this very nice piece of computer hardware. Well, even if you would print that, it would be more expensive. But if you would combine a lot of functions, because it's still milling right, so you cannot have cooling channels, or any kind of these kind of things. So that's where you completely have to change your design process.

OK so those were those were industrial applications which are actually happening now, right? They're going to mount the panel in the Airbus very soon. It should be in production now. And now I would like to talk about the tools we have at this point. And one of them is a shape generator, and we'll talk a lot about that later in this presentation. Other tools is Dreamcatcher, if you are at the registration desk, there there's this-- how do you call this? This caravan, what's it called, airstream? Right, so that's the Dreamcatcher project. And you want to do some feedback you can book a session there, and you can see how the tools work. And you also have this model of the drone structure, it's there as well. You can see it.

So this is something which is not yet commercially available, but soon we will see some tools coming up there. Then there's the Within technology for lattice optimization. So this is the example of Under Armour. So what would you do with it, it's something you can combine with the shape generator, because the shape generator gives you a shape. But for stress, stress is always on the outsides, right? I mean if you would like to be bending, the biggest stress is on the outside. So the best thing you can come up with is a very thin structure, where you have all the material on the outside. But then you have an issue, because on the inside it will collapse, right? So that's for 3D printing you can combine it with lattice structure and you can see where could you optimize, and all these kind of things.

So it's a really exciting technology. And now you might wonder, what's in it for me, right? Because these are very fancy examples and I'm sure 3D printing will be important in all industrial branches, but if you now do something with metal construction a lot of these tools might not be applicable for you. And that's what we'd like to talk about today. That it does have some really-- things you can use, right now. So that's what we-- what we will start talking about.

WASIM YOUNIS: Thank you, David. OK, so basically we're going to talk about Shape Generator in this session here, and the idea around Shape Generator is conceptual. So basically you create a piece of block, you apply your boundary conditions, and then it creates an optimized shape. That's the idea behind it, it removes materials which doesn't take any strain. And then, basically then, you can redesign your product based on that shape, and then you can take it further inside Inventor and do some stress analysis or optimization. That's the idea behind it.

Now I'm simply going to go through a simple example. We've got two examples we're going to demonstrate. We've got the cantilever, a simple example, and the distributor. I'm going to go through the cantilever example, show you the technology, how it operates, some basic workflows, and then David will go to the user interface to show you how it actually looks like, and what it actually can do.

So let's go through the cantilever here. So basically what we've got here is a simple, solid volume. And basically we've got constraints applied at one end, and then we've got a simple load at the other end. And basically the way the software works, it sits inside the Inventor and the key thing here is the material. This is how the software works. The material will be assigned inside Inventor part environment, takes it across into the environment. And you can see on the right hand side, you can optimize the mass based on a percentage value, percentage reduction, or you could actually specify a target mass value.

And so basically that's whatever you can define but you have to specify a material, that's how it works. Now based on this one here, you can see this is a basic shape, it's pretty crude, that's down to the mesh. The finer the mesh, the more smoother shape it's going to be. But the key thing to remember here is even though it asks for a load, the load has got no resemblance on the Shape Creator, it's simply a value.

We can actually-- if you need 100 newtons, or 100 pounds, since we're in the States. I do apologize, it's all in metric, so we'll try to say both things here. Now the idea behind this is that if you're going to do a stress analysis, then use the correct load, because you can transfer the study into stress analysis. The rule is, just apply the load which it is going to withstand.

Now you can take it one step further. And that was a crude shape. Now within Shape

Generator you can actually define symmetry. So you can define symmetry about the three axes. Here we define symmetry about the two planes. And then we get a slightly different shape. Again, it's pretty crude, but you can get a smooth shape. They do take a long time to run. So basically you get a different shape here, it's more symmetrical, and then on the next slide here, we can do another thing here.

Because what happens is, when it's actually removing material when it has zero strain, I think, that's how it works, we can specify preserve regions. If you pick a face and it's got a constraint on there, it's got a force, it will retain that face within Inventor. But here we can preserve volumes of blocks, which it won't touch. So everything in the light blue color, it will retain, and on the rest of the material, can be removed. That's how it works.

