PRESENTER: Alright, so I'll get started while people-- there's a few people still coming in. But it's pretty much time to go here. So I think we'll be a little bit shorter than the full hour and a half. At least, that's my plan. If I talk too much, then I've got a little timer here that'll stop me. So I guess I'll just start off with an introduction. I don't know if any of you guys-- I've been to AU three years now. This is my third. So if you've been in my classes before, then you recognize my face. If not welcome.

My name is David Ponka, and what I do is I work for-- right now I work for a reseller, IMAGINiT Technologies out of-- I'm out of Calgary, Alberta, Canada. So I left, and it was 15 degrees Celsius. And I'm going to go back, and there's going to be two inches of snow on my car. So I'm enjoying the warm weather here while I get it.

I've been-- I've been with IMAGINiT just over five years now working with Autodesk products solely and doing a lot of work there. So I do-- I do work with vault and vault consulting and implementation. A lot of training with Inventor and AutoCAD, of course. And there being a new oil patch up in Calgary, we do a lot of plant 3D as well. So Plant 3D, PNID, and then you, kind of, mix them together and you get implementation's that' do a bunch of-- a bunch of everything.

Before that, I did some time in office furniture, things like a steelcase type of a company that was-- it was like that, but smaller. I was in the product engineering department. And then prior to that, I was in Telecom, kind of, a mixed bag of engineering and industrial design. So I am an industrial designer by if you looked at my piece people from school, but I've been, kind of, living my life mostly in a technical role. And I've always been the guy to help out with software in the office when I have been working so. So that's me.

We're here for the shape generator Kind of, just to break the ice, I guess, a little bit, everyone in here has probably-- you've obviously heard of the shape generator. But everyone using Inventor 2016 or 2017? Yeah. Anybody using something older than that? No, right on. OK, you're in the right place. If you're still using 2015 or older, then you're, kind of, out of luck when you go back to the office unless you can upgrade.

So I'll go over through the first slides. These will be-- I mean these are pretty routine, right. It's just a shortened version of what you saw from the course description while you guys all signed up. So I usually walk the room a bit more, but I'm going to get blinded by the projector so I'll probably stick to this area. So what we're going to do is just look at the shape generator. It's Inventors first foray into generative design.

If you guys have had the chance to use it, that's great. And what we're going to do is we're going to look at, again, what it can do, and then we'll look at new features of 2017 and, sort of, the leading edge of-- Inventor is not quite there in terms of being able to implement it into full production. But in certain aspects it is, and in certain cases, you could probably go that way. So we're, kind of, right on that cusp of change. And that's, kind of, where I wanted to focus this presentation is just on how we can take the shape generator, produce a result, and then not go back to our parametric drafting tools, right, or our normal-- our normal tools.

So, so that's, kind of, where we're going. So we'll look through the first, these four bullet points. And the first bullet point is interesting. I put that on there, because, kind of, the lack of information around there is, kind of, the last major problem, at least on the technology side. I shouldn't say technology, but in terms of the information we're lacking to be able to do a generative design. Whether it's in Inventor or whether it's in another package is, OK, when we go to produce this now, we've got some issues with material properties that we aren't quite sure about. So we'll look at that, OK.

So why are we here, right? OK, so to get the most out of the class, basically, is hopefully you guys have a strong foundation in part modeling, assembly modeling, those types of things already. If you are a simulation expert, you're probably in the wrong class. We're just, kind of, if you've used Inventor and you want to look at the new tools and, kind of, see how it works in there, but maybe you're not an engineer that's simulating or doing failure analysis or something like that, then what I'm probably going to do is tuck under your level of understanding.

All right, and so there's no stress analysis experience necessary in this. So if you are afraid of maybe the reality of the shape generator environment and having to apply forces and things like that, I'll go through that. Again, you may have seen demos already. That's, kind of, going to be-- we'll be into a little bit of a demonstration there for a little while. But I'll try and keep it to simpler terms, and I'm just going to explain some of the foundational ideas between or behind the information that we need to know when we're doing a stress analysis on the material side and then on the forces and things like that, that will apply as well, OK.

And then, of course, we're all here, because we want to be able to include this into our work. Whether we're able to do it little by little, or whether we're able to do it on a larger scale is, kind of, to be determined in each individual case. So that's the idea here. So I've got a bit of a theme going. Every year I try and do a little theme for my questions or where I start my slides for a second, and this one is just, kind of, because generative design and the shape generator are, sort of, leading edge technology. I went back, and I thought, all right, what does the history of design look like? And you can find these on Google, Google Images.

But does anyone know what this is? You probably-- you may have run across it before if you ever do a search for historical val-- I forget the terms I went in, and it was a few pages down. But so this is a Ford's factory. They're designing bomber. It's like us during World War II laying on their plans. I don't think we use this type of drafting hardware anymore, so couch cushions. But anyway, so let's just, kind of, jump back to this.

So I had some questions for you guys as, kind of, an interesting to get a feel for who's in the room, and what you're making. So out of just maybe a show of hands who here is using as a primary design material, plate material-- plate or sheet flat material. OK, so probably a good chunk of you. And then on the other side on the flip side of that, how about casque materials or molded materials. So our molded processes. Yeah, OK. So we've got a fairly even split. And there's a bunch of you probably doing both.

It's-- they rarely live completely independent of one another, but that does have an impact on the shape generator in terms of how you can use it. Because if you're primarily going use it for sheet material, then you might take the parametric path and you might model with existing tools. If you're going to take it through the, if you're molding this and you're creating a 3D solid, you're going to have to go a different route, OK.

Out of those in the room here, so maybe what we'll do is I'll do a show of hands again. The materials that you guys are building right now, or you're designing for. So metals, we'll probably see most people raise their hand, right, steel, aluminums, stainless steels, those things, right, most people. Plastics? Yeah, a bunch of people. All right, so it gets a little bit-- a little bit more rare here, composites, like, [INAUDIBLE] composites, lay ups, things like that, yeah. And then any fiber like wood, fibrous materials? Anyone modeling anything in that industry or that material? No way. Oh.

AUDIENCE: [INAUDIBLE]

DAVID PONKA: Everything eh? Well, it's true. So when in office furniture-- I said I came from that-- we did textiles, steels, aluminums, woods, plastics. So, yeah. I mean, really it just, kind of, I'm trying to get an idea of is there a majority of people in the room that are using steels or aluminums and things like that, because when we go to find our material properties, that's-- that's a key thing that we're going to need to know, of course, or we're going to be very vastly different.

