PRESENTER: Another nice example could be with tools for communication. The first mobile phones, very big and expensive, and now we have small, very accessible devices in your pocket. So we all know what happened with computers and mobile phones. They are now pretty accessible. So my little laptop is there, I can use it without any special training. The mobile phones are for everybody.

So I guess you know what I'm going to say. The same happened with the simulation. Now we have simulation for Fusion that's really easy to use, gives you instant results and it's really accessible. Accessible for multiple prospective. First is how you get to the tool. You just go to www.autodesk.com, click the link, and in a moment you have the software up and running.

It's accessible from the way who can operate it. Well, it's so easy to use that every engineer and designer can use it. And it's also accessible from the price perspective. So it's just $300 yearly subscription, so it's 10 times less expensive than these packages. It's two magnitude less expensive. Just to give you a perspective, that's the difference between spending an hour in a cab and spending an hour in a private jet. So that's how accessible the simulation is these days.

And I'm here today to tell you more about the Simulation for Fusion and show you some examples of that. Well, the evolution continues. And the next step, at least as we believe at Autodesk, is a car simulation. And I will give you some preview of this technology by the end of the class.

What are the learning objectives for today? First I will go through key concepts of Simulation for Fusion that will help you to use the software more proactively because you would understand the key driving ideas behind the software. Then I'll talk about the linear static stress, the most commonly used simulation, and do an example of how you validate this, check and find out if it would transfer the load that is needed.

I will talk about modal, and about how to avoid critical fragments and do an example of that. With thermal, I will describe how to use it and show you how to fix a problem with over-heating. And for thermal stress, I will show how we can optimize the design and make it smaller, but still perform as we want. And at the end of the class I will do a preview of the cloud simulation technology.

So let's start with the key concepts, with the key driving ideas behind the software. What is Simulation for Fusion? Our mission is to simulate behavior of design in various scenarios. In other words, we are an alternative to experimental testing. And that's a core idea of digital prototyping. Instead of wasting your time and money by doing multiple physical prototypes, you can iterate and find the optimal solution using the digital prototype.

But simulation for Fusion is not only about finding if the design would work or would break. It's also going to give you the directional guidance hot to improve it. So the software leads you to the good design decisions. And of course, you have part of Fusion 360 theory data collaboration experience.

So there are a couple of key concepts, loads, constraints, contacts, studies, and results. And these click together to simplest ablation. So we'll start with loads. Loads define forces, moments, and so on applied to your design. Loads are actually causing stresses and deformation of your design.

And you have multiple type of loads. You have to have force, force, pressure, mobile-- doesn't talk about your design. Remote force, this is the force that is acting outside of your design. And for thermal, you have applied temperature that describes the temperature over given surface.

So if it's heat that describes the heat that is generated by the square units of the surface, radiation, convention. There's a cold cooling mechanism of transferring the heat between the surface and the liquid or gas. And internal heat, this is the energy is generated inside of your body.

Let's make a quick example of that. So here I'm going to apply-- quickly allow. So I'm clicking through icon force structure load, selecting the surface. If you think that a magnitude, let's say 1,000 newtons-- and clicking OK.

So quickly, I did the force here. It's very self-explanatory. Icon in the browser, you see a representation of it. You could make it more complex.

So let's go and edit this for us. I'm going to double click it and show you a couple of more things you can do with it. First is that you can select multiple targets, so multiple services that the force acts on. Or vice versa, you can limit the force to change the path of the surface.

So if I will limit target, I can use this little manipulator and eliminate the force just to this part of the surface. And I can also move that to better describe the problem.

Backup with other options, how to specify the direction. Perpendicular to the software, using angle. In this case, you can use these manipulators and change the direction using the angle import.

And vectors will help you to define xyz coordinates for the force. And the last one is the reference angle, when the value can select any entity, and the force is going to be alighted with the selected entity. And similar, there are a couple options for the thermal out.

So for thermal out, I'm going to work with the convention in this quick example. And you want to make the convention using all these surfaces. There is multiple surfaces. So it will take a time to select it. So it is a very useful option to select all faces.

And you just click there. It selects all spaces. And then you can just unselect the faces that shouldn't be part of it.

Next concept is concept of constants. So constraints limit displacement of your design. Constraints describe connection of your design to the rest of the world. That means that you need to make a decision what you are simulating and what you are not simulating.

So for example, if I'm doing a simulation of this computer of the lid of it, I can just model it and then put a bend constraint on the bottom of it. Or, I can say I need to simulate more. So I will make a mobile of the whole computer and put a constraint to the bottom of the computer.

Or, actually, I may want to also calculate a deflection of the desk. So in that case, I will need to draw a model of the desk and then put a constraints to the bottom of these legs. Or, maybe the vibration of the floor is important for me as well. So in this case, I will need to mobile the floor as well, and put constraints around it.

