PRESENTER: So yeah, so in chapter 2, we're going to go through a simple example. So before we dive into the injection, producing a mold, and the all the complex idea, we are going to go through, like, a simple example that all of you can relate to. And that's something you saw yesterday in Jack's presentation.

So I'm going to begin from there, and then, in the last part of this presentation, we're going to apply all those concepts that we learned in the 1 and 2 to produce a [INAUDIBLE] cooling mold and to see that they, you know, what does the design look like now.

So if you apply all those concepts, if you apply the concept of the generative design and use the data from the mold flow, what are the potential shapes available? Hey, welcome. So I know it's the second round for you, but--

So basically, one other thing that we saw in Karl's presentation, that a lot of these car racing-- so a lot of these parts I mean that we design today, we think about the manufacturing, but we forget about the performance part of it. And how these cars, performance of manufacturing and performance, need to be under the same umbrella. Otherwise, you can create the situation where you might have scenarios that the part is going to fail either way. So if it's not secure, manufacturing-wise, or the part is not secure taking the loading, the structural loading, it's going to turn out to be a failure.

So basically, are they think, you saw the video that the guy goes and flips, and so one of the things that basically protects the head of the driver is a rolling hoop. So this is one of the rolling hoops was designed with the additive manufacturing. And so the next story is going to be, so what other consideration has been taken in designing these type of parts, and why additive manufacturing produces the additional layer of challenges, unlike using something like casting.

So in additive manufacturing, many things can go wrong during the 3-D printing. So in the pictures that you see on the screen, you see this picture? So at the bottom of this, I mean, if you pay a little bit close attention to this area, the top piece was actually a part.

And the bottom piece the printer actually printed is called a support structure. The support structure is required. A support structure basically makes sure that the part is supported and not bend away. So basically, it's an anchoring strategy, with most of the power metal printing today.

And so, if the support is not designed properly, a support can fail and can cause a distortion. Now, the build plate itself can fail. The coder blade can hit the plate. So these type of failures happen pretty often, and end result is that, you are thinking about making the design right in a single shot, and it took you three or four times and compounding the cost of printing-- and some of these parts, like one of the parts that you see with the liquid lattice, is actually costs almost $6,000 for us to print one piece. So imagine like printing the six pieces of those, and all of the sudden you are spending $36,000 in the printing cost.

Then the other issue that-- so who has that implant, the other and smaller one? So one of the things that they pay attention to-- the fabric of this implant. So unlike the traditional casted part, you're going to see that the surface is actually porous.

So the density inside the metal printed part is a big issue. And so, if the porosity is really very small, we don't have an issue with that. It could be taken into the account, but imagine, like, having the porosity like this, having the holes inside your mold like this.

So they happen from the variety of reasons that, you know, there was extra oxygen content that got inside and produced these holes. That's one other common reason. Then there are other issues that the laser parameters that we used, and the material that we used, end up causing this.

So this is something that we have to avoid at all costs because this basically adds another layer of unpredictability inside your metal branding part. And for that, I mean, often, you have to do the inspection at the end of the printing process.

And then, so you're making the support. But you have to remove the support. So either use a five axis, three axis machine, or take a hammer and start knocking it down. So in either case is that you have like a lot of extra work cut out for you, even after the metal printing work is done. So cleaning up is a major part of it. And you see that the surfaces around it, it doesn't come out nice and smooth looking. You have to go and polish these surfaces later on to make sure that the surfaces are proper.

So this is kind of like one of the bigger issues a lot of people identified that, in some cases, it takes more time to remove the support material than actually to make the part itself, and to make the mold itself. So you have to be very careful. And there are several different little things that cause this to happen. For example, the overhangs is a big area people try to win at all costs. Occlusions inside the part, the part is actually concave inside. And that causes these excess support material to get deposited there and cleanup needed.

So with that, I mean, they were going to look at a quick simple example and see that the how does the design process works, I mean, with the additive. And then we're going to take it to the next level and apply the same concepts on the molding side of things.

So this is the example that Jeff was actually showing. And so this is basically taking something called an upright-- all of us drive cars, and the upright is one of the part that connects the suspension of the part to the wheel. So this is a double wishbone suspension, and there are basically two uprights that are connected there.