Now you can see the shape here is very, very refined. That's because we defined a very, very fine mesh. So it does take a long time, but we can see the results you get from that is a lot smoother. Now cross-section view through that object. We can see that the shape is pretty hollow, you can see the preserved region areas are preserved. The shape is hollow, so which might dictate that, why not try we could make a hollow shape. So let's make it a shell material.

So this example here is, rather than having a volume mesh, what we've got here is a shell model with some thickness. And it can still optimize it. What it does, it takes, removes the material from the shell here. So based on some criteria-- though obviously this example here would not preserve the regions. This basically is down to 27% mass reduction. And then the example on the right hand side is virtually taken a lot of material out of there. So basically you can see, depending on what criteria you set, you can get different shapes. There's no one single shape in the analysis. You can basically create different parameters, different configurations, different designs, and you pick and choose whatever you desire.

This is basically the three things in there, you need to have a material, you can define symmetry planes, and preserve regions. And that's the main idea, and the loading constraints is basically like stress analysis, if you guys are familiar with that. OK?

Now we're going to go through in a bit more detail. So we can go through distributor example, should probably get a better idea how the software works, and we're probably going to take the results into stress analysis to validate and optimize.

DAVID TRUYENS : Yes. All right, so this was a customer in the automotive industry, and it's the radiator, in front of the radiator, you have blades. Because in the wintertime you don't want to have cold air running over your radiator, so there you can control-- it's the first step to control temperature. And this, the thing in the middle, I call it the distributor, is connecting all the blades, right? So one of the blades is attached to an engine, an actuator, and if it spins this blade, and then all the blades are connected together, so they have to move along, right? So there's only one motor directly on the blade, and then this distributor makes sure they all move together.

All right, so we start with a very simple box because this is our bounding area, right? And I made small pins on the outside because they're necessary, but there's also a different reason. Because I want to have my constraints and it's a general rule in finite element analysis, your constraints have a big impact on your analysis. So the smaller the area is, the more freedom you give the Shape Generator, right? So if I would just apply a load to this face, that's the worst thing you can do because then you block a lot of options.

So in this case I made them quite outside of the structure. I added the symmetry plane, which is actually not really necessary because it's actually an unequal load. But for production, it's easier if it's symmetric, right? And again, as Wasim told, the force does not have an influence on the shape. Because the only thing we-- the only goal we have right now is a mass reduction. So if you say, I want to reduce 50%, well you will have a 50% reduction in mass. If say, I want 90%, it probably come up to the shape which is a bit crazy, because it's not connected or anything anymore, but it will reduce 90%, right?

So this is the first start and then it returns you something like this, right? So the nice thing about this is if you want to go for an optimization, and you run-- you would start with designing something, you run a stress analysis, you see, OK, these areas, there's not a lot of stress going on, so I could change it's geometry, right? If you would do that traditional approach, design something even to run [INAUDIBLE], say I can change something, go back, go forward, go back, it will take you off a lot of loops to come up with the shape like this, right?

So this is what I really like a lot, you start your design-- and I've been preaching this for about 10 years, I've always said you should start your work with simulation. Which always sounds a bit crazy. But finally for dynamic simulation it was all very true because you could work with sketches very fast, very easy, but now we can start out design with simulation, with Shape Generator, right? And we skipped maybe 100 iterations. If you would do this manually, it would take an awful lot of time.

But now the next question we ask is, how am I going to manufacture this, right? Because of course you could 3D print it, but even then you probably have to do some adjustments. So the next thing we're going to do is choose our production-- ways of production. So the first thing you could do, is you could actually just change it to a shape which you could mill, right? Very easy. The next thing you could do is-- it's a plastic part for automotive, so probably it's a plastic part, you're gonna go for injection moulding. And the design would look, on the outside look quite similar, but on the inside it's a whole different ballgame, because for injection moulding you need to work with a certain thickness on the outside, and then the stiffeners they have to be a bit less, because otherwise you get sink marks. There are a lot of design rules in there.

And then there's another technology which I think is really exciting, because all of these things, like the example Wasim just showed, all the material is always on the outside. That's always the best for a light structure. So there's a technology, which is called gas-assisted injection moulding, right? And you can have that with fiber reinforced materials. So what you do is you fill half of the part with plastic, and then in these spots you're blowing gas, and the inner part of the plastic is not yet frozen, right? So it freezes on the outside, so the hot-- not yet frozen plastic will be pushed by the gas, and you end up with a hollow component like that.