Well, but you will need to make a decision. Then you want to stop and eliminate what you are calculating and in what month. And that's what constraints are for, to define a connection between your design and the rest of the world.

And by the way, it's a good practice to make that as small as possible. So in my case, I probably don't want to calculate the vibration of the floor as a part of the analyzers of the computer. We have multiple types of constraints, fixed one, that's like valid or good, bent or rotation, frictionless, that's about slipping two surfaces.

And how I described displacement is very similar to fixed. But it's also accessible to what would be deformation. So there is a force deformation in the fixed constant area.

It's pretty easy to set up a constraint. You click to the constant icon. And then you select one surface or actually multiple surfaces, to put out a constrain.

You see the icon for a constrain. And you can have it full effects, or you can actually remove the degrees of freedom in some axises. And then the icon changes as well.

It's similar for a bent type of constrain. But you have the ability to have it free for tangential infiltration, movement along the axis, and try it out.

AUDIENCE: [INAUDIBLE]?

PRESENTER: You can import whatever design you wanted to into the Fusion through this step or the other known trial format, or you have a good collection through Inventor that's smarter and will help to rate even some features inside.

AUDIENCE: [INAUDIBLE]?

OK. Next is concept of compacts. Compacts, define connection between components. So if you have multiple components, you need to define how they interact with each other.

And if you have multiple type of contacts, the bonded contact, you're going to make sure it's glued or welded together. Separation without sliding allows-- the surface is set but without any sliding. Sliding, no separation means that there is a free sliding, but the surface needs to stay touching each other.

And the last one, separation with sliding, that's a combination of it. So it can slide, it can separate, but the materials can't penetrate each other. And because there are a lot of contacts in your design, there is automatical generation that help you to do that very quickly. And we also have a bigger degree of freedom analysis to help you identify missing contacts and constraints.

So let's take a look how it works. I have this simple assembled multiple components. I need to put contacts between them to be able to solve it.

There's a nice view called groups view that helps you to define missing contacts. So now I switch it on. And it's all grey. It means mobile, no contacts. And I can start adding that. So I will click to bonded. And the software offers me to do the contacts automatically.

I don't want to do it right now. I want to show you how to do it manual. So I'm saying no.

And now I'm selecting two bodies I want to contact together. So I'm selecting this one and that one. As soon as I do it, it isolates the view nicely. So now you can go and select the first surface. And it highlights the second body. And you should select the second surface.

As soon as I did it, you can see that now it has a blue core. But the rest is still in the gray. So they are still missing contacts. But now I will use automatic contact generation.

And you can see that almost anything was bonded together. But this one is still not. So let's take a look on what happened.

And you can see there's a clearance here, so the body's handle touching each other. So I can do manual contacts, or I can actually take advantage of settings here. And there is a contact allowance that tells you when these things will be bonded and will not.

So I'm going to increase the tolerance, let's say to two, and automatic contacts again. And now you can see everything is blue. So everything is connected together. Contacts are presented here in row. So you can see for the last two, you can see the two cores that are indicating where the contact exactly is.

And you can see that these have bonded contacts. However, this is not a good place for a bonded contact because it can rotate. So I can adjust it. I can select these two, and click to edit contacts, and change bonded to sliding. And I'm done with contacts.

AUDIENCE: [INAUDIBLE]?

PRESENTER: Not yet. That's the idea we are talking about for a long time. But we didn't unify the joints in the assembly and joints in the simulation because they are very similar stuff. But there are also differences. So this is still in front of us.

So next is concept of studies. Study contains definition of simulation and results. And studies allows you to do multiple analysis on one design.

So let's say you can do the static stress and modal or one modal. Or, you can do multiple out cases. So for example, you can put multiple types of loads, certain moments, or defined magnitudes. And all of that can be managed by multiple studies.

Well, by the way, this shows how dynamically they are developing the software. In some release, we introduce the static stress and modal frequencies. And then in the autumn release there was the volume mesh.

And now in the winter release, you will get the thermal and thermal status that I'm going to demonstrate as well. So this is the newest addition to the Fusion family of features, and pretty quickly going. So you can expect a lot from simulation team.

And the last concept is concept of results. So results allow you to explore calculated stress, strain, deformation, and make decisions about your design. In results, you will find answers to your questions, like, what is the stress in my design?

What is the deformation of my design? Can I remove my try to optimize my design? So you are finding your answers here in the results.

And the basic idea is that the model is covered by colors. And colors means numbers. And there is the legend that maps the numbers to colors. That's how we communicate the idea about stresses being inside and around the surface of your body.