So how do we apply all the different concepts in how this part was produced? So this a relatively simple example. Producing the examples like mold, really a lot of forces in action. It's a little bit more complex than this. So we're going to start with something simple. So immediately, you have to identify the what is a design goal. So you have a different design goal for the injection molding, but in this case, a design goal is the optimal weight, maximum strength, and proper fit inside assembly.

And so this is the workflow that we're going to repeat later on. Again, so the workflow, the way it works is that we begin with the design, make sure that we verify the strength of the part that, if somebody start with if the question is, can you modify this mold? So if somebody already has a mold, and he's asking you to modify the mold, then you have to basically identify, if the mold was proper to begin with? Are we starting from the right place?

And then the second question you have to answer, is the optimization meaningful? I mean, so if you're looking to optimize a given mold, or a given part, then you ask yourself, I mean, can I make a meaningful difference in this? Is this work going to produce that the results are significantly different than a traditional design?

So I saw some of the molds that I was working with one customer in California. And I looked over the mold, and the inner volume was so small in that, that I feel like there's nothing we can change. So there's nothing optimizable, and the effort will be wasted in that case. Then, if all that is OK, then you can basically either go with the generator design concept, you can basically say that let's let a computer generate the design for us. And so you can basically take the endpoint and connection of it, and let computer create the design for you.

Then, once that is done, then you can basically apply the optimisation techniques. So different optimization techniques are meaningful in different circumstances, and we're going to learn that along the way. So the two techniques that we often use, so lattice and the topology optimization.

There are other types of optimizations, such as parametric optimization. There is a beam-based optimization, and on and on. But what seems to be working well in a lot of the injection molding problems is the lattice and topology optimization problem because we have a lot more control over the inner volume of the material using these techniques, and we can basically add volume and remove the volume from the area where we want to increase the thermal conduction or increase the thermal convection, and increase surface area. So these techniques, often, are more helpful.

Then the next step is building support. So once you've identified the design, created the design, you want to make sure that you can build the support. And if you want to, say, print more than one mold, you're welcome to. if you have, like, you know, like the bigger USM 290 printers, you have like a 300 millimeter volume, and if the two molds can fit inside the same volume, you can save some money by printing two at a time instead of printing one at a time or three at a time, that that depends on, what is the size of the mold, what is the size of the design. And it will save you a lot of cost along the way.

And so that does the nesting, basically, for you. And then, finally, the other important concept you are going to learn is the sliced file generation, and the tool path generation. So what happens is that, if you start making a part like this, so don't think this part came out from the 3D printer like this.

When it came out from the 3D printer, it was, like, really ugly looking. And if you would have seen that, you would have said it was really bad. So a lot of polishing, and a lot of the machining, and a lot of the effort went inside it to produce the part the way it is.

So the way the 3D printer works is it basically starts depositing the material layer by layer. Now in case of for powerband machines, the laser head can move from one direction to other, or it can go from this side. It can move in any possible direction.

Now, if it moves in a different direction, for a different part, guess what happens? You part the materials being deposited differently. So one part might be different than the second part internally.

So you might have like, two molds, but the two molds are not equal, may not be equal. So another big issue with the metal printing is the repeatability. How to produce the same part, again and again, of the same quality, of the same type.

And so, for that, I mean you have to do the two part optimization, and the slice file generation, and feed the printer the slice file. So basically, you're telling the printer, every time, move the printhead, or move the laser pad path exactly the same way over and over. So I have the same part coming out every time. If you're going to start depositing here, here, and here--

So if you go from here, then here, then here, then here, so this is hot, this cool down, you went here, now this is hot. This is cool down. So along the way, you're causing some deformation inside the part because the temperature is not uniform inside the part. But at least you can control the material property. If there is an issue in one part, then there has to be an issue in all the parts. Otherwise, you will have like you know one part of one type, and the second type.

Then basically, it come down to the build simulation. So why do we do a bill simulation? Why do we need to do that? Because, as I suggested earlier, you're printing layer by layer. So in each layer, the temperature is the highest at the topmost section.

And at the bottom of it, which is already built, the temperature is much lower. So technically, that will cause a thermal gradient inside the part. And the metal printed parts tend to bend very dramatically.

