EDGAR AGUIRRE: Thank you for attending this session. I know that the title said topological optimization, but it turned out that that was a typo. It's AM optimizing your workflow using Within and Netfabb. I don't know how that got in there, but that wasn't the original title. So if anyone is here looking for the topological optimization, we have it with Dreamcatcher. But at the moment, the current Netfabb suite does not have the topological optimization.

But I'll just go through the presentation. And so the title is Am Optimization Workflow using Within and Netfabb. And what I want you guys to get out of it is how does the workflow Within in the AM process, learning about the workflow with Netfabb in the AM process, and how to incorporate the machine constraints, how to use those during your optimization process. And also preparing geometries for the AM process.

So what is Netfabb? Netfabb is a software for preparing files for the AM process. It helps you reduce the cost, increase efficiency, and improves the part performance in the AM process. So what is Netfabb? Netfabb is a way of lightweighting component while keeping its performance characteristics as you would expect. So if you have certain boundary conditions that you apply to this part, Within would optimize that part based on those boundary conditions.

At the moment, before I actually wrote this presentation, Within and Netfabb were two different softwares. But now they're integrated into one software. And it's currently in Netfabb 2017.1. And now Within is now called Optimization Utility.

So this is the Netfabb AM workflow. It allows you to bring in a CAD file, import it using either native CAD files or mesh files. It allows you to repair the file to make sure that it's watertight. It allows you to edit, optimize, put build support structures before you even-- depending on the machine you're using, it allows you to pack a 3D packing or planar packing. It allows you to quote, and at the end you can export it out either as an .xtl file or a slice file.

Before we dive into it, it's good to know what your AM constraints are, your machine constraints. Each machine is going to be different. So you need to know what your minimum feature size are and what your printer can print. Also, you need to know your maximum overhang angle without applying supports. At 90 degrees, you know you're going to need supports. At 45 degrees, some printers can actually print that without supports. It really depends on the printer you're using. And also knowing your build volume for that printer.

I'm going to take you through the whole optimization process. So before I even do any type of optimization, I'd like to bring in the CAD file. I'll do a mesh check. I do a print orientation if I have to. And I'll do a remeshing. Netfabb can repair. It can also remesh for other tools, for example, like the optimization utility. It needs a good mesh in order to work with equal triangle lengths. We'll go through this in a minute.

So I'm going to start just showing you videos of what's going on here. Actually, I just turned it off here. Sorry about that. I think I turned something off. Did I turn that off?

It's on.

It's on? OK. Got it. Sorry.

First, I'm actually going to import a CAD file into Netfabb. And here I can actually choose how detailed I want the CAD file to be. And I could choose the precision of this file. Go ahead and import it.

So, print orientation. Sorry about that. Let's go back here. Eventually--

So then the next one is print orientation. So what I'm doing here is Netfabb has the ability to actually minimize supports. And it actually will go and look at the surface of the part and determine what orientation will give you the minimum supports. So that's something that you could do before you even go and take it into Optimization Utility. So you have three choices you can choose. It really depends on which surface area that you want to have a better surface.

In this case, I'm going to try something else, since I actually wanted to lay it flat. And this part doesn't really have a flat surface, so I'm going to use minimization, out box minimization. So it's going to actually pick an orientation that gives me the lowest volume in that position. I can go ahead and now move it to my origin as well.

In the next step, I'm going to go ahead and remesh this part. So Netfabb has the ability to reduce triangles and remesh. And why do I want to remesh? I'm actually choosing a maximum edge length of 0.25. And I want to make sure that all my edge lengths are the same in the mesh. Within or Optimization Utility can do it, but you can actually also do it in Netfabb.

And once you can bring that out, and hide the original part. And now you have a part that you can use for the next step of our optimization. So when you're taking it into Within, what you're doing is you're going to optimize it for lightweighting. So you need to define what your lattice and what your skins are in the part. You're also going to apply boundary conditions. You're also going to do a simulation on that part. You can do it on the actual part and also on the lattice and the skin that you've defined in Optimization Utility. You optimize, and then you can do some post-processing.

I have an arrow coming in from Netfabb. I have another arrow going out of Netfabb. You can actually bring the component back into Netfabb to check some of the surfaces to see if they're actually self-supporting .

So here I'm going to start the Optimization Utility. So I pick the part and I just bring it into the Optimization Utility, or what used to be called Netfabb. So here I actually can select how I want the part to look in this environment. I can go into lattice and I can pick what my unit size for the lattices will be. I actually want my lattices to be variable. You can actually set it to be not variable, but since we do want to optimize it, we actually want to set these conditions.

Now I'm just going to pause it here. On the variable, there's a threshold. 0.2 is my minimum threshold. That's going to determine what printer you're using. So when you're looking at your AM constraints, you also need to think about what's the smallest feature you can print out. So that's where you actually put it in the minimum threshold. So it's never going to go beyond that, so you're guaranteed that it's going to print on the printer you're using.

