DIEGO VALDES: My name is Diego Valdes. I'm a mechanical engineer with Greenpoint Technologies and today I'm going to give an introductory class on 3D printing with Inventor. You will essentially learn a couple of the differences between different technologies that are out there because 3D printing is a pretty big buzzword right now, and there's just different approaches to it and not all 3D printers or 3D-printing technologies are equal.

We're also going to look at what are good parts that are suitable for being 3D printed. And the really meat and potatoes of this lecture is going to be how to orient models in space inside of Inventor and then how to do this in Print Studio.

So Inventor 2016 comes bundled with a 3D printer environment, and within that you can do extra edits to the parts that you want to 3D print so you can prepare them. And then Print Studio is another piece of software that is separate from Inventor, but they're linked together through a button and that one will give you more functionality to 3D print.

How many of you own a 3D printer here? Show of hands. OK. Do you guys use the program that came with your printer, I take it, to print? You're used to exporting an STL file and then kind of tweaking it later? OK. So Print Studio does that for you and Inventor will let you do some edits beforehand so you can optimize it before you get to your STL file.

OK. So first, 3D printing or Additive manufacturing. I like the word 3D printing. It's kind of-- it's the buzz word. Additive manufacturing is really the correct term in that it's different from subtractive manufacturing in that you're essentially adding material, whereas in traditional manufacturing you start off with a block of whatever-- wood or aluminum-- and then you subtract it until you have your shape.

So first, a couple of definitions. SLS, SLA, FDM, and STL. And STA. So SLA is stereolithography. It's essentially-- you get a vat of photopolymer resin and you shine a laser onto the top layer and then you start moving this piston down. You start moving this piston down, moving your print bed-- you're essentially creating the part at the surface level.

You require supporting material for this type of technology because if you shine a laser up here in this part here, it will just fall down to the bottom. So it's a type of technology that requires support material, essentially, is the get go.

It is not very commercially available for home consumption. There's a couple of things. Autodesk has a printer called Ember that uses something similar to this, but it doesn't use a laser. It uses a digital light projector that projects a single image at the same time per layer. So essentially, instead of what this does, it will focus the laser and move it around until it cures everything around the surface, but the DLP just shines it and cures a single image at a time.

Next is SLS, which is kind of similar to stereolithography except in this case you start with a powder of plastic. Plastic can be any number of polymers, but it's usually-- nylon is the most common. So you essentially have your piece here, or your powder cake. With a laser, you cure the top or you actually fuse it together. That's why it's layer centering.

So you cure the top layer, or fuse it together, and then this piston moves down. A roller comes over and pushes some more powder on top, gets it nice and compact, and then you do it again layer by layer. The added advantage of this is that you don't need supporting material because the powder itself is your supporting material.

So if I were to center this part over here after I'm done, nothing happens. It'll just stay inside of the cake-- inside of the powder cake. So it's pretty convenient because you don't need supporting material. You can have a lot of internal features to them that are unsupported, that you don't need to do any cleanup of the supports afterwards. One of the caveats is that you need some escape holes for the material.

So if I were to center a ball then I would end up with a bunch of powder trapped inside that's uncentered that would need a way for that to escape. So usually you need two holes for the material to escape afterwards, and it's a very labor-intensive process. Depending on the size of your holes, you'd actually have to blow it out.

What's the name of the company? Shapeways. Has anybody printed anything with Shapeways here? So that's essentially what they do. They start off with a really big cake and they nest all their parts inside of them. So that's why they have really long lead times because they have to wait until an entire bed is optimally filled. They use algorithms to orient all the parts together and give the most optimum density. So one at a time, they build multiple parts for multiple customers.

Next up is FDM, which is what everybody here probably knows as 3D printing. You know, your maker bots, your printer bots, all that stuff, which essentially is you start with a filament of plastic-- with your material-- it can be any number of plastics like PLA or there are some trademark plastics that are flexible. But the gist of it is that you start with your build platform and with your extruder you extrude some of that filament over into a small diameter. And the heat of the nozzle kind of fuses that together with whatever's next.