In fact, so, in this part, when we printed the initial copies of the part, some of the parts were completely bended so that they couldn't stand on the table like that. They couldn't stand because the base was actually bended. And because back then we didn't really have the tools to do the build simulation, then that was sort of the idea started coming to us that that's a very important part of it, that we have to identify that the material, and the machine parameters, that they have to work hand-in-hand to produce a part that you really need. And that's what you need a build simulation for.

And then, the next step is [INAUDIBLE], all right, so you're going to try different ones. And the end conclusion, in 99% of the cases is that, yes, there's going to be a minor deformation, but it's an acceptable deformation. So you're going to have, like, a deformation within certain tolerances.

So the better way to do that is, yes, there's going to be deformation, so how about starting with a compensated shape? It's a shape that basically is pre-compensated for the deformations so that's an upcoming functionality in one of our tools.

And then, once you are done with that, you produce a part, and so you want to make sure that you have a proper SDL file. So the proper file that you're basically sending to the printer. And basically, if there are any issues, you want a clean SDL file using our tools like Netfabb.

And then, finally, the fabrication. So once you're done with the fabrication, the story doesn't end there. You have to, basically, ensure that the printer actually printed the mold that you had intended for the CAD-- designed for. So you have to actually take it to the metrology, or the inspection part of it.

So usually there's a photogrammetric camera mounted on something like a robot, or a robotic arm, and then it basically moves around, takes a lot of pictures, and the pictures can be connected together to basically see what is the end deformation with respect to the original CAD model.

And the other thing a lot of people like to do, basically, they do the x-ray scanning on the part to see if there is any defects inside the parts, if there is any holes, oxygen bubbles that they created inside the part to make sure that the--

So in the previous example, so this part is actually a display at the AU Pavilion. So if you want to go and check it out, and I thought that would be, maybe, a good example before we go to the real case.

So in usual case, you're going to get a variety of different loads coming from. So in case of injection molding, you're going to have loads coming from the injection pressure coming on it. You're going to have a variety of different types of loads acting on different areas.

And in this part of it, you have these set of loads of acting. And you have to take like a dominant load case and apply on it. Then, once that is done, I mean, if somebody is giving you a design and telling you that, go and optimize this design, go and optimize this mold, make this mold better, it makes sense to basically check the original design first that, what are the potential issues inside the original design. So what are we basically up against?

And so that's what is done in this case. So we ran an analysis to see that, you know, what is the stress level in the original part, so what is original part, or was it good design or a bad design? And what are we solving for?

There are several ways to basically go from that point and onward. And so one of the ways, basically, you know, looking at the generative design, the generative design is one of the ways that we can produce on all these parts. It's not available as a product, but we are working on producing this as a product.

And so, basically, taking the end geometries, and asking the software to produce a design for you, and then, basically, the software will give you the all the possible combination of designs under those circumstances. So the tools like generative design tool that Jeff talked about doesn't produce single designs. It produces like hundreds of designs, all the possible designs that are under that type of load are possible.

And so, within those, you can choose the designs that are most appealing, designs that basically are right for your application. So it gives you a level of flexibility in that area. And then, finally, once a design is picked, you can start cleaning up the surfaces and come up with see the, what did you achieve, and can you basically further improve the part?

So one other way that you can improve the part-- let me just make sure that-- so one other way you can further improve the part is using the lattice-based optimization. So let us lattice-based optimization allows you to control the inner volume.

So in many cases, you don't have to do both the steps. In many cases, one step is sufficient. But if you want to go make the part extra light-- and here, the issue in the molding industry is not necessarily the weight itself, but rather the volume.

So if the volume of the mold is high, it can cost you a lot of money extra in printing. You can significantly reduce the cost, and you make the chances of producing a good mold by reducing the volume of the part itself. And that has a lot of benefits, I mean, once you basically send it to the 3-D printer.

So try sometimes, like a website like shapeways.com, allows you to base some of the designs for 3D printing. Some made the design that is basically completely solid. Some made the other one that's basically a hollow shell design.

And see the difference in cost. It's going to be several times. It's not going to be 20%, 30%. It's going to be 100%-plus difference in the-- and the dramatic difference is that it's not just the material cost, but the laser time. This also is very expensive with these machines because the laser itself is designed to go for certain hours, or a certain amount of time, and then you have to replace the laser to have a different one. And once you do that, the cost actually goes up.