This is a very complex process because if you put too much pressure on the gas, you'll have a blow through, the air just blows through the plastic and you cannot fill it. So it's a very, very tricky technology. But the nice thing is we can completely simulate something like that with Moldflow, and we can optimize our parameters, and we can come up with a shape like that. And that will actually work pretty nicely.

And there's the next thing which we can do, is 3D printing. But then again, it's a very different design approach, because you would like to use a kind of lattice structure, right? So in this case, we have chosen injection moulding. So as I mentioned before, we have this shape, which is a very good start, right? But actually I don't have a clue if it's going to work or not. Because I just said, reduce this amount of weight, and the only thing I'm interested in the first case is, I want to have a nice shape. Because the mass reduction actually doesn't-- it's not a real mass production, because this is a lot lighter, because it came up with the solid block, right?

So here I've removed-- I mean maybe this is only 10% left of the material from my original solid block, right? So I don't have a clue if this is going to work or not, but at least I have a design which I can manufacture. So the next thing we're going to do, is another trick. So how do you get to the shape? You just promote it and then you have a baseline, and then you use your classical Inventor tools to create your geometry, right? And the nice thing about Inventor, of course, is that it's parametric. So we're going to use those parameters in the next step to do the optimization.

So this is a design. So in the next step what we're going to do is-- and this is movie playing-- we actually would like to know the real loads on this component, right? So what we did in this case is we used dynamic simulation, and here you see the blades and the forces, so actually what we have done is do the C of D analysis on the blades. And we have the drag and a lift coefficient, in function of the angle of attack. So we just run a couple of angles, and then we use this data as a force input, and then we have a graph of the forces on the distributor. And we can slate different time steps.

In this case, the highest force is the only interesting one, because the force is always in the same direction. If it would change directions, then we probably should check multiple points, because it's always the biggest force which is a critical force. So once we've done that, we can use the forces from the dynamic simulation, we can use it directly onto our component.

So another thing we need is-- another thing you would like to do-- well, this is in general best practice of Inventor if you want to do a parametric optimization with dynamic simulation, it's always good to create a level of detail so you can really isolate this component, it rends a lot faster. So here we can choose a parametric optimization with the loads coming from our dynamic simulation. We can choose time steps, and we can choose the level of detail. So this component is completely isolated, right?

And then the next thing we're going to look at is which parameters can we play with? So if we're going to open this component, we can play around with the parameters, because it's just a parametrical model of Inventor. So what we can play with, well-- of course the wall thickness is something which is really easy to play with, but then I just came up with the rib structure, right? I just thought, well, I think this component, these pins, they need to be supported better. So I came up and I thought, well, probably ribs in this area could be interesting. But I actually don't really know if it's needed.

So here are our parameters, and we can play with the thickness of the component, that's usually a very easy one to play with, right? The thickness of the component. Then we have our wall thickness, and then we have some different options for ribs.

So the next thing we're going to do is we'll go back to our stress analysis environment. One of the cool things is that the Shape Generator is just actually just a different stress analysis. So you can always switch, if you start with a Shape Generator, you can switch, and you can reduce the forces and the constraints. So what we're going to do here now is run all those configurations, right? And then the next thing we're going to do-- so this is again, a kind of a generative approach. You just let the computer do the work and then the computer will also help you to find the best design.

So once you have run all the-- so here we're going to put in the parameters, which is my design range, right? And then we're going to run the analysis, and then Inventor is going to help you to find the right event. Who has done a parametric optimization before? All right, so this is a tool which is already for a while in Inventor, and I think it's really powerful. Because the computer does all the work for you, right? So here it's running.

So in this case, we have 2 times 3 times 3 so that's 9, it's 18 different options. Which is not a lot actually, you could easily have 100. So if you, for example, run 100 different models, the next question is, which is the best one, right? So we want to have, if you want to go for optimization, you always want to have the lightest one. So then you can add your target. So here we're going to add a target, or constraint, or a goal, and we say OK, we want to reduce the mass, we want to be as light as possible. When then it's going to put all the sliders to the left, because it's the thinnest. So you don't have to run 100 analyses to come up with that answer.

But then there's also other criteria of course. So you don't want your stress levels above a certain limit. So we can enter it as well. And usually in most mechanical designs, as a start, you have three criteria. You want to be as light as possible, and you to have your stresses under control, and your deformation within a certain range, right? So that's what we've done here, and then the computer will say, OK, this is the best design, right?