So you go, again, layer by layer layering your material up until you finish your shape. Now, what happens when you have an unsupported section like down here? You will need to have supporting material, which means that as you print you're going to start printing supports along the side with it that you're later going to break off so that when you get to this part that would be unsupported, you essentially-- the extruder will bridge that gap, and there are some limits to what areas can be unsupported, what the maximum angle you can have. Usually 45 degrees is considered the-- it depends on your filament diameter, but 45 degrees is usually a good rule to keep overhangs.

Now, Print Studio will tell you actually what is going to need supports. It highlights it yellow for you and will add supports for you, but we'll get to that in a minute. And the next is PolyJet, which not a lot of people know about, but it works essentially like an inkjet printer in that it sprays photopolymer as well. And after it's done spraying it into a really thin layer, it will come down with a UV light and cure that over.

It's a lot slower, but the resolution is fantastic. It also requires supporting material because you're building essentially something out of nothing the same way you would do with FDM. Are there any questions from the [INAUDIBLE] point between the technologies? OK.

Now, STL. If you all have 3D-printed before, you know that STL is the file interchange format that we used to go between our 3D printers and our original geometry is more than a bunch of triangles. The name SDL, I think, is derived from stereolithography. I think 3D Systems came up with that name. And it's nothing but a surface model-- a surface representation-- and needs to be a fully enclosed watertight solid so that your 3D printer software can recognize it as a solid after the fact.

So, right technology. It depends on what you need to do, right? So good candidates for printed parts are composite. So typically, if you would need a part that's composite fiberglass and that requires tooling, if the tooling is too expensive and it's a cost-prohibitive part, you should consider making it out of a 3D-printed part.

If you have complex multi-surface parts, or multi-solid parts-- for instance, you need to make a pipe assembly and you have to have a gusset inside and you don't want to have to build three different parts just so you can have a pipe with a gusset-- you would be better just 3D-printing the whole thing with your internal features. Also, if you have a lot of curved surfaces-- because when you have an end mill, it would take a long time essentially going around that curve with a ball end mill as opposed to just 3D printing that.

The more complex the geometry is, the better candidate it is for 3D printing. Just stick with subtractive manufacturing. If you have flat machine pieces-- machine [INAUDIBLE] pieces-- simple extrusions. In the case of you're building a building, you don't want to build it out of 3D-printed material. You want to extrude a bunch of beams and then bolt them together. That's what makes sense.

And one of the bigger ones is access to traditional manufacturing. A lot of the times we 3D print parts because we don't have access to a mill or we don't have access and we just need one, right? Because we are hobbyists. If you're an industry and you want to make a bunch of them, then you might want to consider investing in the tooling, especially if you're going to do long runs-- a lot of high-volume production.

So, as a summary, kind of consider 3D printing-- additive manufacturing-- if you answer yes to any of these. Do you need tools to do your part? Then you should consider maybe doing 3D printing. Do you need several components to make up your assembly? 3D print it. Do you have any complex surfaces, internal features, anything that would require-- that you can't access with an end mill or you can't subtract material with? 3D print it.

Do you need a quick prototype? That's actually the number one reason why most people print. You just need a part now? Do you need to see it and feel it in your hands? 3D print it. And then is there a long lead time to the part? You don't want to get a quote from a manufacturer to make 50 of these if it's going to take two months to do a mold. Might as well just 3D print them. It'll take you longer to be on the phone negotiating quotes. Have any questions up until this point? OK. Let's move on.

So this is just a quick summary of the differences between laser centering, FDM, PolyJet, and SLA. Essentially, it breaks down into what the shelf life of the part needs to be and what the part size needs to be, more or less. Laser centering and FDM are really great for long-lasting parts. The plastic on the FDM parts is really durable. It doesn't have the tendency to warp over time, so good candidates for that.

If you just need to make a really high resolution but short shelf life, consider PolyJet or SLA. The reason why these are a longer shelf life is because they're polymers that are cured with light, essentially, so that degrades over time. But because the resolution is that much higher, they're good for building molds. A lot of the time my workflow requires me to make a quick prototype-- make a mold-- and then make a bunch of parts based off of that. But if my initial prototype is not good quality, then I'm wasting my time sanding and building it up to the quality. So this is good for essentially that. Parts that you need to use to make other parts.