So let's quickly take a look at it. Now I'm in the results, and I can see the legend here. So for example, I can see that blue means 35 to 50 or more.

I can use [INAUDIBLE] to actually take a look inside to my model and eliminate some parts of it from this pipe perspective. So for example, I can do a thresholding and show you just pieces of the desire that I loaded a lot.

So these are the pieces that may be programed in the design, or the design at least is value-- so the materiality, the transfer in the stress. Or, I can do from the other end, and show you the part of your design that I actually over designed it. Because there is-- the safety factor is really high. So you can maybe remove the material in these parts of your design.

Or, you can actually show both. So you can show the combined view of the pieces that meets your attention because of-- and to design it, and because of over designing. So these are like the areas of interest of your model.

Well, the legend also lets you to switch between different types of results. It's kind of interactive control to take a look what was calculated. So you can switch to stresses. There are multiple types of stresses, displacements, and strain.

So these things now clicks together to very simple simulations device. So you start with creating simulation. Then you create a study.

You may review the material. You set up your constrain. You put out a load. And then you will see the results. So this is very basic waterfall from beginning to end to do the simulation.

And I can quickly show it. So it's easier if you click through the simulation, and create a simulation. You create a study.

And you can see that the icons are aligned. So you will step by step in the toolbox. So now you click through the materials.

You make sure that the materials are right. You put a constrain. Next, is load. Let's say 3,000 Newtons. And you're good to solve.

And in a moment, you have your results calculated. So this is really easy to use too. Every engineer designer can use it to make a decision about a design, and make a design better.

Now, let's talk about linear static stress. This is a very commonly used simulation. So let's take a look at what it can do for you. The linear static stress, calculate displacements, and stresses, strains, and reaction forces caused by applied loads.

There are few assumptions like small deformations, both deflections and rotations, and lineal material behavior. This actually is most commonly used simulation. Out data shows that about 80% of users are using lineal static stress, even like the very specialized users that simulation analysts pay a lot of money for, bigger, expensive packages, they they are still using linear static stress. So this very useful type of simulation.

And I'm going to do a quick example. I'm taking this jack. The function of this is to elevate the car if you want to change your wheel. You are using this one to elevate it.

So you are using the handle to move the screw. Here's the car. Here's the surface. And it just elevates it.

And my goal is to verify it would work. I know that it should elevate about 600 kilograms. That's 6,000 newtons. And I know that the safety factor should be above 2. So let's do the verification.

So the first step I'm going to create a simulation, and create static stress study. Well, now let's take you through the design. There's a handle, and u-screw, and an aligner.

These parts, I probably don't need to simulate. I can replace that by constraints. So I'm going to suppress that from the simulation because they're not part of the problem I'm solving.

And now I have here the upper arm that I could transfer the force to the main body. And then it's a screw that holds this together, where the bottom part, essentially touching the ground and transferring the force there. So let's start with a definition of the simulation.

I'm going to put a load there. And it's going to be 6,000 newtons, for 600 kilograms. And here I remove the aligner. So instead of that, I will put that frictionless constant it kind of represents, the car that is touching here, but can slide.

And it's held to the ground by the surface. So I would put the fixed constrain there. OK. So I think I'm done for simulation with the definition. However, I see the yellow car here indicating that there's something that needs my attention. So I will click to check.

And it's going to tell me that actually the contacts are missing, which I forgot to do the contacts. So let's select the degrees of freedom to show how it looks. And now you can see that the green color says it's fully constrained, fully fixed because there's a fixed constrain between the ground and this part.

And this one is potentially fixed because there are some constraints, but not enough. And the rest is in that color that means that it's not fixed because I didn't make any contacts there. So let's do that.

And instead of doing the contacts manually, I'm going to ask for automatic generation. And now if I display it, everything is green. Everything is constrained. So I'm ready for simulation. It would be good to solve.

And now it do generates quickly the match and solves the equations to give me the results. You can see it's pretty fast. I have a small, not much powerful computer. But the calculation takes just seconds there. And here are the results.

So the legend shows me that actually the minimal safety factor is 2 point something. So it's above 2. So I verified that this jack will do it's function. It will elevate the car, and it will not break.

But I can actually take a look and learn more about my design from these results. So let's display stresses. And I can, for example, take a look what parts of my design are loaded a lot. So I will do some thresholding.

And now you can see that this screw is actually loaded a lot. And it kind of makes sense because take a look at how big this body is, and how big the upper arm is. And the screw is also taking the path of the power. But it's really small. So probably I have a little over design these two pieces in reference to the size of the screw. And then you can see that also around these pins, there is some stresses accumulated. So probably there's a space to make these kind of better, maybe change the dimensions there. And also, in the path where the car is actually touching the jack, it means there are some stresses. So I'm good. It's over the two safety factor. But still, I can do improvements there.