So if you enter value, zero is not possible, but if you enter a certain-- in this case one millimeter is OK, that's within my tolerances, it will tell you like, this design says, this design is, make sure that the stresses are not too high, the deformation are within the limits, that it's the lightest one of those 18 in this case, right? And then you can send it back, because the simulation is always your play environment, so you can play around with those parameters, once you say, this is my perfect design, you can send it back to Inventor and then you have created a shape like this. OK.

WASIM YOUNIS: OK, David.

DAVID TRUYENS : DAVID: I have my last one. This is injection moulding, part two. You also should be able to create it, right? So this is a mould design within Inventor. Who has ever used that? It's a really powerful tool. You can design a complete mould. You can also do an injection analysis to see-- because injection moulding, you actually want everything to be filled at the same time, because if this part is filled sooner, you will get warpage and all those kind of things, so we can do all those kind of things. So we can actually go from a conceptual shape to production. I think now it's your turn, right?

WASIM YOUNIS: Thanks, David. OK so what we've got here, I hope that's given you an idea of what the technology is all about. So I'm going to go through a series of examples of how people have used Inventor here in the work environment. So I've tried to use different examples. So let's just start off with the first one here.

This is a company called Destec Engineering, they're involved in the oil and gas industry, refinery type of thing. And you can see we've got here, a clamp assembly here, so one of the things they wanted to do in here, what they want to make sure that the bolt pre-load or the torque, is sufficient to create a seal between the blue seal and the pipes. Now one of the things we cannot do, bolt pre-load in Inventor. So we found a workaround to see how we could achieve it. So basically what we've done is, there's 8 bolts, the model is symmetrical, so basically what we did was, we basically created a quarter, or 1/8 of a model.

And this example here is just to give you an idea-- I mean, there's different ways of doing it. We tried to achieve it with half a bolt. So basically, based on some calculation which I'll show in a minute, we worked out-- using the American Petroleum Institute Industry, worked out the load for the equivalent torque. That's how we worked it out. So once we work out the load to be applied on the half a bolt, which you can see here, the rest of the boundary conditions are very similar. So the key thing was to apply the load here.

So once we got the bolt, load, and the boundary conditions, we wanted to look at the results, interrogate it, and find out whether the blue and the green component you see on the screen had a contact, no gaps in it. That was the idea behind it. So basically they made use of Inventor with a clever way like this. And one of the things, I'm not sure if you're familiar with this, has anyone used ground planes inside Inventor? So this one, the other way around, basically we've got the full reflections and full images, so you can see on the right hand side there, you can actually use ground planes and reflections to be able to see the whole complete bolt. Because we can't do symmetrical results display in Inventor at the moment. And this is one way of doing it.

Another example here is GKN Land Systems Ltd, they're involved in motor sports. And they basically wanted to analyze the propshaft and yoke example in there. And basically the design criteria is basically high yield limit and a moment. Now this one again, was a little bit tricky, because if you can see on this next slide here, the component they will trash is that on the right-- on the right hand side, that little piece. There's no need to analyze the whole prop shaft. Which meant that you can't play a moment on the component. So we suggested they create a dummy component, apply the moment on the dummy component, and then transfer the moment that way. So basically, these examples. And then that basically gives some sort of stress results.

Now the stress in this example, maybe if you're not that clear, is inside the component. Now, we can't slice-- we don't have an option in Inventor to slice a result. So what we do here is we basically use the Inventor command to do section views. So we did a half section view. And then we could see the results inside the component. Now, obviously when you do that you might realize that your results of going hollow, just because you're meant to only to the surface mesh. But the results are correct. So you can see there, the results should be symmetrical around the circumference of that perimeter of that component.

And you can make use of principle stresses and normal stresses, so we can plot another graph there, just to get an idea. There's a little hotspot there, but the stress should be similar around the perimeter here. So basically you use different stress results to get a better idea of what's happening here.

Now this one here is Aerospace Design Facilities. They're involved in designing the structure to connect the cameras. This camera is used for filming. This is in UK, Pinewood Studios, and the ultimate goal here is that when the helicopter is in the sky, they want to make sure that there are minimum vibration of the camera at a certain speed. I've got 393 revs per minute there. And the design restrictions is that the whole structure you see on the right hand side, can cannot exceed 50 Kg, and the fixing positions are more or less all fixed. So they are limited in what they can do here.