Now, to the meat and potatoes. 3D printing with Inventor. Why do you need that? Right? You can just export STL right from the File menu. Well, Inventor comes with that new 3D print button there, which I'm going to show you in a second what it does.

By the way, all of this is also in the training materials. There's a PDF guide to all of this that you can download right now and read through if you think I'm going too fast or the video is going too fast. It's all detailed step by step in actually more detail than this because this is prerecorded. So if at any time you need me to stop, let me know.

OK. So this is a helmet I made earlier this year. I'm essentially going to go here to the-- I want to select the part that makes most sense for 3D printing. The helmet doesn't really fit in any of the printers that I know of or my friends have, so I'm going to select the smaller part there to print.

As you can see, it's a single solid. Right there. Solid 24. I'm going to open it. And you see the part in and of itself is not really ready for 3D printing. It's not really oriented in any way that would make sense just because this is the way I modeled it in space. It's kind of wonky from any angle you look at it.

So what I would have done-- and they're also too far apart. So what I would have done traditionally is I would have done some edits to my solid right here and then get them together in a way that I would have them ready for printing. They're still not really oriented properly, or at least they're close together and I know that they'll fit inside of my print bed. I know I can orient them later in whatever program I'm going to use for my 3D printing, but I have this problem, right? I'm affecting my design by editing the part.

So what I want to do originally is just not have to do that. So that's where the 3D print environment comes in. But first I'm going to show you what I would have done in Inventor 2015. So just go to the menu. Export. CAD format. And then save as STL. And go through the options here and you can set the units. Your resolution-- that just tells you how many triangles you have. And then you have more fine control over what the maximum edge length can be, what the aspect ratio-- so you're going to go ahead and click on Preview.

And you see the part looks pretty much the same as the original. Now, if I turn on STL Facet Edges, I can see the triangles, and I can see the resolution is what I'm expecting. A lot of the times the triangles will be too big and that will actually show up in the print. Even though you're like, ah, it doesn't matter, but they will show up.

So right from there you go to Options, and you can adjust the resolution right there. You see that I have like five times as many triangles on that edge. So that's what I want. I can close and click Save. Or I can access the options again from here if I want to, but I'm not going to so cancel that. And I'm going to undo all the edits I did. Go back to my original part, and now I'm going to instead go to the 3D print module. Bam. There it is.

So right off the bat, what you see-- I'm going to pause it real quick-- is a 3D printer here named Sirius 1. And I have a volume that actually matches the volume of my 3D printer. So right off the bat I can see if the part's going to fit or not. This is a fairly small part, so it's kind of unfair to have such a big printer for parts. But yeah, you can just make a quick estimate. Yeah, these parts-- they would print like this, but if I have smaller ones I would want to get those two pieces closer together.

Now, there's a bunch of printers that come preloaded here, and I'm going to show you, but just-- I want you to know that this has a bunch of planes. This is nothing more than a part. It's a derived part. You can even see there's nominal design here. This is my original part. It's a derived part. And all the edits that I'm going to do on here are not going to translate back to my original model.

The planes here are used to orient the part. Now, you can see the printer bed plane is on by default. This plane shown here. And the others are off, but you can select them when you're orienting your part.

So first and foremost, I'm going to select the printer. If I click on the dropdown menu-- is this thing on? Yeah, it's on. OK. Yeah. That's the Sirius 1. Click on the dropdown menu. You see it comes pre-loaded with a bunch of printers, but not all of the printers that you might expect. That's just to optimize your workflow. But if you click on other printers, you can see that it has them sorted by manufacturer and it has a bunch of them. To make them appear in the main menu up there, all we have to do is designate it as you favorite. And you can also make it your default printer, so whenever you enter the environment it will be selected.

So if you have one printer, like most of us, you're just going to make that the default-- your favorite-- then you can actually go and unclick all the other ones because why will you ever need any of the others?