I also pick the topology that I want. And we have several topologies that you can choose from. Not all of them are self-supporting. There are some that are. But I've picked the x, which I know that will give me a high chance of it being self-supporting.

In the next stage, I want to pick my surfaces. So in this case, I'm going to pick the top one as a surface that I actually want to keep open. I want this area of my part to be open. And I choose this surface as a way of indicating that it's going to be hollow, or that any beams that touch it will be removed.

In the next region is going to be my skin. So I'm going to skin this part. And I can also make the skin uniform or variable. In this case, I do want to make it variable. Again, this is all just playing with your variables and seeing what works for you and your machine. So this is the final image. So you notice that the top is open. You have a lattice, and then the rest of the part has been skinned.

Let's go to the next one. Now again, in the original workflow, I said that you can actually bring back any part that you create in Within bring it back into Netfabb, and just do a check on your lattice structure. So what I'm doing here is I'm doing a cut view into my part, and removing a part, and looking at my lattices. I'm going to actually use the surface selection tool, and I'm going to indicate the down skin area. So I'm going to go to advanced selection. So this is the area that will need supports if I set it to 40 degrees. I'm looking at my lattices. Nothing's really green except maybe the center of the node. But that's OK. Once I do the smoothing, that area will not need supports.

So now that I'm happy with that, I can go back into Within and do my next-- so if there is a problem, you can always go back to the settings and maybe adjust the unit cell on your lattice. I skiped one. OK, here it is.

AUDIENCE: Do one more? Do lattices? How do you create lattice structures today?

EDGAR AGUIRRE: And then this next stage is where I've actually added the boundary conditions to my part. So here I load up a new boundary condition, which is going to be my constraint. What part will be restrained in this design? So I go ahead and just select the areas that will be constrained. And at the moment, they're all nodes, and you're picking nodes. So I'm just going ahead and picking them and doing a fill in mid area. And once I'm finished with that, I'll go ahead and apply the pressure of the forces for this part. And I go ahead and just select the areas that I know a force will be applied to this part.

So I've now selected both of my-- and here I add the force. I'm actually adding 700 pascal on each of the areas that you see there. I'm going to go ahead and run simulation. I can run simulation on the part itself even before I do this. This is just to see that the part that I'm actually going to optimize can handle the loads. But based on this example, I'm going to go ahead and I ran the simulation. It took about three minutes, but I actually quickly cut this video by three minutes. So what you see is you see the skin displacement and the lattice displacement based on that simulation.

This is the part that I simulated before I even ran the simulation on the lattice and on the skin. I ran it on the whole part just to make sure that it can take-- oh, go ahead. I'm sorry.

AUDIENCE: Did it apply that pressure? What direction did it apply that force?

EDGAR AGUIRRE: Right. So if we go back on the video, when you actually add the constraints, there's an X and Y and Z. And so I just set it to Z at negative 500 pascals, and it was pointing down. And you can adjust those parameters depending on where the loads are being applied to.

So let's see. It's the same one. Sorry. Oh, here it is.

I'm in the optimization. Here you actually can indicate how many iterations you want for the optimization routine to go through. There's all these other settings that you can apply to this part. So it's going to go through five iterations. And you could tell it to stop after 10 or 15 minutes. So after each iteration you'll see the lattice change. And here I'm just showcasing the information at the bottom.

We're actually on the third iteration, and you'll see some of the nodes getting thicker and other lattices staying the same. So it's strengthening the areas that need to be strengthened. So it's optimizing the lattice as well as the skin.

So here it tells you how much lattice has been increased and how much the skin has been increased as well in that graph at the end. So in the end, you'll get something that looks really organic. And you know that this optimized part can withstand the forces applied to it.

So in the next slide, I'm going back and doing some post-processing. I actually turned off the surface trim. Surface trim just removes the lattice from the surface. I set it to intermediate. I can change the beam counts, and I can do some smoothing. All these can be done after the optimization process.

If you set these before, it's going to use a lot more processing power, so it's always good to do this at the end. So once this is finished, you'll get a really nice looking finish on your lattice, and you won't get any protrusions that you saw before, the lattices sticking out of the surface.

AUDIENCE: [INAUDIBLE]

EDGAR AGUIRRE: No, it should not. No, because you're you're just applying a slight smoothing on it. So now that it finished, the next step is to take it back into Netfabb. And this is called Print Preparation Workflow. So you've taken it back into Netfabb, and you can do the wrapping or repair the lattice and the skin are both two different shells. So you need to actually combine them together, Boolean together. You can do mesh editing if you need to. Then you can bring it into a workspace, and a workspace is just a representation of some machines that we support.