So basically the way-- we want to do modal analysis, we want to find out the first dominant frequency. So how do you work that from the rpms, divided by 60 seconds, and that's at 6.55 Hz. That's what we've got to avoid. So if you look at the first set results we get 6.97 Hz. It's pretty close, it's not critical, and then if we-- that equates to 418 rpm, so we're OK with that, but it's pretty close to the 393 rpm.

The next dominant frequency is because the helicopter has got 3 rotors there, we just times it by three, and that's the other frequency we have to avoid in the analysis. And again, we're OK, it's not critical. So the idea here is that, as David mentioned earlier, is that we're going to basically optimize the design so that we can increase these frequencies so they're not within 10% or 20% of the dominant frequencies. So you can do this using the parametrics. So basically the original design was four millimeter thick-- or I think two millimeter thick, and we've increased it to four millimeters.

So basically by changing these, we can then have a look at what's happened here now. Now obviously it's increased to 7.78 Hz, so what's that equate to then? 466 rpm, so that's pretty safe, so we can operate at 393 rpm, 466 is pretty fine, but that-- if we operate at that speed, that's when it starts to vibrate. It's called resonance. And then we're OK with the dominant frequency. So these are different examples.

Does anybody do modal analysis? Wow. 90% of the users I've spoken to use stress analysis. Modal analysis is only very rare. But these aerospace industries, the automotive industries, make a big use of the modal analysis.

OK, this one here is a company called Croft Associates. They're involved in transporting hazardous materials. Very dangerous stuff, and they basically want to make sure that the base of the lorry or the container, is strong enough to transport 200,000 newtons, I think it's one ton, and they've got to make sure that the yield stress of the material does not exceed 66% of the yield material.

Now this material here, example here, it's all sheet metal. And Inventor does a great job with a couple of seconds, maybe minutes if I'm honest, converts everything into shells. Now what this basically means is that you have two choices inside Inventor to mesh something. It's solid volumes, you can do that, it'll take a long time to run. Or you can simply press a magic button, that's what I refer it to as, it converts everything which is thin, based on some standards, into surfaces. The blue corner block's a solid block, so we leave them as volumes. So we can combine volumes and surfaces in the model itself.

And then basically the results display is exactly the same whether it's shells or solids. What you see here is, we've got a high stress at the corners. Now based on the loading that may suggest it may be under compression. So what we can do here in Inventor is we can look at the first principles, and you can see the stress as compression. Now typically when we're analyzing metal, they don't tend to fill in to compression because the yield limit is very, very high. So what we're going to do here is we can look at the first principles, and you can see the stress has moved to different locations right bang in the middle, and it's a lot lower.

So this is the tools which are available to you within the stress analysis environment. People assume the stress is very high, but it may be in the compression, so you can change the results to first principles or even the normal stresses. And this is just basically giving you a display of the displacement plot of the whole container. OK.

Has anyone used frame analysis inside Inventor? Beam analysis? Now this is an example from Planet Platforms. They're involved in the maintenance of the aircrafts in the aerospace industry primarily. And they want to analyze-- the goal here is to make sure the structure, the key main primary structure, is strong enough for two guys and their maintenance kit. And they've got some standard design criteria, safety factor of four, some displacements.

Now the key here is that if you want to use frame analysis inside Inventor, it has to be-- the structure has to be created using cut and center or frame generator. So basically what happens here is that once you go into the frame analysis environment, you press a button, it converts every single structure into lines and also called as beam elements. And everything else which is not standard goes into transparent mode, invisible.

And then from then on it's pretty straight forward. Play your load and constraint as normal, there's beams, and you can get displacement results and stress results. Just like the stress analysis environment. So what you can also do inside frame analysis, it's very, very easy to pick all the key structures, and pick and choose what material you want to change it to. You don't have to go back into the Inventor environment. This is a library, the Inventor library which you guys create. So you use the same library. So you can use it within the frame analysis environment, and you can customize it. So basically you want to decide now whether aluminium, or aluminum, or mild steel is better.