And then you can use the Netfabb optimization tool, the lattice optimization tool in the existing design and produce a design that's basically for that improvement to the design that we created. So we're going to use the same lattice-based optimization principle in the injection mold side of it.

And the two big benefit that it provides-- so one other big benefit is that you reduce the volume of it dramatically. Second thing is that you improve the stiffness. So it really depends on the area the forces are coming from. And you can improve the stiffness to make the part a lot more stronger.

The Third thing is that, often, you have to control the thermal conduction inside it. So you want a lot more heat to be dissipated closer to the where the plastic is being injected from. And in the area far away from it, you don't want that the recirculation of the heat. You want to have it, like, as even distribution of the heat, so you can come up with the plastic part that has a predictable quality. And here you're basically controlling the inside of it in the volume of the part itself.

So basically, you can do the lattice optimization, come up with a design, and the volume is going to go down from the original design. And then, if you basically take it to a tool like the Netfabb, you can create the support structure for the part. And basically, the other issue that you will be facing is placing the part-- having the right orientation inside the print tray.

And this is a big major issue that-- if you have this part if you print this part, say, at a 45 degree angle, so if you put this inside the build tray like this, you're going to have like a supported structure all in this area. So a more supported structure means the cost is going up. If you can turn this around this way, now the supported structure is only in this area. So the you're supported structure actually went down.

So why would I put it in the 45 degree angle in most of the cases? Because that's the only way the part will fit inside the printer tray. So if you have like a 300 millimeter by 300 millimeter and 300 millimeters, and if my part is, say, like, 325, then you have to do it in this direction. That's the only way to do that. You cannot do that any other way in those circumstances. So you have to make sure the orientation is right.

And then the placement. So you might be able to fit more than one part inside your build tray. You might be able to have, like, a two or three part. So this is the-- and if you are orient it right, you can have the multiple part fitting in the same tray.

And so having the supported structure and then, finally, doing the simulation, the build simulation. So making sure that the part-- you do the simulation to ensure that the your machine parameters are right. And the material and the machine that goes hand in hand, they should be able to produce the part without too much of distortion.

So it's basically showing the distortion of 0.6 millimeter, in this case. And so that's basically, we can say that it might be under the acceptable range, and acceptable range is different for different size parts. In this part might be OK. But if you have, like, a small mold, it might not be OK because it might end up producing a part that's completely off the original targets, original tolerances.

And so yeah, the fabricated parts, that's the fabrication piece of it. And then, basically, so this is the photogrammatic camera and the robot that's in our Birmingham office. And so this part actually went through the inspection, and, during the inspection, you can basically see that that is the ASD geometric tolerances matches with the original CAD part that we designed.

And it's a very important part of it because, I mean, who knows that something went wrong along the way, and the part is not of the accepted quality. And now you're basically producing 200,000 parts using the same mold. And so do you want to lose all those 200,000 part because some tolerances got mixed up during the printing process? No. So you want to have, like, make sure, double check this matches with the tolerances of the original design.

So a photogrammatic cameras, and our tools like the power inspect does a fantastic job in that area. So basically, the robot is taking pictures of the part from several different angles. And you can basically take all those images and basically stick them together, and superimpose with the CAD package to see if the tolerances actually match this back to the original design. And if they they're not, then you have an issue.

So in this case the, maximum deviation was found to be 0.1 millimeter. For this particular application, it was acceptable, but for the other target application, it may not be acceptable. So that's something you have to think about. So any questions so far?

AUDIENCE: [INAUDIBLE]

PRESENTER: Cost of 3D printing? Yeah, so it's called the shapeways.com. It's actually a service provider you can, basically, instead of owning a 3D printer, you can send it over to the Shapeways. They print it for you. So you can basically submit the job and get, like, a job estimate that the how much it is going to cost to 3D print.

AUDIENCE: [INAUDIBLE]

PRESENTER: Yes. Yes. So we have power inspect and power shift. And we have a few experts in the room from that. You want to make a comment on that?

AUDIENCE: [INAUDIBLE]

AUDIENCE: [INAUDIBLE] and then the laser, or whatever, to fuse the particles together, correct?

PRESENTER: Yeah.

AUDIENCE: So what sort of temperatures are we talking about when they are fused together?

PRESENTER: You mean at the centering temperatures?

AUDIENCE: Yeah.