Or, I can display it from a positive perspective and actually display parts of the design that are not loaded much. So let me do it, try holding in a positive way. And now you can see that actually this bottom part is almost not loaded.

So the loader is really small. It's actually about 26 megabytes only. So it means that I probably can do a lot of organizations there. And it's also kind of self-explanatory. If the force is going down through the spin, and it's free, it rotates around the spin, this one doesn't carry much load, is there to position it under the car, and to better fill the space. However, it doesn't transfer any load, so it could be much tinier.

So this is a type of the directional guide as the software gives me to do a better design decisions. And this one additional thing that will help you to better understand function of your design. And that's animation. So I'm going to adjust the deformation by a factor of two.

So now you can see that while this deformation that's in red, a fractional of a millimeter, is shown in a big way. So you can understand the deformation. And now I will start animation.

And animation shows you how to design behaves when loaded. So you can see that as the force is coming here, it deformates. There's a deformation here. And you can also see course changing. It means the stress generated inside of your design.

You can actually combine the thresholding with animation. So if I were to do a little of thresholding here, you can see it starts with a 0. So it is transparent. And then the stress is generated.

And if you will take a look here, the stress will never go to this area. So this is not loaded much. And you could do optimization here. Or, I can do a thresholding only in the upper part of the design. And then you will just see the pieces of the design that are loaded. So it will highlight you how the stress is coming through the areas of the design that may need some of your attention.

AUDIENCE: How can you see where the real [INAUDIBLE]?

PRESENTER: So the easiest way is to switch here to displacement. And they'll tell you the values. So will see that the biggest one, the direct one, is actually 0.9 millimeters here.

AUDIENCE: [INAUDIBLE].

PRESENTER: Well, if I will go to the actual-- while the deformation is so small that the naked eye is not able to see it. So you are not able to analyze it. That's why we have this.

You can press here. That's a magnification tool that will make it so big so you can understand it in the forms. It's like that if you will zoom it. So it is better understand your design is good to zoom in because you will clearly see what's happening.

AUDIENCE: [INAUDIBLE]?

PRESENTER: I think what you are both really interested is this number, the safety factor. Because typically, they tell you it needs to be above two. If it is above two, that's safe. So if you say, I proved that it's 2.4, so you are good. You are covered because your design is safe.

This is just like the internal behavior of your design that gives you the actual guidance to improve, because the design is not perfect. You can always do improvements that maybe there are some holes, or need some pieces here and there to be better, and have the stress more evenly distributed through your design.

AUDIENCE: [INAUDIBLE].

PRESENTER: Sure. Sure. Let's do that. So I would put another load. And let's say put it here to the surface. And let's say it would be like 6 kilograms, so 500 newtons. Let's find out what happens.

AUDIENCE: [INAUDIBLE]?

PRESENTER: Right. I will need probably put something here as well, as there is a friction between the car and the-- but now, without that, you can see that it's-- let's turn it to this way. So let's use animation again to show that, maybe the fast animation there.

Yeah. And you can see that now it's actually moving in the two directions. But that's based on the data. There is a friction less constant here.

So we can move. But in reality, it would not move because the friction between the jack and car is too big. It would not allow you to move.

So probably I will need to add another constant here to limit the placement, because there's a log to free movement. And assume as I do it, actually, the friction between the surface, and this area, and fiction here would probably eliminate a possibility of the jack fall out, because it will be kind of-- the forces will create so much friction that it will not be allowed to slide.

AUDIENCE: [INAUDIBLE].

PRESENTER: Right. Right. Right. I would need to simulate. Right. Right.

AUDIENCE: [INAUDIBLE].

PRESENTER: Right. Right. In this case, I would probably need to add a simple model of the car frame or something, and limit this one. And it will emulate the whole system actually turning because--

AUDIENCE: So I think, actually, what we're doing here is just a linear analysis, right? So it's that one point in time, once it's being jacked up, right? So if you look at something like Inventor, which has dynamic simulation, you can actually look at the process of jacking it up and then do an FDA study at the end to look at what kind of nodes were generated, view for the motion, as well as the weight of the car. And I think that would give you a more realistic of that entire process, rather than just this one snapshot.

PRESENTER: Thank you. So let's move to the next one. And that's modal. Modal is the really interesting one, because it's easy to set up, but gives you interesting results.

So modal analyses, finds the natural frequencies for your design. And these are frequencies that should be avoided because it can cause breakage and damage of your design. And I have an interesting example here, the wine glass.

And maybe you have seen int he movies the scene where somebody is holding the glass, and singing. And the glass breaks. So let's calculate if it is possible and what would be the frequency, and what are the shapes of that.