And then the one I'm afraid to ask is, in here how many of you guys have used the simulation environment in Inventor professional? Yeah, so a good chunk of you. OK, so it's going to look very familiar. If you've used shape generator, you've seen that already. But we'll go through this anyway. It's, like I said, just, kind of, be warned that it's a hard place to put this class. That's why I started it right in intermediate, because we are doing simulation. But we're-- I'm not expecting you to know any of that stuff on the front end, OK.

So just, kind of, an introduction to generative design. We've all most likely heard of it. This is a picture from a software called Autodesk Within, which is something we can't sell as a reseller at IMAGINiT, because it's just-- it's not in the standard portfolio. But this is, sort of, the leading edge software that does generative design. Generative design is a rules based design, right. You have some, sort of, a volume or a mass, and you say, OK, I need to do this here, I need to have this stronger, or this thinner. Or what I found interesting about this picture from the website is that you can actually within this type of in generative design specify where you want center of gravity and a lot of physical aspects of this.

So who remembers this? Two years ago this was Jeff Kowalski's keynote at AU. And he showed this-- that was of generative design, kind of. This is done with Autodesk within, and it was-- it was a great talk and very inspiring. What I did is I tried to recreate this in Inventor and see what we came out with. So if we bought shape generator in Inventor, it's a generator design tool, how close could I get. Well, I'm not using within. But this is what I came up with. OK, it's not quite the same. What we saw in the last one was-- it's manifold, right. So I'm missing this pipe splits into a y and comes out at the top. All I really did is put a pipe through and then apply some forces.

And some problems that are present, right, we've got floating-- little floating bits here and there. And it's not nearly as beautiful and elegant as what we saw in the last slide, but with a few settings and a few tweaks we could make this a workable product, OK. Maybe not as appealing.

All right, so like I hinted at before is that your design process, whether you're creating flat parts or volume illness voluminous parts, it's going to-- it's going to affect your process. So when we get through-- I'll show you-- I'm primarily going to stick to a 3D part instead of a flat part, because that's where things get interesting with what we can do with the result. But when you have the flat parts, while the options are similar, you're more likely just to take a straight road to-- back to your party environment to modify that part according to what you got with your shape generator.

OK, what's interesting here is that we're right at the convergence of these technologies. All right, we've got 3D printing coming together, right. So when we do a generative design on a 3D part, the only way to make it is going to be to print it, right. We can't cast something like that, because we've never-- you might be able to do some sand casting or something like that. But for the most part printing is going to be the way to go, because you won't be able to create any other process around the design-- the type of designs you've got.

And so we've got the 3-D printing, which we could say is pretty-- it's maturing. It's still-- it's still-- the speed of development in 3D printing is insane, but we're pretty solid with the foundation of what we've got, especially when we're dealing with single materials, and we're dealing with smaller parts sizes. There's a lot we can do there. So the generative design technologies, the 3D printing technologies, they're all coming together. What we're sort of lacking, though, is the materials, right. So like I hinted at before.

So if you go and you want to say I want to 3D print something, yeah, we can print it out of metal. We can print it-- there's some samples here. So on this slide, these are all taken out of Shapeways website, but if you-- and has anyone used Shapeways or seen them? So I've got some prints up here, actually, I'll show you later on. You're welcome to come and take a look at them, but some things that I've printed off through Shapeways of generative design and things like that.

But what we've got up here is some steels. So this is a stainless steel on the left. There's bronzes over here. This part is made out of aluminum, and then we've got plastics and multicolored materials in there as well. So they're coming along, but if you want to say make something out of steal or make something out of aluminum, whether you go here or whether you go somewhere else, the lack of material information that you need for stimulation is, sort of, striking.

OK, so if we take a look here, if you can find that information or if you can test for that information, that doesn't necessarily validate it immediately, because what we're missing or what can change there is that a 3D printed material is-- steel, for instance, comes in a powder form. The aluminum also comes in a powder form, and it's las-- the laser. It's a selective laser process that hardens it and fuses it together like a weld.

And so what it can do is that's going to vary by producer. The-- by process, so if there is a different optional printing process or a printing machine, right, both of those will impact that. Even between production runs, machine settings might be different. There might be-- you always have to, if you're doing a CNC process, you have to set your offsets and zero a machine. Well, the same thing is going to be true of a CNC-- a 3D printer, pardon me, with temperature and positions, and so you could do print to print and have different values for your material properties.

OK, and then structure is also a big one. This is not-- this, again, depends on the material in a way, because if you are printing from a metal material and it's a powder-based material, whether it's steel, aluminum, or even plastic, it's fairly homogeneous. All right, but if you do a PLA or an extruded process, then you have almost like a grain or a structure to your 3-D prints, right, and that will affect also your physical properties. And that could be by design or by part.

So the takeaway there is that you've just got to be real careful. Examined each individual case and make sure that, OK, if I am generating this from a generative design process using Inventor, that you see the results and then you see your print and you know that, kind of, OK, this is in line with what I would expect. So a quick sanity check.

So this is off of another website, Spark's Engine. This guy did a test on these materials. And really all I wanted to point out here is that with the same part, but a different structure of this extruded process, you had very, very different results in tensile strength, how they would how they would collapse or crush under pressure. So those are the types of things that you need to, kind of, be aware of, and that for a lot of printing processes, we just don't have the infor-- enough information yet to be able to do a generative design and say, oh, it's just plastic, right, create that design and put it into use. Because this could fail very easily, if we don't know or if we haven't tried it, OK.

So I mentioned Shapeways. They do have a lot of information. So they're one of the few that

I've found that does post physical property information for their products. There's-- I'm sure there's others out there. This isn't 100% comprehensive. They're still missing a few things on here, like Poisson's ratio is not on there. So what I've done is just-- I just put up a quick show of this. And they've got this information in here with their units, and this is what you need to be able to plug into Inventor to be able to generate your design.

And then for something like Poisson's ratio, if there's any missing information in there, you can always go to another website that, at least for the raw material-- Matt Webb has the actual-- this material here. There's a shortcut right from the website onto that material, and you can actually extract that information from there. If you look at the numbers, it's a little bit interesting, because they're quite-- they're a little different.

Aside from Poisson's ratio, if you start looking at Young's modulus and a lot of those physical properties, they vary from that spec sheet to the one that's on that web. I would always go back to this one, because in this case it was tested for the specific process. So if I can point it out right up here right where it says through selective laser melting, OK.

All right, so just, kind of, a primer. Like I said, I won't get into this too deeply, but you do need two physical properties to do an FEA analysis. Young's modulus and possum's ratio are the two primary things you'll need. And for those that have maybe have not dealt with these values and what they mean, it's always nice just to, kind of, go back to a picture. So Young's modulus is basically how a product deforms against a certain stressor or strain force that's been introduced to it. OK, so an elastic band is-- it's got a fairly linear change in shape as you apply a force to it.