PRESENTER: I mean, those are, like, the melting temperature of the powder, and depends on the material you're looking at--

AUDIENCE: Right, it's very localized, right? Has their been any consideration of fusing it together and a cooling technology so that it does not have a chance to dissipate through the rest of the part that's already built to minimize distortion.

PRESENTER: Yeah.

AUDIENCE: Has their been anything like that?

PRESENTER: Yeah, there are a couple of guys who are producing those-- they're coming up with-- there's nothing like that that exists in the market right now. But there is a company called Xjet in Israel, and they are looking into the basic instant cooling technology. They're basically promising to come up with a printer like that. They had one printer in the last show that they promised to deliver next year, RS owns part of them. But I'm not sure what the other pitfalls are going to be because they have so much of cooling infrastructure around it, and they basically promised to remove the supported structure also, saying that the support will be soluble in some sort of liquid. We don't know yet, all the details.

So it's more of an experimentation phase, those type of things. The power bed technology is a lot more mature. So it's a basically known devil, the known demon that we know what the problems are. All these new technologies that are coming out, that we don't really know what the problems are, that we're expected to face.

But a lot of people are thinking, and doing a lot of research along those lines, like can we cool it down instantly. But it's just that the reality is that almost the 70% to 80% parts are being printed in the powder bed, one type of powder bed or the other, and the second is the eBeam side of it. In either case, there is no chance of having a cooling line around it. It's just a closed chamber.

Cooling lines, opening up, means that you're allowing the oxygen to flow somewhere. And oxygen means that you're going to have holes inside the geometry, and it will not be able to achieve the desired density inside the part. So porosity is the issue there. Any other questions? Sure.

AUDIENCE: [INAUDIBLE]

PRESENTER: Yes. So basically, there is going to be some distortion inside the part. But you can basically preplan for the distortion. And that's what our simulation tool is, the process simulation tool is, inside the Netfabb. So basically you're never going to get it in a situation in additive manufacturing when you have a zero distortion. There's always going to be some-- so we have to basically figure out that, you know, how much you can digest the distortion.

And in many cases, you can come up with, like, a very minor amount distortion, the acceptable amount of distortion inside the part. So, and that is something you can predict ahead of time. If you go with this machine, and machine has, like, a laser parameter that the laser was this much what, and the powder that you took, powder properties are usually well known, and you can basically--

So our Netfabb simulation package, that's exactly, will run the simulation, will tell you this is the [INAUDIBLE] distortion expected. If you feel like this is an acceptable distortion for you, then go ahead and do that. If not, then look for the different machine. There are many vendors in the market-- or look, like, a slightly different variation of the powder. So you're going to find the powder, like, in several different gradings inside the market.

So you're going to have the grain size of the powder matters because one grain size works with one capacity laser. So search for a different powder. You can still print in aluminum with a different grain size powder in those cases.

And there are several vendors. So they're starting from Alcoa, which is the largest vendor right now that was selling powder, you're going to find, in Germany, there are a lot of mom and pop shops. I mean, they're selling powder. And they'll tell you they can produce a powder of any grade.

So there's going to be some minor, minor distortion regardless of whatever you try. But you can predict it, control it, and plan for it.

So this is the design workflow that-- it started with the initial design with the applied load. You made sure that there is no aliasing issue. You come up with a new design. You optimize a new design, and you basically make sure the new design is printable. You took it through the process. That means having the right support, support that doesn't break from the original part. You know, that's the other big issue that I showed in the initial slide. Support itself might fall apart. And then, doing the, finally, the metrology, the inspection a piece of it, to have like--

So generative design can help you identify that it's a tool. There's one other way of doing that. There are many other ways to do the same thing. Lattice-based optimization can help you remove the further weight. And then we went through this whole process, supported structural generation, slicing, hatching, tool path optimization, and the identification of distortion due to the additive process.

And then, finally, [INAUDIBLE] is the inspection, very important part of it. If you forgot that piece, then, again, you have issues there to worry about.

Now, so the chapter 3 that we're going to begin with is an example of applying the similar principles on a [INAUDIBLE] cooling mold. And so the molding industry has been looking into the [INAUDIBLE] cooling mold for a while now.