You can do the build supports right on that workspace. You can apply parameters and tool path on that tool. And then you can export it out as a slice file.

So in this video, I just clicked on the top to export it back into Netfabb. So I get a .3mf file. And a .3mf file, we're using that because you can actually put material properties in that file as well as which machine you want to use. Right now I'm running a repair right directly from the work plane. I have a list of scripts that I can access down here on the bottom. So it's going through and just repairing any holes and just wrapping both the surface and the lattice together to get one shell.

And you'll notice here that I do have a volume now. Before I didn't have a volume. There was something wrong with it. And I can always go up and look at the platform overview to see that I do have a valid mesh. And once you've finished that, you can go and bring up a workspace. So we have all these workspaces.

This represents the SLM 280. So this is the build volume for that printer. I can just drag the part that I have, that I brought in, over to this machine. And once I have it there, I can now position in on the platform, move it above. I know that I'm going to print it in this orientation. So I can move it up a bit off the platform for the supports. Now if it was a flat surface, I probably wouldn't do that. I don't really have a flat finish on this one. I can go ahead and apply supports.

And Netfabb has standard scripts that come with it, or you can make your own. Here, I'm actually going into areas that I might not want supports, so I'm actually decreasing the minimum area that I would want supports. I don't want any supports in the lattice. So I can actually play with this minimum area so when I run it, I shouldn't get any supports on the lattice. So without any supports, everyone else except where I would have lattice.

And if you do see a support inside this part, you can always go into the list of all the clusters. And what a cluster is just a surface that has been given a support. So right now you have all these clusters. I can go ahead and choose a cluster to remove it, or I could just pick on that cluster and remove it the other way. And I've also applied a support on another area that didn't have supports, because it maybe I had the minimum area too low.

Here I'm actually picking my build strategy. And each machine has their own build strategy. Contour, hatching, these all the parameters that you would actually put in this machine. And you can actually edit them here in Netfabb and then send the slice file as well as all those parameters to the machine. You can have multiple build strategies in one parameter in these parameter files.

So right now I'm going to go ahead and just do an animation of each layer. And if you want to see how the hatching or the laser paths are going to be on this, part you can go ahead and just go through each layer. It can also be animated. So you see it's just going through each layer. The purple is the supports, the green is the hatching, and the blue is the contour of the part.

And once you're finished and you're happy with the results you, can go and export out to a file that the printer can actually understand. We also have other build workspaces that actually connect directly to the printer. So in this case, you would have to send the file to the printer.

So this was kind of a quick way of introducing you to Netfabb's AM workflow using the Optimization Utility that used to be called Netfabb but now is called Optimization Utility. And if you've noticed that they both complement each other, and I was able to do this very quickly using just one set of tools. So I think that's the end of my presentation. If you have any questions--

AUDIENCE: So maybe a couple of things for me to add. This is an automated way of optimizing the lattice structures. Based on the loads and battery conditions, the computer shows you how the densities should be on the lattices. There's another functionality in Netfabb which is called 3S, which gives you the manual control over how you want to create your lattices. So we have heard from a lot of companies, they say they don't want computers to do that. They want to decide how the densities should look in the lattice structures. So this is another functionality that's included here where you can actually finalize this. There's no optimization limit, so you have to do it.

The other thing I want to highlight here, the optimization is based on Nastran that's included in this part, which you see in a lot of Autodesk products-- Fusion and so on. So that's based on Nastran. And then one thing about the lattices, I think everyone who's sitting here maybe has probably some idea on how you want to use lattices.

So there are multiple reasons to use lattices. Some people use it for design, just to create an element of design. Alternatives, in the mechanical area, you can use it a lot for performance reasons, too. Performance, but also cost saving. So if you were doing 3D printing, you want to remove a volume. So one way of removing the volume is replacing it by lattices.

Obviously, you want to make sure your structure doesn't break. So that's why we have the lattices with optimization. And that actually ensures that your product is stronger than before in many, many cases. So that's one thing.

The other thing is through lattices, you can also increase the surface area. So for the automotive industry, there are probably more cases for that, you can actually use that to increase performance of your parts.

You're using less material, and you're increasing print time as well.

You reduce your material, so you reduce your printing costs. I think we have multiple examples of you can reduce the weight, and reduce the costs, but also increase performance. So that's why you could use lattices.

AUDIENCE: [INAUDIBLE]

AUDIENCE: [INAUDIBLE]

AUDIENCE: [INAUDIBLE]

So one of the things that our software and hardware team is experimenting with is metal plating can be expensive. So some companies buy metal printers, some don't. So one of the things that they're experimenting with, and you'll probably see more often nowadays, you print in [INAUDIBLE] and then you cast it. So that's definitely an option to consider too. I think there was another question.