Poisson's ratio as you can see here is, sort of, the change of strain in an axial direction as it's having-- as it's facing a strain in a lateral direction. OK, and that's a ratio. So if you can find those two values, you can likely do your FEA analysis. You will need one of these two in addition, yield strength or ultimate tensile strength. And what we'll see is we'll see when we get into Inventor where we're actually going to plug these in, and we're going to select which method. This has to do with the safety factor when you're simulating.

And then so this chart, here we can see there's my yield strength. So yield strength is right where my part actually breaks, right. So it deforms up, and it's elastic meaning it'll go back to its original shape after the force is taken away, right, until it hits the yield strength. At the yield point, right, it permanent-- it gets permanently deformed. And it doesn't go back to its original shape at all. And then ultimate tensile strength is, sort of, the maximum stress introduced or force introduced before it breaks. OK, and you have a clean break. All right, and then so another visual. So here's your fairly linear change in shape for a force applied in Young's modulus description there.

OK, another good thing to understand about material properties is their makeup or their structure. OK, so we have three primary types that we can select in Inventor, isotropic materials, transverse isotropic materials, and othotropic materials. And really it's-- all this is saying is it's a description of the grain direction. Inventor can only simulate-- if you've used Inventor FEA when we're going to use the shape generator, it can only simulate othotropic-- pardon me, isotropic materials. So homogeneous materials, no grain direction, it doesn't matter which direction you apply a force into this thing, it's going to behave the same way.

As soon as you get into a grain structure, if you're orthotropic, wood is classic, because if you stretch it in one direction, it'll break at a different point. If you stretch in another direction or parallel to the grain, each direction it's going to have a different Young's modulus, a different Poisson's ratio. So you can't simulate these in Inventor. You can simulate them in other tools, some mechanical and things like that. Things that go over my head. I couldn't set that up. I'll show you where the entry points are for the Inventor's in Inventor, but we're really just dealing with isotropic materials.

And then the tricky part of this is that there really is no 3D printed material that is 100% isotropic. There's-- they come very close, but because they're all done in layers, there can be a little bit of a difference based on the layering direction than there are in the sight of the transverse directions. For things like metal, so this one's here-- if you come up and take a close look at this later on, you can see the layers. But they're pretty much-- they're pretty much welded together, and it's pretty much homogenous. So you can, kind of, assume that those values given from the spec sheets across every direction will work.

If you are, of course, if in doubt, I'm not saying-- and when we go through this, I'm not saying do generative design and be done with it. You always want to go back and simulate and verify, right. So maybe in the future we'll get to the point where we can self verify or run a check right away afterwards. But we're not-- we're not there yet, and I wouldn't say that it's ready to go.

So everybody familiar with these dialogue boxes? If you've ever edited a material in Inventor, this is what you've see, at least on the physical side. I'm not showing the appearance settings or the general setting, but what we're down under here is under the mechanical drop down. And you can see, so if I select isotropic, transverse, or orthotropic, you get different selections. And so there's your, kind of, your illustration of where you're going to see one Young's modulus value and one Poisson's value versus three. So when you find those values, drop them in there, create your own-- your own material, and then away you can go, and you can create that in Inventor.

OK, so I was just going to show you that real quick. This one won't take that long. Does anybody recognize this. This is-- this is probably a little bit more obscure. This is Tamron. You probably get it from the hyperlink. So back in the 80s, Tamron I guess in Japan or wherever they are, they're on drafting tables. I don't have one of these anymore. My dad used to.

But let's-- let me jump back here. OK, so if we-- if we just take a look at where we want to get these. By the way, who uses a service for printing. Do you guys in terms of a split, do you print out of house or in house?

AUDIENCE: Both.

DAVID PONKA: Both. Whenever-- you probably have one or two materials in house, and then when everything needs something different, you go out, right? Yeah. Right. So I just, kind of, wanted to take a quick look here. So we've got aluminum and metals brass, bronze, steel. Shapeways is a little bit more on the artistic side. So if you're a creat-- you have a creative flair, they like that a lot.

But if I-- so I'm looking at the aluminum stuff and scrolling all the way down to the bottom here is where you'd find that material data sheet with the information on it. And so what we can do is we can pull out-- pardon me-- max tensile strength, my yield strength, right, and then I've got my modulus of elasticity, which is my Young's modulus.

And so if I take these now and I go to Inventor, what I want to do is for those of you that maybe haven't created your own material library or you're going away from standard, you might need to create some 3D printed materials. If you've got a custom library you can access, just click this-- I don't know globe, I guess, up here. And then inside you'll be able to edit your material.

So if I say, we probably want to start from an aluminum value, you can see I've already entered one in here printed. But I could just pick any value, and then I'm going to add it. You can only edit materials up in this top area called your document properties. So the-- it's not a materials course per se, but it is something we need to know. So just, kind of, I'll cover the basics of this.

If you're going to edit your materials, add them to your document materials. You can't edit them from this-- the library view at the bottom. But I'll hit edit here, load this up, and it usually just takes a second. And they're on my physical tab, I can expand my mechanical properties and select my isotropic, or if it's orthotropic tropic, and you can see how it changes those in here. So if I say in here my Young's modulus, now I'm going to have to go back to the web site here. So 64, right, and you just enter these and fill them out. Pardon me. There we go. And I think it was-- yeah, GP. OK, it'll do the conversion for you. So whether you're putting it in metric or whether you're putting in an imperial, right, just include your unit string, and it'll do the conversion for you. You don't have to worry about that. I'm not going to run through and finish this off completely.

Poisson's ratio here is actually the same. So it's 0.33. If I go back to-- I've got the tab up here for Matlab, and this is the material that they specified her for the aluminum. And if I just click on view data, you can get this extra information whether you want metric or imperial. You know there's a lot a lot of available information out there for getting materials, OK. So Poisson's ratio is right in-- right in this area here. So 0.33 for both, OK, since it's not unit specific.

If I take a look at my printed ones or my printed property, and I've just got that all filled out in here. OK, once you have edited your property, and you've got all of your values filled out, what you need to do if you want to make it available for all of your other parts so that you've got to--- you can start editing this. If you just saved this document right now, this material is only saved in this file, right, in this particular file.

So you really just have to right click, and then add it back into one of your libraries that you have available. And then you'll have it available not on your own machine at least. And then if you're set up for it, if your CAD administrator set you up so that you can access that through your corporation or through your network, then everyone in the office will have access to that.

OK, so that was just a quick aside there. All right, so before we before we get right into the shape generator itself, what materials are you guys printing? So how many of you guys are printing metal right now? Yeah? Are you using prints in production or for natural products like shippable product?

AUDIENCE: R&D mostly.