So you see I selected the idea builder and now it's here in the list? I get some options, or some extra settings, I'm going to show you in a second. So just to ensure that whatever is on this menu matches what your actual printer is, you can see there's some extra parameters here. Manufacturer, model name, what technology it uses. FDM. Now you know what FDM means. And the print volume and what the origin is.

Here's the Ember that I mentioned earlier that uses the DLP technology. As you can see, it's a lot smaller. Four by six by 13. We have it downstairs in the main expo hall. If you have a chance, go by the Spark booth. They have like three of them running and they've printed all sorts of sponges. So really high resolution.

OK. So I'm going to select the printer here again. Replicator two because that's the one I have, or I wish I had. And you can see it doesn't fit, right? So that's the first problem. I have to orient it.

There's a couple of-- here, essentially, the main things you can do-- you can set the orientation, set the position, partition it, and do direct edit. Direct edit is essentially the same direct edit that you find inside of Inventor, which allows you to move faces, move solids, split solids, that kind of thing. Rotate solids. Cell orientation is kind of like a 1, 2 click move to bed type of command. Set position allows you to select an edge of the part in a printer face or printer wall and they will offset it off of those. And I'm going to show you how that's done.

So I'm going to use direct edit real quick now. Since these two parts are a single solid, I'm going to have to select the faces to move them. See right there I'm selecting faces? I'm going to select all the faces here and just move them with the manipulator.

I can also size, scale-- why would you do that? You can see it adds the direct edit into my tree as if it was a regular part. So I can undo all of this at any time. So, again, same environment that you're used to. It's a part. Nothing more. Only thing is it's not being translated back to the original model.

So I'm going to go ahead and rotate the whole solid, in this case using the manipulator. But it's still not in the correct orientation. I wouldn't want to print it like this just because it's kind of top heavy, so I'm going to go ahead and do the top set orientation.

The bed is selected by default and it's asking for a face. So I'm going to select the back plane here. And you see it also kind of centered it for me, so it's doing some smart things in the background there. But my parts are still not quite flat the way I would want them to be. So I'm going to do a direct-- I'm going to click OK, do a direct edit, again select the faces because it's a single solid. I know it's a 10-degree rotation here, so I'm cheating, but in reality you can snap to two faces.

I'm done and it's still not in the correct location, so I'm going to move it. I'm going to use this. You can see it's going to go through my bed, right? It's not really smart about where the bed is. So I'm going to snap to that other face, and there you have it. Both of them are flat against the floor.

I click OK and I'm going to use the set orientation button. That's like the wall. Like the back wall. And you can select an edge or a face. I'm going to select that edge, and you see the part looks like it's pretty ready to print right now. It's lined up. My tool is going to make nice, straight edges there.

It looks like I'm ready to print. So from here, I'm going to go to the print options. You see this really familiar dialog box? It's nothing else than the STL options that we saw earlier. Same options.

You can, again, toggle the mesh display. I don't know why I turned it off there. And you can see the triangle. Same thing as before-- make sure that it's matching your expectation for resolution and then the Save STL button essentially is a Save as STL all in one go. And from here again, you can access the options once more if you're really nervous about the options being not right.

So at that point, that can be done. I exit my 3D print. You see my part is not altered, but I have a 3D print object in here. That's the one that's controlling the edits inside of the part. Double-click it. It takes me back to my part as with the edits.

There is another feature here that I haven't mentioned, but I'm going to go in a second, and that's partition. Partition does nothing other than split your model, add a hole, add a peg all in one go. And you can even explode the feature later on and see what it's made out of, but it's really useful if you're just in a hurry and don't want to set up a sketching plane and all that stuff. You can just click the face, drag the manipulator back to where you want to split it, and then click on the points here and select where you want your pegs to be.

So there it is. You can't see it really clearly here, but it made a protrusion and the hole at the same time. And they're linked together by the scale factor of 1.05 by default. You can see the whole kind of [? phantom ?] there. I can change the diameter or I can drag this ring to change it.