So you can try it here first, then you can go back and change it. So let's make it-- so let's change it to aluminum. And you can see the deflection goes higher, obviously because it's more flexible than mild steel, and then the stress is in the same location. The stress won't change, it'll be in the same location. Whether it's higher or lower, you can check that one out.

OK, now this is what I like about frame analysis, is that you can, bending moments-- shear force diagrams-- when I was college, it used to be a nightmare to create these. I wish I had something like this, because you can create complicated shape-- bending moments, and then you can look at every single-- the one thing it won't do is, you can't create a report for every single member. We have a product called Robot which will do that, but this is only per beam. What you can also do here is the plot any of these details on the actual model itself. So you can actually visually see what is actually going on as well.

OK this is the last example here. Now this is interesting from a point of view, because the goal here is we want to make sure that the structure and the panel can withstand 50 miles per hour. Now we can't apply wind speeds inside Inventor, so we have to convert that into some sort of pressure or force. So basically, there are different guidelines based on where you are, but basically there are information on the World Wide Web, how you can convert that.

So, basically, the idea right here is when-- first of all the first thing you've got to do here is, you go simplify into the main structure. And again that is done using a level of detail representations. And then what you can do here is, the main column is converted into surfaces, and the structure behind the panel is still kept a solid, because the structure there is not a uniform cross-section, it's tapered structure.

Now the formulas on the right hand side here, it's an example from the World Wide Web and Google, and obviously some companies, this company has a whole lot of documentations, whereabouts they are in the UK, and they have to convert wind speed into force. So this is a simple example to show you how we can convert the wind speed into forces, and then once you've got the force value, you can then apply that as normal on the structure itself to be able to look at displacement results. You can see from here the displacement will be maximum, away from the column, because least rigid. And then you can look at the stress results here. So the software doesn't care whether it's a solid structure, or whether it's a shell, the results will be displayed on the whole component itself.

OK, so now I just wanted to sort of talk about some guidelines of how, based on best practices, of what you could do. Now I don't know-- I have come across a lot of customers where they simply have stresses which are way beyond the yield limit, and the first gut reaction is, it's failed. Now the trouble here is that in Inventor, we can only analyze material onto its linear behavior. Which basically means in English, it has to be below yield limit. Now if you put a super duper load on it, it's not going to follow this wiggly curve here, it's going to go and follow the dashed line. Which means that the stress is going to keep on going up, and up, and up, and then you can see that red line there, so the stress arrow will keep on going higher and higher.

Now, it's the biggest headache for a lot of customers. If the stress is below the yield limit, then everyone's happy. As soon as it goes up there, they panic, and they say, well, it's going to fail, let's beef it up. The whole point of using analysis is not to beef it up, it's to go the other way.

So basically it requires a lot of intuition and experience. If you make these designs, you'll have an idea. It's localized stress, it's not going to affect it. Now obviously, I'll mention this one again, we have softwares where you can actually analyze that, and see if it is localized or not. But what we suggest is that-- make sure the stress is below the yield limit. Then the software will give you realistic results. In other words, the safety factor has to be above one. That's the idea behind it.

Now I have supported a lot of customers in the past where they just put a very fine mesh, run one simulation, and they look at results, if it's below yield limit, they will say it's passed. Now what I would recommend based on best practices is that you should have at least three sets of simulation runs, and then look at the stress values in the area of interest, and make sure that it's within 10%, as a magic number. It may be 10%, may be 5%. If the value is not within 10%, then I would suggest you go and put a finer mesh on there and see what happens. But run a minimum or three simulations at least.

And then, within Inventor we can refine the mesh, whether it is globally, so the whole thing changes uniformly, or we can have the ability to just refine the mesh in the local regions where the high stress is. And then also we've got something inside Inventor called the automatic convergence. And that basically allows you to set the criteria to three iterations, four iterations, let the software run it, and then hopefully, touch wood, fingers crossed, it comes back and it says the results has converged. So you have two choices here. So you can either-- I call this process the manual convergence, or you can use the automatic convergence, one within Inventor itself.

Now this one I thought I'd mention, we've got a simple tube here, and you have three options here. You can mesh that using solid elements, which you can do, and you get a stress result here. Or we can actually convert it into surfaces, and get the stress result, or even in beam analysis, because the cross-section here is very, very-- it's uniform across the length. You can't read these here, but in an ideal situation, something like that you would do as a beam analysis. And then you can do a simple hand calc just to combine-- these are-- I'm not suggesting that you should use one or the other, they are there based on best practices, you can choose whatever you prefer.