So the modal analysis, it's really easy to set up. You just put up a couple of constraints, and you take a look to results. And again, animation is it's a really useful tool here to see the results.

So I'm going to run animation. And you can see was the shape for the first mode, while the next frequency is going to make a deformation in a different direction. And can review all the frequencies that all of them is causing slightly different type of deformation.

The mode five is really interesting shape of deformation. Mode six seems like rotation. Mode seven is, again, interesting deformation. And mode eight is kind of a movement.

But these are more the critical one. The first frequency is the most critical because it requires the lowest energy to do the breakage. So we found out the frequency that would break it.

AUDIENCE: How do you [INAUDIBLE]?

PRESENTER: Well, this is without units, because the vibration, you can't have what will be the yield deformation. It will just break. So depends on-- and actually, it will vibrate more and more and more and break. So there's actually no real units on it. It's just showing you the shape to understand how it vibrates.

And let's find out, just for this case, for the fun, how would this in reality. So if you take speaker here, put a wine glass and a microphone, and start changing the frequency, it actually will break in about the same frequency we calculated. So the calculation is giving you the same results as your measurement.

And if you take a look at the shapes, you can see that the measured shape and the real calculated shapes are all the same. There's just, they fell beneath the magnitude because it's just calculates the shape, not the magnitude.

But you can see that it bows that calculate the frequency, and the shapes that were measured and calculated are the same. So this is kind of a nice example of how the digital prototype can replace the physical one. Because doing all these measurements takes a time. It costs you some money, at least the cost of the glass. So it's nice to be able do it digitally instead of wasting your time and energy to really break the glass.

Next, is thermal stress. So for thermal stress, I'm going to show you how to fix the program of heating. Thermal, analyze temperature distribution in your design caused by heat input and output. And the assumption is stabilized heat flux. So it's a system that's in the balance.

And I have here a simple example. I have passive CPU cooler. And I know that there is a certain amount of watts generated by the CPU. And the maximum allowed temperature in my case is 50 degrees of Celsius.

And I'm getting the report from the field that's essentially overheating. It's going slightly over 50 degrees of Celsius. And I'm going to show you how to fix the problem, how to change the design to go under the 50 degrees Celsius. That's my goal.

So I'm going to create a simulation, and create a thermal study. And now let's take a look what I have here. I have the circuit board, the CPU, and the cooler, very simple assembly. So I will start with definition of my simulation

First, I will do the internal heat because I know the pattern of my CPU. So I can select the CPU. And it will then define that it's 39 watts. And now I'm going to start with cooling. I will use convention, and select all the surfaces here for the board.

And I know from tables that there's actually 30 watts per square meter in Celsius, cooling perfection factor. And the temperature inside my case is about 30 degrees of Celsius. So I defined the cooling of the board.

And now I'm to hide these two to define my cooler. So I'm going to repeat the convention, select all the faces again. But now, actually, it's bottom surface needs to be excluded because it's glued to the CPU. So I'm going to go remove it and keep just other surfaces being cooled by air.

And I'm going with the same 30 watts per square meter in 30 degrees of Celsius. So it was very simple definition of simulation. And now let's find out how would the design perform.

So I'm looking to solve. And it's preparing the modal. And even for such a complex modal, like this one, the machine can solve-- It's pretty easily able. So it takes just a couple of seconds to do that. It's pretty well optimized solely for analysis types of tasks.

And here are the results. And you can see that actually, that there is the other problem. It's 52.96, almost 53 degrees of Celsius instead of 50. That's my goal.

So I'm going to fix it. There are multiple ways how to do it. But I'm going to select the easiest one. And that's changing the dimensions here. So let's go back to the modal.

AUDIENCE: [INAUDIBLE]?

PRESENTER: These are kind of taken from the table as just for this conversation. So it could be more complex to find out for various type of materials, and geometries, there are some empirical rules, how to select it. But for that case, I guess selected this one, just to demonstrate it.

AUDIENCE: [INAUDIBLE]?

PRESENTER: What I did for this demo, I just searched internet. And I specified the type of the program that find an empirical value that people are typically using in these scenarios. So it looks like these factors are a lot of to the situation that you're in.

So the easiest way is to find out, like will it? But if you are not into it-- a side note-- I guess, if you are expected doing thermal, you will have your own tables, or your company will have some standard procedures what factors you should use. But if you don't, Google will give you good suggestions.

So I'm going to change that just a little from 13 to 15 millimeters. And you can see that now this is going to be slightly higher. And I'm going back to the simulation. And I'm going to solve this bigger modal.

And you can see that I fixed the problem. Now there's a maximum temperature lower than that

50 degrees of Celsius. And now you can again do some analysis. So in my case, let's take a look how the heat is transferred.