And so the initial designs were, like, the cooling lines that actually drilled inside the mold, and if the cooling lines are too simple, then you can do that. And that gives a certain benefit in terms of the cooling. But when the mold gets lot more complex, when you have, like, a cooling channels wrapped around it like this, in that case, that the drilling the holes inside the mold, it's just not possible. Additive manufacturing is the only way to do that in those situations.

So basically, we're going to go with the example of this blender. And we're going to utilize the housing of this blender to see how can we make a better mold for this housing, and what does the optimisation look like for the housing of this blender?

So design goals, I mean, so that's something you have figured out that, what do you tend to achieve in the end? What are my key issues? What am I optimizing, doing all this work for? And so, basically, one of the thing that you want to make sure that your volume is lowest, and the printing cost is least.

It's important, because if you ask some of these bigger companies, I mean, you can ask for a quote. I mean, try some of these sites like the Shapeways, and you'll find that some of these molds are very expensive to print. It can cost you, like, you know, tens of thousands of dollars to print a metal part.

So having the lowest volume, and having the least printing cost pays up very quickly. And then you want to make sure that you have, like, a optimal strength or maximum strength inside the part so that the different types of loads that are coming on the part, it is able to withstand all those different loads.

AUDIENCE: [INAUDIBLE]

PRESENTER: Powder is less of an issue. It's the time inside the machine. And the reason for that is that the laser, the laser time. So lasers are designed with a certain life in mind. They don't last that long. And if you need to replace a laser, then that is basically is a lot of money.

In fact, I was talking to somebody at a very large metal printing company, and they were saying that the majority of the profit comes from the lasers, not from the printer itself. The printer, actually, they are selling at loss.

AUDIENCE: [INAUDIBLE]

PRESENTER: Yeah, yeah if you have repeated failures, then also the cost keeps adding up pretty dramatically.

AUDIENCE: [INAUDIBLE]

PRESENTER: So yes, it goes for like few years. Usually, once you install it, the stock machines, melt vending machine that comes from the manufacturer, it goes for like, whatever, the-- usually, they have a certain warranty period on it that it's going to last for that long. But, ultimately, it depends on what do you end up printing with it.

So if you're taking an EUS machine and using the aluminum, which has a lower melting point than the titanium, then the laser is going to last longer. If you are constantly keeping printing titanium for it, the laser is going to last a lot less amount of time. And then the laser itself, in different machines, are designed differently.

So, for example, the SLM machines, they do better with the titanium powder, simply because the default laser produces the very high impulse. It doesn't produce the lower impulse to begin with. So, in those cases, the aluminum gets over melted, and the solidification basically has an issue in those cases.

So what usually these printing bureaus do, they spend a lot of time in characterizing these machines with the material. And they are very sort of ritualistic about using this material on that machine only. They don't use a different machine because the costs can quickly ramp up to a very large amount if the machine and the material hasn't been matched up properly.

AUDIENCE: [INAUDIBLE]

PRESENTER: Exactly, replacing the material is the other big issue. So in case of the FIT that Autodesk owns that, they have large feed systems that goes inside these machines. So you have machines lined up, and then you have a lot of the powder inside the big bags, and the feed system goes directly inside those machines. So once they basically characterize the machine for a given material, that's the only power that goes in there. Nothing else.

And then, obviously, you know, all of us are worried about the cycle time. We want to have the least possible cycle time. And so, basically, having the least possible cycle time means doing something about reducing the cycle time. And a big part of the cycle time is the cooling itself. The cooling makes up almost 61% of the time that the you're going to spend in the overall cycle.

And that's one thing we can control with the help of the optimization, that we can basically design the better cooling. We can reduce the volume, and we can design the better cooling. If we can control the inner volume of the mold itself.

So basically, the starting point is basically beginning with what do we know about it? And often, people who have some experience with the mold flow, and like Jay that they have hundreds of years of experience with mold flow, and Gail.

[LAUGHS]

So they can basically do the quick simulation and can tell you that the this is-- can basically predict the map of it and can give you a rough idea about it, what this part is, what it's going to do if we inject it using a given polymer.

And so, basically, you have a gate location, the gate from which the polymer is coming in, and so, basically, there is a hot polymer that's entering inside the mold from this location. And as the plastic flows inside it, you're going to have a temperature map-- temperature is obviously going to be higher where the plastic is coming from. And the temperature is going to be lower on the other side of it.