OK, now these suggestions here are strictly based on my personal experience, here. Because what happens, if you take that piece there, for example, and you go into Inventor and say, right, find pin bodies, there's a button there. If that ratio is one to 200-- what I mean by that it is one mil thick, and 200 millimeters long, it will find it. If it's less than that, if a ratio one to 50, it won't find it. But it's up to you, if you want to do that or not. OK, so here I would recommend that a ratio of one to 50, you should use a shell model.

Now long components, this is something that here, what I suggest here is that if the diameter ratio here is typically 100 times, then I would recommend you use beam elements. Now I still see a lot of our customers still using solids because that's what they're familiar with, that's where they're from, to just press a button and it does it. OK, but basically that's what are my personal suggestions, but as I said, the software will take a value of 250 and then find it. If you run the simulation, what will happen is, Inventor will give you a warning, saying that, are you sure you want to treat it as a beam or a shell, and you say yes. You carry on, you say ignore the warnings. I'm sure you do, OK.

OK, so I think we are running a little bit ahead of time, so we'll have sort of plenty of time for Q&A. Now what I want to mention here is that we have a series of classes here. Now this is the Inventor simulation class. Now what I've got here is a list of common applications which customers want to do, but they cannot do inside Inventor. And the typical examples are, bolted connections, now we did find a way around it, but in some cases it's a little bit difficult to do a bolt simulation within Inventor.

Buckling, this is a very common scenario here, is that basically this is for long, thin, slender components. We cannot form buckling. You might do a stress analysis on it, and it might say it's passed, but there's no way to find out whether a component is going to buckle or not. Obviously thermal stress has never been there. I'm not sure whether it will be there. That is a very, very common request from a lot of customers who want to do some basic thermal analysis, but they cannot do it.

Fatigue, a lot of customers at the moment do hand calculations, they want to find out-- and this is something where you have a scenario where the load is very repetitive, and they want to do fatigue. So I think most of the customers who use Inventor basically use hand calculations. And obviously you want to drop something, if you drop something, you want to be able to find out what it is going to permanently deform. So these are sort of-- the list is horrendous, but the these are the most common requests we come across.

And then non-linear, it basically means that whether the material is going to go beyond yield, it won't follow the straight curve, it's going to follow the wiggly curve, here. So this is one way to find out whether the high stress you guys come across in the Inventor stuff, whether it's realistic, whether it's that high or not, you can run as a non-linear analysis. Or you can simulate large displacements, the best example I can think of is a fishing rod, which is pretty flexible. If you tried to analyze that inside Inventor, results would be very, very-- displacement very small. If you do it in something like in nonlinear software, the displacement's going to be very, very high.

So these are sort of applications, and then this is a list of other applications we've come across, where people want to be able to do all these. So the option is to do a workaround, or hand calculations, or intuitions. And basically what we have got, we've got something called a simulation boot camp, this class worked out nicely, for the first one. We have a class after the kick off at 1 o'clock, and this is basically the part two version of this class, here. We're going to use similar examples, here. We're going to basically go through what else you can do inside Nastran In-CAD. And we've got simple examples show you what sort of things you can do, so you might be intrigued the results you might get from this.

And then, I think the classes, we have some spaces available, but if you find that it's full, we have a hands-on lab around 3 o'clock. So I think we have plenty of spaces on that one, I gather. And I would recommend that, it is very similar to Inventor, it's an opportunity for you guys to just try it out and see what you think of it. And then in the other class here we're going to go through a lot of other classes which are going on, so if you're interested in fatigue, heat transfer, that type of thing, just go through them classes as well.

Right we how quick are we? May be talking too fast. OK, so we've got 10 minutes or so. Any questions, anyone?

[INAUDIBLE]

The way Inventor works is that if you specify all the loads simultaneously, they will be applied simultaneously.

[INAUDIBLE]

OK, any more questions, anyone? OK, well.

[INAUDIBLE]

I think everyone's probably keen to head off, but we can talk afterward if you want, any more questions. Anyone asking any more questions? If not, I would recommend if you go, it helps us, how you found the class, if you can provide some feedback, it helps us to make it better for next year, as well. And thank you very much for attending, guys.

[APPLAUSE]