At this point, I'm not really doing anything parametric because I'm 3D printing, so who cares. So I can just do whatever I want. But if you want and you explode this partition, you can right-click explode, which I'm not actually showing in this video, it will show you the sketch and it will show the formula that links them together is essentially what I would have done if I had done this manually, but it does it all in one feature.

And you saw the sketch there briefly with the two diameters also. And they're linked by the [INAUDIBLE] function. Double-click to edit. You can just drag the points again. Move them. The computer's a bit wonky there with the display. And I'm going to show you a section just so you can see the hole and the peg all in the same go.

By the way, this whole thing is being recorded so you can later see this video. And you can also access the notes, which are, like I said, in more detail than this video.

You can move that plane over. And you can see the whole on the peg. Now, you probably don't want to print it in this configuration, right? Because you're going to have some weird internal voids in there. And in fact, you would only do this if the part doesn't fit inside of your 3D print volume, right? But because I made this with the split command, I actually have now two solids.

I'm just going to direct edit, select the solid, and drag them out of the way and get them ready to print, which I could print in that configuration if I really wanted to have a two-part assembly for no reason. I'm using the set position example here as well, selecting the wall in an offset. You can select up to two walls. xy.

AUDIENCE: [INAUDIBLE]. If you were doing that, do you need to separate them? Is the offset big enough that you could just print them in place without [? fusing? ?]

DIEGO VALDES: It depends, but there's no point in doing that, right? You might as well just keep a full part. Right? One of the things I mentioned earlier in the presentation is if you are doing a part with internal features, that's a really good candidate for 3D printing. So you don't want to split it in that case. But yes. You technically could. As you probably saw, downstairs the printer guys have an elephant that has articulating legs and whatever and it prints in that state. As is. So yes. Totally possible.

OK. At this point, I'm going to click on Print Studio and it should export it to Print Studio. But for some reason in my home use license, it didn't go that way, so I'm going to pause it really quick and bring it. Has anybody used Print Studio before or seen it? How do you like it?

AUDIENCE: [INAUDIBLE]. I like the features and support. If you need extra support--

DIEGO VALDES: Yeah.

AUDIENCE: In fact, actually I have a question for you. [INAUDIBLE]. [? Picking up ?] support. I haven't really dabbled [INAUDIBLE].

DIEGO VALDES: Sure. Yeah, we'll get to it. So, first and foremost, Print Studio is not exactly-- it's not Inventor. It's actually called Spark, or Spark powered at this point. I think at one point this is going to go away and it is going to become fully fledged Inventor or Autodesk integrated. Don't quote me. I don't know.

So the controls are different off the bat. So that's the first thing that threw me off. To rotate is right mouse click, zooming is still done with the middle mouse wheel, and panning is also a middle mouse wheel. You can see the same thing. We're presented with our print volume already and my print area. This is Ember. This is actually upside down because Ember prints upside down. You guys haven't seen it. It's really neat. It's like Terminator rising out of the vat of polymer.

So first and foremost, options. So you can here select your printer. If it's connected via USB, it will autodetect it. Or you can print later. So, essentially, pick a printer from the list and it will generate the G code for that printer right there.

I'm going to select the replicator too because that's the one I was using in the previous example. As you can see, it also comes with kind of locating features on the print bed, so I know this is my front of the printer. I'll know that's the way it's going to look. If I want my print to be facing me as I'm printing it, because I like to look at my enemy in the eye, I'll know which way to orient it.

You can select material. You can select-- it comes with some profiles. So best, standard, draft. That essentially tells it what the fill density needs to be and what the sparse and fill pattern needs to be. For those of you that are familiar, or not familiar with it, the infill is essentially material put inside of the print so it can act as a support material to make it into a solid. And since you don't need that infill material for your surface quality, then it can be sparse, right? It can be filled into like a honeycomb pattern or rectilinear like a crosshatch type of pattern. So to get started-- yeah.

AUDIENCE: As far as your [INAUDIBLE] is there a lot more?

DIEGO VALDES: Yes, there is a lot more. I don't know about temperatures and whatnot, but in the Word documentation or the PDF or the class notes, I go into more detail into what the settings are. Essentially, I took a couple of screen grabs of what the settings are, but they don't really delve into too much detail. Any other questions? All right.