If you have multiple gates, then you're going to have like a different type of temperature map inside the part. So that is something we can predict. That is something we know in the beginning.

And what also we know that what is the clamp pressure that we expected, we also know what is the pressure inside the mold cavity during the packing phase. So what is the packing pressure inside the whole cavity. So those are the things we can basically predict, that that's basically is our beginning point, prediction of the-- what are we making with this mold? Can we simulate it? And the mold flow is a very handy tool to predict that.

And using that, you can basically design cooling lines. So that's, again, the functionality starting 2016 release inside the mobile flow that you can basically create the cooling lines and see that, if one configuration works better or the other. So once you know how the part flow of the plastic and inside the mold cavity, you can come up with a decision on that, you know, what sort of cooling lines it was for you. So we're not going to optimize the location of cooling line, but we're going to optimize the other side of it.

And then you can use that data, the clamp pressure data, you can use the data coming from the pressure that is coming from the mold cavity, the polymer pressure, and apply that on the mold to see that what is the level of stress is on the part. Now, you can do in several different ways. I mean, you can basically create a stress map directly from the mold flow data.

In this case, what I did, I created the zones of the pressure inside the mold cavity and applied average pressure inside those zones to produce this pressure map. And I added the clamp pressure on top of it. So you can simulate the core and the cavity using that.

And then you can run the analysis, the inner volume optimization in tools like the Netfabb optimization, and it will come up with this type of design. So this design promises to support all those loads, but, I mean, obviously you're going to see that the temperature is really high inside of this cavity. And if you go with this design, the conduction is going to be really weak.

So you can basically improve the conduction inside the design by filling this cavity with the thermally conductive material. So that's one way of approaching it. You can fill this space with thermally conductive material like some type of copper, or-- so we said-- let's take one step further.

So now that we know that there are some known issues there, this volume could be compacted even further. So can we do something about it even better? So this was sort of the initial design that we-- so we told the initial design, we designed the cooling lines around it, and then we filled it with the lattice.

And we basically optimized the lattice using the loads that were applied. That was the initial step for us. And if you're OK with filling in with certain conductive material, your journey stops here.

AUDIENCE: [INAUDIBLE]

PRESENTER: Yes. And that's one way. I'm just coming to the story, basically. It isn't complete yet. So basically I'm going to quickly just run through a quick-- so basically, this is what we did. We designed the cooling lines, and we came to the initial design.

So now, what you can do, you can get some suggestions, either a generative tool on the what the next level of design should be, or you can come up with it looking simply at the temperature map of the part. So there's a lot more lessons on the temperature map of the part itself.

So one of the lesson is that if you look at the temperature is really high in the red area, and it needs a lot more removal of the heat, temperature is relatively low on the other side of it. And so you want to avoid, in this case, recirculation of the heat, but this might change if you like other good locations. So for this particular case that you can basically reduce the recirculation or the heat by removing some of the material. So you can basically customize the inner volume, and you can come up with the design, something like this, that basically has just the right amount of volume at the right location. So you can provide the proper thermal conduction in that area.

So, basically, the core and cavity now would look something like this. And so, basically, what we did, we did the initial design. And we did the injection molding simulation to see the distribution of the pressure, distribution of the temperature. And in a conventional mold design that we begin with, that we started with, we apply all those concepts and basically come up with the sort of initial guideline design.

And with that guideline design we basically optimize that design by removing the material where the forces were not causing any structural integrity issue. So removing some of the material and making sure that there's a proper amount of material available in the area that there would be conduction is proper so this is going to be over-- there's a new design, and the improved design, from the previous design that has been used.

And in this improved design that we basically customize the inner volume. So this is not done by the optimization [INAUDIBLE], done manually to increase lattice volume in the area where we want to improve the conduction. And in the area that we want to avoid the recirculation of the heat, area that are anyway getting cold far away regions from the gate location, we basically, essentially, remove the material from there to avoid the water lines to redistribute the heat. So the heat basically goes directly outside of it.

And then the other thing that we often do is the cleaning of the geometry. So basically, we want to make sure that the end design is also that we can represent properly, so you might want to do some cleanup work, and that's what we did to make sure that all the surfaces are right, the design looks as expected, and so that we give a clean geometry to the 3D part printing vendor.