DAVID PONKA: R&D and stuff like that, OK. So there's not very many people. How many people are printing plastic? Yeah, a lot more, right. So that's the classic one. Plastics been around the longest. It's got the least-- the least unknowns and the greatest acceptance. We've been doing that for 10 years plus, right, 10, 15 years. OK. So.

OK, so what I'm going to do is when we go through the shape generator, I'm going to use this race pedal assembly from Tilton. The nice thing about the Tilton's stuff is that they have stepped models on their website, and they are actually simplified versions of their shipped product. So you can see-- you can't really see because it's black, but in here there's some ribs and things like that have been machined right out of it, but they don't supply that information to you. They just give you a solid lump, which is perfect for what we need in the example, because what we want to do is we want to design our own structure in there that will remove that material for us.

So the shape generator is a parts only view. OK, If you want to if you want to perform a generation on [? weldments ?] or an assembly of some sort, right, you're going to have to get it into a part form whether you do a shrink wrap or some operation. But really we're looking at the part level, OK. If-- once you've got your part, we'll preset out material properties, OK. We always want to have the material that's going represent the part modeled in the part, [INAUDIBLE] our modeling premier manufacturing process.

And then, kind of, an interesting thing there is that the simulation engine. It is a simulation tool that is creating your shape generation, and it is the Nastran shaped engine so that was added. Autodesk bought Nastran a couple of years ago. And so that's been worked in now with the shape generator, and that's if you-- if you ever-- you know you bought a license, it'll pop up, and it'll tell you've got a Nastran license in use, OK. Although you don't-- I don't believe you need Nastran in CAD or separately to be able to use shape generator. You just need Inventor Pro, right.

OK, so if you've set your material properties on your part and-- I'm just going to stand back here so I'm out of the light-- and you want to do some experimentation what you can do is you can actually just like the Inventor simulation environment, you can override your material. So if you want to test different aluminums, steel, maybe steel against brass, you can select your material, and then just like simulation say I'm going to try this and then I run the simulation on it, except we're doing generative design.

Any time you see the little exclamation mark, it either means you're using the generic material, which you should never be doing, or you've got a zero value somewhere for Young's modulus or Poisson's ratio or something like that. OK, so if you see that on another material or a custom material that you've made and you've got it in your list, then that's an indication you've got to go back double check your properties, OK.

And then the last thing on the material page is your safety factor, and so this just brings us back to that slide where I had the formulas for the yield strength and the ultimate tensile strength, right. So based on your material you're going to select one of those to move forward with your job-- your shape generation.

OK, as a quick overview, I apologize for you UFEA users. You know all this stuff. What you've got is you've got to before you create your generator design in Inventor, we need to fix certain things in our design. So we've got three types of what we call constraints. These are not assembly constraints. These are generator constraints or simulation constraints that basically tell the simulation tool where-- what can move and what can't and how can it, right.

So our fixed constraint might be something like this on the bottom. So if we were actually going to mount this to the floorboard of a vehicle, right, and we wanted to be able to simulate that, we would do maybe a fixed constraint on the flat portions of those bosses, OK. They will not deform, OK.

You can do a pin constraint, so anything that's got a shaft or a hole or something like that running-- running through it, bolted connections. OK, you can do a pin constraint, and then you've also got frictionless constraints. Frictionless constraints allow side-to-side motion or translational motion without deforming the surface that you've selected. So if you-- if you run a simulation on this, and you've got a force applied, I'm going to use this, sort of, split surface here as my frictionless constraint, and then that surface that I've selected will not deform, right. It'll be. It'll maintain its original shape.

OK, after you've got your constraints, you've got your material selected, what we need to select is our forces. OK, Well there's a number of these, so I'll take this fairly quickly. For your standard forces just be aware that the size of the surface-- you can select a surface, you can select an edge, you can select a point. You typically want to use surfaces. If you pick a point to apply a force to, 100% of that load is going into one tiny little spot, and you're not going to get a good result from that. So when you apply a force, typically, you're going to do it on a face, and you would say, OK, I'm going to apply a certain magnitude of force to this face, and it'll split that force over that face, OK.

So that's-- this here is just the standard force applied. A bearing load, you wouldn't just do a force. So if you've got-- what's a good-- both of these are good examples on the bottom, I guess. But if you've got a load, this bar is maybe hanging on that loop that's being held, right, you don't want to just put a force, a downward force on that inside surface, you want to put a bearing load. So the bearing load is special in that it changes the magnitude of the force over the cylindrical surface in the direction of the forces application.

OK, I won't get into detail with some of the other ones. Gravity is applied to the whole body. Gravity's an acceleration, right, not a force. A body load is also an acceleration. It's the part itself that you're simulating being applied against itself through an acceleration value, OK. And then your moment loads would be like loads that are applied like a crank shaft or a turning force against an object, OK.

All right, pressure, it's basically applied-- you select a face only, and it applies the same magnitude of force to-- in a normal direction to the entire face. So if you think of air tanks and scuba gear, right, you've got an-- tires in a car, you've got a good picture of what a pressure is, right, and how it applies to a face. And then like I said with the body load and gravity loads, right, it's an acceleration. It's a fairly simple calculation to get to the force applied in this case based on the mass of your part. All right, here's where it gets a little bit different from the simulation environment.

Now we need to tell Inventor, OK, if we've got all of these forces applied, we've got my constraints added, I need to tell it what I'm going to-- what I don't want it to remove material from. So that's the preserve regions. You can color them however you like. I've got green here, and in this case here, I'm just saying, OK, don't move or remove any material from that area, because while it might be able to, I know I want a full surface for a proper connection, bolted connection or pins and so on.

OK, this doesn't have to be precise. It can be done through just like a massing out of an area. You've got a cylindrical selection or a box selection. If you've done any model simplification in the assembly environment, you've probably used something very similar, OK. All right, and then symmetry. If you want to apply symmetry, you can only apply symmetry one time or in once in a model, but you can apply symmetry along multiple planes. So there are options there. We want to, in this case, keep the symmetrical along the long axes or the long direction here. But it can be about the UCS, the center of mass, or the bounding box the-- kind of, the extents of the part design, right.

And usually it's actually-- I've really not had to reset that. It's been very good most times. The time when you do have to reset it, you can just create yourself a UCS feature and tell it to use one of the planes on the UCS feature. OK, so one of the-- your own work geometry in there.

All right, and then we move into our-- it's not our final step, but it's, kind of, the one that has the most impact on our shape, and that's the generator or the shape generator settings. So if you get into this area here, this is where you're going to set your mass target. So how much of a reduction in material or in mass do I want, OK? Keep in mind this includes your preserved regions.