So it starts with a CPU. And you can see how it is transferred through the cooler. And then you will see that the board, because it's from ABS plastic, it's material isolates. It doesn't take any part of the cooling.

But there's one interesting thing. If you will go back, you can see that actually this shape, it's kind of rounded. So it probably would make sense to not do the cooler that's kind of a cube type of cooler, but do it more rounded, because it will be much more aligned with how the heat is transferred.

And now it's like the designer's decision. Would I make a cooler that would be more expensive to produce, because producing, it sounds really cheap. You take a block of our medium and just do these cuts in it. Or, would you invest in something more expensive, and do the cooler a different shape. It will perform better, but the production costs will be high

So these are the guidelines that the software gives you. And now it's up to decide if you will take advantage of it, and adjust your design based on these guidances. Now let's talk about a thermal stress.

Thermal stress, it's actually a combination of the thermal and lineal static stress. So it's a two-step simulation. It starts with calculating the thermal stress. And heat causes deformation. And deformation causes stresses.

So the first stresses are calculated. And then the stresses from within statics are calculated as well, and combined together. So that's like the most compact analysis we have right now. But you combine both the stresses from heat and from the static loads.

And in my example, I'm going to optimize the simple brake system here, and find out what I can do to make it cheaper and smaller. So here's my design. Let's create a simulation and thermal stress study. And if you take a look at the design, you can see the rotor.

The rotor is connected to the wheel. And it transfers the torque from the wheel. I have to break pads. These brake pads are touching the rotor, and actually creating the friction that's slowing down the wheel.

So the temperature is created here because of the friction. And there are also forces that are acting. There are 5,000 Newtons force to each of these brake pads because of the braking system, and because the friction is also causing a torque. There is a tangential force, 3,000 newtons by each brake. But that's like removing the energy from the rotating wheel.

And we know that's actually the temperature here is 220 degrees of Celsius. That's like what we know about. And we would like to have the safety factor bigger than three, but not much bigger. So that's my design. And that's the goal.

And here the caliper is actually holding these two brake pads. It's doing two things, holding the brake pads, and also transferring the heat out. Well, because I know the temperature here, I can very much simplify the simulation by moving the rotor, because the temperature is known. So I don't need to calculate the rotor. And I can focus on the caliper with the two brake pads.

So let's start the definition of the simulation. I'm going to hide the caliper to see just the bake pads and define the simulation there. So I will start with a structure load.

I know that here, there's the load. And it has two fractions, the one that is causing the friction, and the second one that is the dimension. And I have here the direction type for vectors that's really perfect fit for this example. So on the x, I know it was 5,000 newtons. And on the z-axis it was 3,000 newtons. You

If I click OK, you can see now the force is actually having a direction that's combination it's z and x fractions. And I'm going to repeat the same thing on the second surface. Go to the vectors, and specify, again, 5,000 and 3,000.

So these are forces. And now we know the temperature is 220 degrees of Celsius. So let's add a caliper there. And I will ask for a better picture, because I know the value of the temperature. And I will select the two surfaces here and specify 220 degrees of Celsius.

So I'm done with the definition of what's happening on my brake pads. So let's display the caliper and hide the brake pads. And let's concern the caliper. These are the two screws that are holding the caliper to rest on the car. So I'm going to add a fixed constrain there to both of these screws.

And the cal is also cooling the system. So I'm going to put that load. That's actually the convention.

And I'm going to select all the surfaces here. But now I need to remove some, because actually, here are the break pads connected. And although, the internal part of the design is not helping much with the cooling. So I want to remove these internal surfaces.

And I know that the external temperature through degrees of Celsius is the current temperature. And for convention value, I know here that they have a value 00.04. It's [INAUDIBLE] in this case.

So this is the definition over the simulation-- so the surface, again, showing the yellow car. So I will go with a brake check, and I didn't specify any contacts. So let's go with automatic generation. I will just confirm what is the surface I just need to do. And now I have fully defined simulation. So I can solve it and find out what's happening.

AUDIENCE: [INAUDIBLE].

PRESENTER: They are. They are just hidden. But they are calculated. If I should display, you can see that it was calculated inside. I just--

AUDIENCE: [INAUDIBLE]?

PRESENTER: And now you can see that the mesh is in the car. So it doesn't look good. It looks like the passage of the calculation wasn't nice with such a coarse mesh. So I can show you the settings here where you can change that.

You go to the mesh. And then this first value is the average size of the element. So as you go to FEA based on the item, you split your body to multiple small pieces. And for each piece, you define the equation, how it relates with the rest of the design.

So now it's about total percent of the size, the size of the element. So I'm going to make small, just 5%. And order to solve the problem with this rounded things, I'm going to ask to create curved mesh elements.