AUDIENCE: [INAUDIBLE]

AUDIENCE: Just quickly, the analysis that's being run by Natran, is that a linear analysis, or is it a non-linear analysis? I know a lot of materials that are alloys which are non-linear by nature, so is that a non-linear analysis?

EDGAR AGUIRRE: It's a linear analysis, yeah.

AUDIENCE: [INAUDIBLE]

AUDIENCE: [INAUDIBLE]

AUDIENCE: [INAUDIBLE]

AUDIENCE: Yeah. So actually this morning we had a session about that, like mechanical analysis of the lattices. If you haven't been in the session, I suggest it's going to be available on demand, so you might want to watch it because Dan, who is Andy's colleague on the consulting team, earlier was showing how you can work on the mechanical analysis after you create lattices, take it out and so on. Andy is from our advanced consulting team, so these folks have a lot of experience working with different types of customers, different types of lattices, so if you have any questions.

EDGAR AGUIRRE: Any more questions?

AUDIENCE: What kind of technology is this?

AUDIENCE: What?

AUDIENCE: What technology is this? Direct extrusion, laser centering, [INAUDIBLE]?

AUDIENCE: [INAUDIBLE]

EDGAR AGUIRRE: The tool pathing, well, there is a tool path utility that comes with Netfabb. But the tool paths you saw there, that you saw on the workspaces, were all predetermined tool paths. But Netfabb also gives the ability to make your own tool paths as well. I think any machine, yes.

AUDIENCE: So can you actually have it export to G code?

EDGAR AGUIRRE: In some cases you can, yes. In terms of the metal printer, you're just sending them a slice file. But there is ability to actually control how that pattern of the laser gets processed. For FDM printers, we can send a G code to an FDM printer using the software. You saw part of the list of the printers that we support, but there are EOS, [INAUDIBLE] down to our Amber printer. I guess our Amber printer started the whole development process, developing an ecosystem around this printer, and we're applying it to other printers as well. I'm sorry it's kind of short, but--

AUDIENCE: It's fine. But that's why we wanted to leave some time for questions. So how are you planning to use lattices? Just for our understanding, how would you use lattices in your process?

AUDIENCE: Mostly for material optimization.

AUDIENCE: Material optimization?

AUDIENCE: Less waste.

AUDIENCE: Yeah. That makes sense. Especially if you print them out, it can be very expensive.

EDGAR AGUIRRE: Are you from Made in Space? Are you guys just strictly using polymers?

AUDIENCE: At this point, yeah. But we're working on expanding to others. But yeah, we're looking at ways to optimize material usage for polymer printers.

AUDIENCE: Anyone else going to use lattices? [INAUDIBLE]

AUDIENCE: [INAUDIBLE]

AUDIENCE: That makes sense. Actually, we have examples, too. Aerospace, if you save material, you save fuel, you save a lot of costs, so that's how you save. Yeah, we have a lot of customers in aerospace.

AUDIENCE: So has anyone been using Netfabb before? You?

AUDIENCE: I use it to fix all my SDL files.

AUDIENCE: So I think at AU, they spoke about the new release too, right? Here, today? So you will see we have a new release coming soon in a couple of weeks. So what we showed you is in there, but actually more is coming out in that. So what Edgar showed you is in the Netfabb ultimate. So if you want to go our website, see more, you can see there. But also, just one of the goals is to add more and more based on the customer processes and simulation in the cloud after you create your lattices, and you've built the support structures so you can test those too.

And also then, I think something is coming for the post-processing too, right? [INAUDIBLE]

--simulation in the cloud that you're going to have, and then Netfabb Ultimate. You had a question?

[AUDIO OUT]

AUDIENCE: So that's a great question. We have Within Medical, just so you know. So what we showed you is the standard Within that's gone into Netfabb. This is, I would say, automotive industry, aerospace industry. But also I see a lot of interest from industrial machinery. Obviously, it doesn't prevent other customers to use it. Like under armor used Within technology to create the shoes that you probably saw with Fusion and Within. And they sold like 3,000, I think, shoes in an hour or so. That's what I heard. So they use it. So we see a lot of interest in the consumer too. But I would say for us it's automotive, aerospace, industry machinery.

But medical is another industry that we actually work with. We have Within Medical. So this was not part of it, but it doesn't have optimization, does it, the Within Medical?

AUDIENCE: Within Medical uses pore sizes, and then it has a variance on that pore size. You get a bit of control over that, but it isn't using optimization. So you get a special lattice called trabecular, which is very good at osseointegration. And the user interface is very focused at producing parts quickly and efficiently. So that's the main idea behind Within Medical.

AUDIENCE: Any other questions? So yeah, if you have any questions, we are here. Or I would say if you go to our website, you can always contact us through the website, too, if you have questions. And Within Medical is also on the website, actually.

AUDIENCE: Thank you for attending.