So if you start preserving regions and you get too aggressive with your material reduction, then you're not going to make that target. And it'll tell you right away. As soon as you try and run a simulation or run a generative design, it'll come up with this error in your run dialog box and just basically say, hey, you've tried to remove too much material based on the forces you've applied, and you can't you can't get a result, OK.

And then you've got a couple of other options here. Minimum member size and your mesh resolution, they're tied together. If we talk about minimum member size, is what we're trying to do is say, OK, well, I don't want artifacts like this where they're not connecting to each other. And your minimum member size is to try and give it a minimum size that it will-- it will-- and I'm saying try, because it's not always successful because it is linked to your mesh. But so I want to try and close these off and have them be-- the smallest member can only be so large, and I specify that size.

Your mesh size has to be at least three-- or sorry, the minimum size has to be at least three times the mesh size, but the larger the better, OK. When you get into actually doing your solve, your mess-- the mess setting, you might be tempted to just say, well, I can just cut my measure in half and be done with it, and then I don't have any problems with my minimum member size. But then you could be sitting there for a day or overnight for a simulation to run. The shape generator is one CPU, just like Inventor, where it'll use all your cores, but you're not going to be able to share that burden across a network or any cloud infrastructure or anything like that.

So you want to just, kind of, keep that in mind that as you're building this up, the finer the mesh size, you could get very quickly into hours of generation. And there's going to be a point at which the trade stops, right. Do you have enough information from a simpler shape generation with a coarser mesh? That again depends on how you're going to take the mesh after, and what you're going to do with it, OK.

Here's two examples of minimum member size turned on and turned off here. There's not much difference in this one, but what you can see is here on the left there's a little bit of an artifact here hanging in space. And that was just simply one setting turned on and off, OK. And it was successful in this case over here making sure that they were all a set. You remember back on-- so that pipe manifold on my third-- third real slide? That one, it didn't matter whether I put minimum member size on or not, it just-- some of the-- it just still created some hanging elements there. So it's an attempt at creating a full and locked cross sectional size, but it's not always going to be 100% successful.

Right, then all that's left now is just to basically generate your mesh. It's a one click-- one click tool. If you've-- if you've got all your settings set up, if you already have a mesh, then you just have to actually go to that mesh. You'll see anything that's out of date with the little lightning bolts on here, and you just update your mesh that way, OK, and then-- and then run your shape generation.

All right, we're getting a little bit more modern here. This is early, early, early, early AutoCAD or CAD workstation. So let's take-- let's take this pedal here. And what I'll do is I'm going to take this right up to the point where we run this simulation, and then we probably won't have time to have it finish. So I've got another file with a finished simulation. We'll work with that afterwards. It's parts online, so I'm going to take my brake pedal, in this case-- and actually, before I do that let's, actually, take this gas pedal first. I don't have a simulation on the gas pedal, but what I wanted to do is just show you-- oh, pardon me-- show you just the difference here in the way this is going to work.

Now, I've got-- here's my shape generator button, so it's a one click entry. If you're working with sheet metal parts, it will also be on the 3D model tab, so not on your sheet metal tab. So you've just got to click one over, and you'll find your shape generator button in there. But you can run shape generation on sheet metal parts or standard parts. All right. This-- I want to point this out. Normally, we just click past this, and we say don't ever show me this again. But this is important right now, because what there-- our focus, kind of, like that point that we're right at the cusp of is being able to use this as a preliminary design or an actual production ready design.

So it still says preliminary, and I would-- I would still say that in a lot of cases, that's true. Although, if you validate your designs, you've created an output and you validate the output. There isn't any reason why you couldn't use that. Maybe cost becomes a bigger issue and production runs and things like that. But it is a perception-- pardon me-- preliminary design tool still at this point.

All right, when I'm in the shape generator, it will automatically create a case for you. So for those of you that do know FEA, right, you're not going to have to go and create a study and then-- and then build out. It already creates a study for you. Well, I really just wanted to separate that is that I've already created a couple of studies on this one. So when I entered my shape generator, it just brings me right to the last one I was working on. And I can go ahead and you can see I've got two studies here, but I'm going to go and create another one.

So maybe I've got two results, but I really want to lose those results. And I haven't-- maybe haven't saved them, but my force application, where I've put my loads and my constraints, I just want to try some different things. So I can do-- I can create a new study here, and then down at the bottom select shape generator just in the bottom here.

Once you select shape generator, there's nothing else to set. OK, so that information that you need to know for the stress analysis environment all goes away. You can see everything's great out here. Just give it a name or leave it at the default, and then I'm right back to that same setting that I was before on with the gas pedal.

OK, so let's go through real quick here and double check my material. So it is that printed aluminum material. OK, this is-- these are real properties taken from that website, so. And it's a fairly homogeneous material, so we should be able to have a fairly-- a trustworthy result from this given all of the inputs. And I go in through here and create some constraints. So there is a fixed constrain-- pardon me, a pin constraint there and there, because both of those hinge. And I'm probably going to get a couple of this-- a couple of these wrong. And I'm going to put frictionless constraints here and here where the bushings would go, OK.

If you under constrain your model, so it can move too much, you'll get an error. It'll be a fairly cryptic error. I forget the exact syntax, but the first thing to check when you get an error in your-- when you run your generation is to make sure that you fully constrained to your model, so that it doesn't have freedom to move after forces have been applied.

OK, I'm going to apply a standard force, and I'll select this top surface. So imagine someone in real life here-- there's a brake pedal. Someone is pushing at the top of the pedal, and if you, kind of-- pedal forces can vary a lot in what people are capable of. But I think a high number that's fairly safe, maybe a bit on the high side for most cases would be 300 pounds. OK, so if I say a 300 pound force here. OK, hit apply. It'll do the conversion for me. I'm happy with the direction actually. I could try it in another direction, but that's probably about the right direction. And we'll just click OK to apply that force.

I don't need any pressure on here. I could put in some bearing loads. For instance, if we took our-- now I've got a fixed constraint here or my pin constraint-- but if I took a load in here anyway, my-- given the calculation for the ratio of change, we're looking at about, I think, it was 2000 pounds-- I can't remember now-- right, that would be applied in the opposite direction at this point and your brake pedal. OK, assuming that we're fully-- we're fully at rest.

OK, I didn't get the direction, so let's go back here. Keep an eye on your model browser as well, because it'll have all of the-- all of the nodes for everything that you've added, preserved regions, all of your constraints, and so on. OK, and so if I select this here and just flip that guy around, OK. So there's my forces applied. OK, the rest of them in here, gravity's not big enough in this case to make any difference on my part. So I'm not going to apply it. The other ones here, remote forces and body loads are applied below.

A remote force, by the way, is one that's-- you can't actually select directly, because the force is applied a set distance away from your geometry. And what you can do is you can apply a distance value or an offset value, so you make a selection then put your remote force in a position outside of your model boundary. So that's what that one is there.