So let's confirm it and, of course, solve again. So it's going to make it again based on my change requirements. And now it looks much better. This is really circular now, and the mesh is not that coarse.

And the result is that actually the safety factor, the minimal safety factor is four. And my goal was go to three, or slightly above three. So its over designed from that prospective.

So let's take a look at what we can do with that. There are multiple things. I may do it from a cheaper material. I can change the shapes. Or, I can just permanently make it smaller.

So let's display from this perspective and go back to the modal. And I'm going to modify it. And here, bass of degree, 40 degree Celsius. This is this angle. So I'm going to make it smaller to same some material. And I'm going to put 30 degrees of Celsius-- I'm sorry-- the angle in degrees.

And you can see that the result is now much smaller. So let's go back to the simulation and show it again. By the way, there's the icon that tells me that it needs to be solved. So even if I forget it, I did some changes, the software will notify me that the results are not up today.

And as a result, you can see we improved the safety factor. It's closer to three. That was my goal.

So let's take a look how to design performs. I can first take a look how the temperature distributed. So let's display temperature, and find out how it go through it.

So it's generated by friction here. And then it's transferred through the brake pads. And then the caliper is going to take that energy, and distribute it out, and cool it. And you can see how it's going out. So there's some interesting situation happening here.

You can see that cooling here is not optimal because-- the material is not solid there. So the cooling is slower. So maybe this is some other improvement for future changes in the modal. So that's another example of the directional guidance that the software gives you.

And if I take a look to the stress, I can do some experimentation as well here. So for example, I can display parts of the design that are not loaded much. So I'm going to try and load here.

And height of the pieces of the design that transfers values and keep [INAUDIBLE] not loaded well. And if I will display it, you can see that actually, this part of the brake pad seems to be over designed. I will show why it is there.

So if I will display the modal, you can see that here is the part of the brake, the part that is actually touching the rotor. And this is just a supporting area. It's slightly bigger.

But it looks like it's-- could be smaller. So we can take the directional guidance and make this smaller. Another area that looks to be over designed are these screws. Let's find out how big they are. If I will go to the Inspect, and select the ammeter here, it's a tread 3 millimeters.

So I have two thread millimeters screws that transfer the force that's about 3,000 newtons on each brake pad. So it's a total of 6,000 newtons. It's this 60 kilograms.

So probably two 20 millimeters big screws are too big. So it looks like the problem is aligned with the results of the simulation. So as of next step I can take the directional guidance and make these smaller.

But I'm not going to do it now because we are limited by time. This is just an example of how the software is going to help you step by step to fine tune your design and make it better and better.

So the last piece is the cloud simulation. Cloud simulation is a new technology we are working on. It has multiple aspects. And today, I will show you just a little subset of it.

So let's take a look to the first scenario here. We have the average part of small assembly, stuff like I was doing in today's class. We will focus only on one study at the time. And it's about just 100,000 elements in the mesh.

So for local solve, I'm using my average computer with 80 gigabytes of memory and one 2.4 gigahertz CPU, so average computer. Even on this small computer, it solves things in just seconds, in maximum minutes.

So it's very fast and responsive. Our service optimized for static stress for these small parts and assemblies. If you would compare it with cloud simulation, you have similar time experience. You don't see much improvements there, because even if the cloud service is much more performant, the time is about the same.

So you will see much better things and richer collaboration and data sharing. That's the value of the cloud simulation, even for the small assemblies. But I'm not going to talk about it today. I'm going to focus solely on the time aspect of it.

So let's go to the second scenario. We have larger assembly with more complex part, but still focusing only on one study at a time. So let's take a look how it works.

So for local solve, I'm going to call the solve. And now it's happening. So it's going to mesh and solve. And because this is a big part, it actually takes time. So now I only spend two hours, three hours, I'm still waiting for my results. And in about four hours, I'm getting results.

So I spent four hours to do analysis of this component. Now, let's compare it with a cloud simulation. With cloud simulation, I click through the cloud solve. And it starts to do the job on the cloud. So I can continue to use my fusion.

I can review what's happening. I can go to the visualization. I can do whatever I want because my fusion still works. It's not lowered by the heavy calculation task because it's happening on the cloud. And when the results are done on the cloud, I can see the rotification here. I can click to it, and describe the result.

So it was just one hour here. And that's because the cloud work is much more powerful. Here, on my older computer, with just 80 gigabytes of memory, it starting slopping to the drive. And that slows things a lot. So cloud work is much more powerful.

And I'm saving actually three hours. Instead of solving four hours, I'm solving just one hour. And what's even more important, diffusion is not impacted. So I'm saving actually four hours of my time. Because in my first example, the computer will slow and not responsive because it was full only by the calculation.