And so this is the kind of geometry we supplied to the final design vendor that this is something that you need to print. So this is the new design. So what we did in the new design is took the mold flow results, applied that as a basis for the optimization, optimized the design, at just the inner volume, and then, finally, we basically cleaned up the surfaces to come up with this final design.

And once you have that design, once you've basically identified the design, the next step, basically, goes to producing the proper support structure. And so see that there, one of the issues that you can see right away here is that they since there is a little nozzle here sticking out, it's causing so much of extra support material to support it.

So if you want to reduce the cost of 3D printing in this case, the answer is really simple-- if you can cut this as much as you can cut this in size, your cost of 3D printing is going to go down pretty dramatically because so much of support material is basically, is right just underneath it.

So basically, you know, know, there's always a small opportunity for improvement, even when the design is done, even when the design is final, for, basically, can we make some minor improvement to make sure the end design is printable at the least amount of cost.

And the other thing that we do is, inside our Netfabb tool that we have like a printer envelope for a lot of the different printers. So you can basically really select the-- so EOS printer, in this case, I just selected from the menu. And basically it brought up the envelope of the printer and generated a supported structure that's relevant for the EOS M290 machine, in this case.

Then, basically, doing the distortion analysis to make sure that the distortion is not beyond the expected zone. So you want to make sure that, if certain parts are getting distorted too much, if that is the case, you want to have additional support in that area so that these parts as supported. Otherwise, they see that the kind of amount of bend that is happening. So in fact, Netfabb took care of it by adding the extra support here, in this area, for that. So you want to make sure that this kind of thing never happens.

Then, the next step is, basically, finally, 3D printing it. And this is basically showing the EX1 machine that they agreed to do some of the work for free for us, so we thought that we can show their latest printer. So this is a different technology. This uses a binder jet. So binder jet is actually made of the two different materials.

So one is the regular material, and the second is a binder material. So binder material is basically infused inside the regular material. So it doesn't actually do the centering. It does the binding of the material to produce the part. And that, sometimes, is a cleaner way of producing the same thing. And it works. If it works for you, in a given scenario, then it's a fantastic technology.

And finally, we already discussed the inspection part of it, doing the inspection to produce the design. So this is the end design. So we started with a completely different bulky mold, and we ended here. And the beauty of all this is not what you see. It's also inside because, here, we have all the inner volume under control. So we can basically change the conduction and the convection properties as we want.

We can increase surface area by adding the extra turbulence, by improving the surface area, changing the surface in any given way. We can change the inner volume of the mold itself. They have like a lot more conduction, the heat is basically exiting in a certain area, and then we can also avoid the recirculation of the heat. We can basically say, we don't want any more heat in this area because the area is getting cooled a lot earlier in the phase, it has lower temperature to begin with. So why do I want to have my hot water to go and touch the part that has already cooled down. So we can avoid that in those situations.

So, in a nutshell, we can use the mold flow results that basically can form the basis to design of the mold itself. And we can use the lattice-based optimization, to improve the quality of the mold. That means remove the inner volume, work the inner volume to have the better thermal properties, and have the less cost. So remember that when you are removing the inner volume, replacing that by the lattice, you also are, along the way, reducing the volume of the part tremendously.

So the not only is going to be, part, a lot more probably better, but it's going to cost a lot less to print. So the in 3D printing, the complexity comes free. So don't judge it by the-- there's a lot, all these jagged lines there. Because in 3D printing, it's all about how much of the volume we have to print.

So less volume means less cost. And again, we made sure the structural properties are met. And the process simulation can really help you identify. And that's a very important step. You don't want to send it to the 3D printer and waste your $10,000 if you don't know that it's not going to work for you. If large amount distortions are possible, and if there are areas that you can already improve to begin with.

And inspection so basically, that's the last part of it, you know you went through all this process, just go one more extra step, do the inspection to make sure that they end part is of the quality that you originally intended for. There's no point in having like-- you did all the effort, you spent all this money, you made this mold, and now you're making the hundreds and thousands of wrong parts out of it, what is the point of it? One more step inspection will basically help you insure that.

So that's the summary of everything. And I will take any more questions. Yes, so this whole class will be posted on the AU website, and-- they are recording the audio, recording the video here, and they will take my presentation and post it on the website and it will be available to you.