Now my preserve region tool, it works quite easy. In terms of your selections and how much you're able to manipulate on this, if I just select this back of the brake pedal, what it'll do it is it'll take the extents of this and, sort of, actually-- it just took that whole extrusion. It included it right away. So right shape, no, I don't-- no, I don't want to change that. In reality, this is most likely a pad that's-- it's going to be bolted on anyway. But for the application here, we'll just say that that's fine. And everything in that region will stay 100%. It's not going to remove any material from there.

And the last two areas here I'm going to preserve are this area. Now that's-- I don't want to preserve a square area here, but I'm going to pick my cylinder. And the cylinder just happens to be the exact same size as my selection. Whether it goes beyond the limits, not a big deal, because it's not going to-- it's empty space anyway. And then I'll select this one down here and just drag this out. OK, you can set specific values if you'd like. You can measure-- use your measure tools in your context menus if you would like to be precise. But I think most of the time just dragging it to a close approximation is good enough.

Our next step is our symmetries. So I'm going to come in here and select. Now I don't-- I definitely don't want it to be symmetric around that plane, so I'll select the next one over. And then just toggle on or off any other plane that might apply. So I just want this guy to be symmetric about that one. Your other selections are here. If you've got work-- a work geometry you've created your own UCS feature that you want symmetry about, all right, I can select that from the top, or I can use the automatic placement about center of mass or that bounding box. OK.

Coming towards the end here, let's reduce this by say 55%, and then maybe just in the interest of time, we're going to get an interesting-- probably not a usable result, I'll set my mesh size to two. OK, now if I did want to turn on this minimum member, it's-- zero's not going to work. But if I said, OK, 10 millimeters, OK, half an inch maybe less than that, what-- as soon as-- remember, I said these two are tied together? OK, so if I enter a member size that's too small given my mesh size, when I click, OK, it's going to give you a warning and a recommendation.

All right, so three times the mesh element size. So it's looking for 1.2 in my mesh size or smaller or really what's better would be to have your minimum size larger, OK. All right, so no I don't to proceed with my current value. So I'm just going to say no. Let's turn this guy back off and click OK. OK, so we're basically set to go, I just need to generate my mesh first. It's as simple as just clicking the mesh view button. Generate your mesh. There's the representation of that mesh on my screen, and I can go and say generate shape and click Run, and so away it goes.

OK, so I'll leave this here. You can ignore the warnings for the most part. What you really need to be wary of are any errors, right. Your generation will stop right away, and you're not going to get a result. So we've seen a couple-- we talked about a couple of those cases that might come up. But let's while that's going-- we'll see if it finishes in time. It's always neat to look at comparisons, so I put together this one. This is a-- this is the gas pedal-- gas pedal, clutch? I can't remember, which side this was. And if we took, sort of, the perimeter of that and just made it a sheet part, maybe we're going to create an assembly for the pedal instead, right, instead of just having a machine at a one solid lump or printed.

And what I've got here is-- it's a bit small on screen, so I apologize-- but I've got my minimum member size at half an inch with the mesh value of 1.5, and you can see the result. Now it only took a minute and a half, but my result is basically unusable, Right. I can't-- I can't use that for anything. I can't even model around that really, because I don't know what's going on in the upper area here or whether these are going to be OK the way they're showing.

So if you increment these downward a little bit, so we went down to 0.35 of an inch, changed my mesh value, it's getting better, right. And then better still at 0.3 of an inch, and I've got these further along here. And you can see our time value. So three minutes, four minutes. We're really not talking any major impact yet, but we're getting a much better result, OK.

So on the right hand side, we're at 0.17 of an inch, mesh value 0.6 six. It's a small part, right. It's not it's not a big part. We're talking something that's about this big. Last slide on this one. So here is where the times got pretty long. So 0.15 an inch for my member size, a half or 0.5 for my mesh element, my average mass element size, and I got a really nice result. Took me 11 minutes. OK, that's still not bad really. I don't mind waiting for that.

On the flip side of that, I went down to 0.1 of an inch and there was no sign of it finishing. My computer was not overheating, but it was blowing hot air and spinning in circles not doing anything. So there is a point at which you're going to just be like, there's no point in making this any finer. So that'll just come with experience, understanding the size of your part, right, and how fine the detail you need in it. All right.

All right, let's see if this finished, if we got a result. If not, then I'll pull up the result that I had. I've got set. OK, well there it is. So that's-- there's my generated shape. OK, the mesh is a bit too course. I didn't actually turn on the setting there to the minimum member size, so it didn't join these together. And this is going to be a difficult part to work with.

I could do two things with this now. When I actually finish, and I'll say promote shape, then I can say, OK, send this to my current part file or send it to an STL file, OK. So in this case really the part file is the best option, and I could go ahead and I could model around what I see. If you can imagine that these probably need to connect here, what's happening is if you look at when you do an FEA simulation, then you get a result, it's a colorized result, right. And you've got a gradient of color based on the internal stresses that have been applied to the part.

Effectively, that's kind of what we're seeing here is that there's a threshold, that safety factor, whichever that it's not-- when it crosses that threshold, it just removes the geometry after that. And so that's kind of what we're seeing going on here. OK, so this isn't usable. Well, so what can we do? We can do the one that we've seen-- if you've seen the demonstrations, so far, you know this is pretty much the same as what we've seen if you looked at the 2016 R2 release, and a lot of the posts that evolved-- or revolved around that. We can promote this.

And this is really the way, because this is a preliminary design tool-- this is really the way that this has been suggested for use is to say, hey promote this, use it as a guide. And then I'll come back to my-- with my parametric modeling tools, and I'd sketch in and around here and create my own cavities and extrude them. And for a flat part, that's really easy. And I think for the shape generator when you guys are doing flat parts, that's probably still the primary workflow. You're super close to production using it in part of your design workflow from day-to-day use, OK.

What we do have here is when we get into 3-D models, it gets a lot more interesting. Right, OK, now I've got to make splines, and I've got to do extrusion. Then I've got to loft new features, and it gets very difficult to manipulate those parts using a mesh promoted on top of your 3D model for anything that's not a flat part so you really are looking at printing OK well if you're looking at printing that's not going to change so much how you design this, because now we can do two things. We can take the mesh, and we can use them directly for printing, or we can-- we can take our parametric model, if we've got something that we like a little bit better and use that as well.

OK, if you are going to use this here, you can get creative too. You don't have to just use your revolves and sweeps and lofts. What's relatively new is the shape-- the free form tools, pardon me. OK, so you could create a free form shape and push and pull and use your t-spline technology, and then go back to your Inventor environment and combine your shapes and create a void that way, right, through the combined tool. So there's lots of options in there in terms of how you can do it. I don't think any of them is necessarily easy and not something you want to be doing a lot of detail in any way.