So nice saving, even on that one scenario. And now let's take a look to the what if scenarios. I have the same design, but I have two variants. And while the reason to use the digital prototyping is to analyze the variants, that's the core of the value here, you want to analyze multiple variants, multiple what ifs, to find out what would be the real solution of your problem.

But let's take a look, even just two variants, how it looks at a local solve. I'm going to call the first one. And now, fusion is really busy. I see the modal dialog. I can't do much here. And my entire computer is slow because all the CPU power is used to do this simulation calculation.

So I'm waiting the same four hours for the results of the first local solve. And when I'm done and I see the results of the first one, I need to go to the second one, to the second variant, and do the same thing again. So I'm clicking to the solve again.

And there are a lot of results in front of me to wait for the results of that second variant. So that's a very time consuming thing just for two variants. Well, imagine if you have six or four variants, something more yield, it will take a long time.

So infest eight hours to wait for it. So you need to do sequential source of two variants. Fusion is frozen, and computer is not responsive because it's loaded by calculation.

With the cloud simulation, it's very different. You call the cloud simulation for the first scenario. You see the indication it's solving. And you can immediately go to the second scenario and show it again.

And now you can do something else. You can maybe set up the third simulation, or do whatever you want in your fusion. And on the cloud, the calculation is happening without any impact to performance of your computer.

So for example, the simulation of this knife during the solution of the first two variants. And then as the results are calculated, you see, again, a notification. And you can view the result.

So now you can analyze the results from the first simulation while the second one is still solving on the cloud. And in about one hour time, the result of the second simulation are coming as well, because actually, they are in parallel on the cloud. So you can see the results as well.

So in the total time of one hour, you could post the result. And that's because there are power calculations. So independent of how much variants, how much what ifs you are doing, it's still this one hour of the one solve. So here you are sequentially counting altogether here. There's no additional time because you have unlimited power of the cloud in your hands. And you can do power calculations there.

Fusion is not impacted. So you're actually saving the full eight hours. Your whole working data is the difference between you're sitting in the front of the frozen computer or being able to [INAUDIBLE] more. So that's just a quick preview of what's coming in the near future. We are going to do another cloud simulation that's going to make even a more complex calculation much either too.

So let me summarize it what we did today. We talked about the key concepts, such as loads, and constraints, and so on. We verified the check design, and proved that it will work and transfer the load.

We found a frequency that's going to break the wine glass. We fixed the program, it overheating, using thermal. And thermal does help us to optimize the design of the brake system.

And the course was kind of a preview of what's coming next. So before opening into question, let me check this with the team. And we'll have to prepare the simulation for today's class. And now I'm here to answer your questions. So please feel free to ask. OK.

AUDIENCE: Are there any tools to [INAUDIBLE]?

PRESENTER: Not yet. We don't have this yet. We have adaptive meshing, so the surface automatically adjusting it. But we don't have a manual control there.

AUDIENCE: [INAUDIBLE]?

PRESENTER: All right. All right. And here with adaptive mesh environment, you can actually let the software identify the pieces. And it loops. It will make the mesh smaller and smaller on the target. So we can switch this on. And it will now do multiple loops, and kind of automatically optimize the mesh.

And there's one thing that I actually didn't show today. And that's the voxel mesh. We are using-- in a case, there is issue with the mesh. So with a traditional package, what happens that if the mesh fails, we need to go back and simplify the modal. You'll need to fix the modal, actually change our modal to help the software to generate the mesh.

Here, we took a different approach. If the mesh fails, we switch to the voxel-based meshing. That's kind of a different algorithm that does the mesh that maybe not perfectly covers the body. But it kind of bridges-- adapts with issues. So without asking the question, it gives you the solution.

The cost is that the resolution is lower. However, engineers typically don't have the knowledge to do the simplification to fix the mesh issue. So the voxel mesh is kind of our plan B. And if things go wrong, it still gets that result.

AUDIENCE: [INAUDIBLE]?

PRESENTER: Yeah. Yeah. Yeah. It does. Well, let me go with a simple example here. And if I go with adaptive mesh environment. When I start a solve, you can actually see the [INAUDIBLE] inside. And here, there's a new icon, the plot that actually shows you what happened during the solve.

Does it make sense?

AUDIENCE: It's very similar to what [INAUDIBLE].

PRESENTER: Right. Right. Right. It's based on the same solver.

AUDIENCE: [INAUDIBLE].

PRESENTER: OK. Any other questions? So I will be actually located today from 2:00 PM to 3:30 in the fusion office here. So if anything comes to your mind, piece come and we can talk about It. I can do some demo, and we can show it.

OK. So thanks a lot for your time. And see you next year.

[CLAPPING]