OK, what is new? So in 2017, what we've got is the ability to project geometry. So you can take that sketch, and you can project it if you've promoted it to your part file or if you just open up the STL file using any CAD, you can take those and you can project mesh edges, mess vertices, and mesh faces on occasion. I've had some trouble with mesh faces, but you can-- most of the time you can project those both. And you can use them directly for construction geometry or as sketch geometry.

So now you've gotten a lot more accurate, because I can say, hey, let's use-- let's use this geometry right in here, and actually use the shape generator result exactly, right, or very close to it. I can get right on top of that, OK.

The other new tool is, sort of, the mesh handling tools. There's a tool called fit mesh face. So if you go over to your surface panel, there's a fit mesh face tool that really, sort of-- it's like a painter. If you've used Photoshop, or if you used any image editing tools, use an eraser that's, sort of, like a paint brush, and you can select using this, sort of, a selection, there's a brush size here that is always relative to your screen. So where you zoom in or zoom out, the brush size is the same relative to your screen. But then you just start drawing or dragging that brush across your model, and it'll use the mesh vertices that you've promoted or that you're using this tool with to extract surface geometry from, OK.

It can be pushed too far to the point where you can't get a valid surface, but if you try and keep your shape simpler, don't try and create a fit mesh face that follows a full toroidal shape, or if it turns in on itself, it's going to fail. You're better off to do two surfaces and stitch them together at the end, right. And that would be your goal here would be to start creating surfaces, such that you can actually-- you can maybe thicken them or you can stitch them together and create a volume at the end, OK.

OK, model-- more modern yet. This-- did anyone use one of these, the old drawing tablets? Yeah, I was never lucky enough to have one of those. I tried, but at the time we never had it. All right, so I'm just going to switch back over here. So what do we have if we want to use these modeling tools? OK, well I'm just going to jump in here, and here is my, if you've done the promote so under your-- in the shape generator, you promote the shape to the model rather than creating the STL file, here's what you get, right. You get your model or your mesh, kind of, right inside your design here.

And so what I could do with this is I could say, OK, our standard method is create a sketch, and what we've seen typically demonstrated is that we would just start using a spline tool or align tools and draw rough approximations, right, not being too careful to get right on top of the mesh. What we can do now, though, is we can say, OK, well, let's project geometry. Now this gets a little tricky, because I've got my model on here already. So if I-- we have to wait for my select other to come up, but I can probably select-- there we go. There's my mesh edge.

OK, so you've got a selection now for vertices and edges. Edges are tricky to use, because they often intersect themselves. And when you try and create an extrusion out of them, you're not going to get a valid result. You going to get a self intersecting loop, and it's just not going to close. So what might be better is to select a few of these and use them as construction geometry. You can alternatively select the vertices, project the vertices, and connect the dots in a way, right, and just start at drawing line to line-- or pardon me, dot to dot.

OK, the other thing, even if you are using this process and it is a flat part, I mentioned we could use any CAD. Write it as an STL file. And solid works, that's why. There we go. There's my STL file promoted out of the shape generator to its own file, all right. And what I can do with this result-- OK, this was my last best result that we saw on the slide. The one that took 12 minutes. I can also just say, OK, well I'm going to create this into a part file. OK, now it-- any CAD with STL files is always still going to convert it to a file, then you'll have your mesh inside as a node the same as if you were promoting it to the other part.

So if we step back here, you can see there, right, there there's my mesh right in there. When you bring this into the-- you open it directly in Inventor, it is still going to be a mesh underneath a part file. So the first time you save this, you're creating an IPT, whether you do nothing to it or whether you start adding geometry. But I'm going to start a sketch here, and you can see I've also got sketch selections.

So right on top of my mesh, I can select this planer face. And then I could come in now, and it's easier to actually make selections, because I don't have any part geometry in the way. And I can start selecting my edges, right, or there's a point, right. And you can start making sketch points and edges, and you can draw and create a very accurate 2D sketch around your shape generator results.

All right, if we've got the other model here where it gets a little bit-- a little bit more interesting. OK, so what are we going to do with this one? OK, I can't-- this part here, if I was going to create a parametric model of this, what I could do is use my fit mesh face tool. Which you'll have to pardon, I'm not going to be to zoom it in either. So this tool on my surface panel, OK. So it's a little truncated, because the screen resolution's down a little bit. But it's just-- just look to the right of your extend command, and then in your fit mesh face, you can actually select your mesh faces, and, sort of, extract the geometry or the vertices right off that to create a surface.

So there's my, sort of, my cursor, and I can bring this up or down. And you can see it just, kind of, previews my surface geometry here. If I drag it down a little bit, and you've got a different type of face selection, you've got an auto fit, which it'll try and find out what the best fit is based on your shape, a planar face, cylindrical, sphere, almost like your primitives, right. On the right hand side is a spline. So if you know right away that you're not going to be any of these primitive shapes-- which I'm not, because I'm going to try and create a surface inside here-- then what you want to do is you want to select this directly.

There is a tolerance value that you can set. I usually bump that down just a little bit, but this is just going to help me with my surface generation. And then now you can either do single clicks or click and drag. So if I start clicking and dragging, right, you can see it selects all of these, and then if I let go, it'll calculate for a bit. The more it-- the more it-- the more you have selected, the more it will calculate, and the longer it'll take.

And like I said, you've got to be careful. It's not always better in my experience with this to select every mesh facet you can. You want to be selective, because you're looking for an average. And what you're doing is you're, sort of, extracting that geometry. You don't need to be directly on top of it for every vertex on every mesh face.

So what might be better to do with this one, if I cancel and just start the tool over again, would be-- so you saw that surface was, sort of, twisting when I maybe didn't want it to be like that, right, would be to select, sort of, sections that are important to me, right. And you get a much, much more refined surface. And you're still right on top of your shape result that you've got from the shape generation, OK.

It'll take a number of those combined together to get into anything that you can actually stitch, but if you use the planar tools, you can use any of the other surfacing tools to generate your own surfaces, you can still sketch, right, and create geometry. So you can combine these all together to create right from your mesh to create a brand new solid, if you wanted to. OK, that's all I've got for this part here. So if we jump back, those are-- those are pretty difficult, right, and they still require a lot of manual work. And what we're right on the edge of is being able to take this result and just say, OK, I've got my shape generated. I'm done with it. Let's make it.

OK, and if we do that, you've got to pay particular attention to your mesh, because now these minimum member sizes, they're critical, because now that's not a mesh I can print. But if you get a more refined result where you don't have any hanging edges, what I could do is I could send this STL file directly to the printer. And then not only that-- that's great, but then you have the problem with, OK, well, how do I document this design, right. I've got this mesh that I've got a part for, but I have no CAD design that I can use, right.

Well, in 2017 we can actually push this right into an assembly, and we come back to this. OK, so you can take this, use your AnyCAD tools, place the STL file directly in your assembly. It will still convert it to an IPT file, but then you can use your standard joint tools and constraint tools to assemble this, OK. And so like we saw with sketching where I've got my tools for sketching, I've got projections for edges, faces-- mesh edges, mesh faces, mesh vertices. I have those same selections when I'm using the joint and constraint tools. OK.

OK, this is maybe-- looking back, this is probably out of order. You can't in terms of documenting your design, OK, we've got a part, we've got it printed, we've got it in our assembly now, but how do I validate that design? That's, again, another troublesome thing, right? Inventor, if you go into the Inventor FEA or simulation environment, you can't import a mesh and simulate on an imported mesh. You can only generate new meshes.

So the in CAD tool there is a little bit little bit behind, but what you can do is you can import meshes in Sim Mechanical, so if you've got that. Fusion 360, so if you've got Fusion 360 what you can do is you can convert. There's a new module. It just came out Fusion 360 where you can convert meshes into solids. And then if you've ever tried, I don't know if any of you've tried the mesh enabler on the Autodesk exchange, it is, I think, a little bit overwhelmed by the meshes created by the shape generator, but it is another option that you can attempt.

So what it'll do is it'll try and create a solid model, a base solid in Inventor out of your mesh if you've got one, OK. If it fails, you get surface geometry. So all is not lost. You still have a fairly complete surface with the mesh enable. So there are some tools that you can, kind of, turn around and reverse and get back into and verify your design or validate it using FEA tools, OK.

Oh, I'm going backwards. Pardon me.

OK, so here's our-- here's our selections. So if you just mouse over, right, even though this is a mesh, I've got cylindrical joint selections. I've got plainer selections, so I can do flushes and mates and inserts, rotational joints, all of those standard tools you've got. OK, a couple of things you need to be aware of is that when you use the mesh, and you-- if you have to you're not going to get a mass value out of it directly in Inventor either.

You can't derive it, so if you're not going to be able to get a solid out of the mesh enabler, you have to override values like the total mass, right. So either measure it-- measure it after production, or if you've got a volume value, however you get that value, it's going to be an override that you manually push into Inventor, so that when you put this into this full assembly, you can see here's my sheet metal design, right. When you push this into your full assembly that you actually get an accurate value when you're finished, because while you can put that in there, it will convert it to a mesh-- pardon me, an IPT value for your part files. You're still going to have to calculate the math by another means.

OK, so this is the last, sort of, aside here. We're doing pretty well on time. So let's just take a look at this. Not this exact one, but if I open up my assembly here, so my brake pedal is missing, and I'll just place this in. Now you can place STL files, right, so the AnyCAD functionality comes into play, and you can take these, bring them into Inventor, and they work just like any other part now. Other than looking differently, you'd be-- you'd be hard pressed to actually in the assembly environment see much of a difference in behavior.

So I'll say, OK, let's create a joint. Pardon me. I don't have-- that's pretty loud. OK, so if I take my joint and just make my selection, you can see that as I-- as I mouse over, I'm getting those two standard selections. If I select the middle one here, this one's pretty easy. I can grab the center of that, and it's inverted. So we'll flip that around. I get that in the right direction, yes, OK. And I can click OK, and it's almost there. Now, let's do a constraint, and I'll show you. So if we mouse over the side with my flesh constraint, right, I can select edges and axes and faces, and I've got all those same selections here on my meshes.

So if we just wait for a select other to come up, you can see over here. So there's my mesh edges, right, vertices, right, faces, and so on. And you can select all of those using your standard tools in the assembly environment. All right, so let's take this last one or my final step. I'll grab that face and assemble it to this face over here, and now we've got my brake pedal right in the assembly, OK. And this file as an STL file could be sent directly to your 3D printer and printed.

This one here, so I printed half size. But if you're interested later on seeing-- so I did that with this design. Just went straight to the online service and had it-- to Shapeways and had it printed and sent right out. No, manipulation pass the shape generator. So it's just-- it's just almost there, right, so it's within our grasp. It won't be long before we're doing generated shapes. We saw on the first slide where we've got that, sort of, that mesh where we've got the rules where we're placing the center of gravity, and we're telling it where I need it to be stronger, where I need it to be flexible. We're almost there. I don't think it'll be too much longer in Inventor.

The last thing I'd like to point out here about meshes is that there is another tool that you could use called mesh mixer. It's been around for a little while now, and what you can do with mesh-- there's a whole lot you can do with it really. It is another 3D printing tool. But if these-- if these kind of facets, maybe you're frustrated with some jagged edges, or you saw on the other side I had an artifact. I don't know how far back I'm going to go. Not too far, I don't think. This one here. So I've got an issue right in here that I don't want in my print. Well, I could use mesh mixer to actually edit the mesh geometry directly.

So it's not an Inventor tool per se. It's not something you can add into Inventor, but it is an Autodesk tool that's available, and I encourage you to take a look at it. You can also connect directly to 3-D printing services also from it and say, OK, here's my mesh let's send it out. But it'll do smoothing, it'll create a solid, and help you generate geometry and look at your mesh in more detail there.

OK, here's another one that was created. I believe this was also created using the-- pardon me. I forget the name of it now. The other tool of--

AUDIENCE: [INAUDIBLE]

DAVID PONKA: Yeah. So yeah, so that's it. So it's-- this is, kind of, where we're going right to be able to create that stuff right off the bat. So my last slide, kind of, parting remarks. It's just we're right on that edge, another couple of versions. It's something that if you can get into and start playing around with, maybe creating some parts, looking at it, as 3D printing becomes a more cost effective production solution and as Inventors features around generative design are enhanced, as we know they will be every year, this will definitely be something that you can look into in the next year. Even right now, right, in terms of being able to say, hey, well, we've got this part over here that might be a perfect candidate for a generative design, OK.

And this was taken, so I've never met Pete Baxter, but this was an article that I saw posted just a couple of weeks ago, and it was, sort of, the future of the design world with emerging technologies. And I just captioned what he-- it said about generative design and the designers role in generative design and that we're going to spend, of course, less time doing the modeling and more time studying the rules and verifying the design afterwards. So something we think we can all look forward to.

So thanks guys. If there's any questions, let me know. We're a little bit early. Feel free to come up and take a look at these. I've got a steel print up here, some plastic stuff, and which-- and this generated result, so. All the best. It's still early in the conference, so we'll talk to you later.