So to get started, in this case I'm going to import my file, and it's going to expect an STL, or an OBJ file as well if you're a 3ds Max user. I'm going to select the vents together, which is the one I did. You can see it brings in more or less the same orientation and position as before. Right off the bat, I have options to move it. So I undid all my really careful, meticulous placement in one go right there. You can move it to bed in case I didn't line them up. See, move to bed. It brings them down.

If my solid for some reason might appear here, moving to bed will move it into the bed. And you can do an auto layout. If you have more than one part, it'll try to nest them. I'm not sure what's going on in the background. Sometimes it makes some wonky decisions as to how to orient the parts. You can also scale it, but I recommend not to scale parts unless you really don't care about your scale. Because I designed this to fit something else. I don't want to scale it.

You can also flatten it in any other orientation. It will give you these planes to select what orientation. In my case it was already in the best orientation, so when I clicked on flatten it didn't do anything except show me that.

You can see the layout here. The workflow for printing is pretty well laid out. You have import, layout, repair, supports, preview, and then export. So I'm going to move-- you don't have to go step by step. You can skip steps if you know your geometry is OK, but I'm going to show you real quick repair. No problems added because this is an Inventor file, of course, and it's going to be a watertight solid as it is.

You can go select regions, select polygons, select individual parts, and I can do all sorts of things like crop, deletee-- give it a second-- separate. This one is pretty useful because, as you remember, these are a single solid, right? So I can separate, and then my model browser down here-- you see, I have two separate parts that I can now manipulate separately, which Inventor is kind of finicky when it comes to separating parts. If I'm not slicing through the part, it won't separate them. So, in Inventor, if I wanted separate those two solids I couldn't have. I could have done a copy design, but I don't want to do that.

You also have the option to patch holes, plain [INAUDIBLE], plain cut, and balance. Now, balance shows you kind of the CG of the part. You can move this plane up and down to see how the CG changes as you slice up another part. And there's this nice suggest base button, which will essentially get your curved surface-- if you have something without any flats-- and it will make a flat. In this case, because the polygon was so small, it actually broke it. You'll see it.

So I'm going to say, yeah, this is going to be my base because it's curved in this direction. And I'm going to cut the base, so it essentially just cut a hole in it. So it turned it red, telling me that it's not a watertight mesh anymore, so I'm going to have to now do some repair work. So it'll make a little tiny flat-- not the most practical example to show here, but it's kind of showcasing what the functionality is. I'm going to go back, and see, it even tells me with an exclamation mark here there's something wrong with your model. You should repair it.

Go back to analyze and repair. And it's going to select show me the errors and auto repair. It looks kind of wonky, but it's still watertight and it'll still process and it will still print.

Now, I could have just patched it manually and just replaced the polygon back on, but I just went with auto repair. And here, I'm just playing with the auto layout buttons. You see it does something arbitrary in the background. I don't know why it decided that to line them up like that was the best idea, but I guess Print Studio knows best. So I'm going to trust it.

So I'm in the support section. I'm optimizing rotation. Again, I don't know why it's doing that, but if the part was oriented in a way that is not conducive to printing properly, it will actually find the part with the best kind of orientation. So it was already flat, so it didn't have to do anything. So if I absolutely need the part to be in this orientation for printing, because maybe I want my striations to be in that orientation, I can go ahead and move it. And I'm going to add supports. This is my favorite part

I'll select the part, add supports, and boom. There they are. It highlights in yellow the parts that would have been unsupported. In this case, the gradient or the-- I guess it's not supporting enough. So when I click optimize rotation, you see it automatically puts it back down in the best orientation for printing.

I'm going to move it back to the bed manually because why would I want to print it like that? As you can see, it actually doesn't go through the bed. It's a lot smarter than Inventor in that respect. So before, I could just go through my bed in the previous example and here it will clash. So clash detection is pretty handy in this case.

And now I'm going to move off to the preview section maybe. First I'm going to put them all the way I want them. Again, rotate. And you can scroll back and forth between your workflow. Like I said, it doesn't need to be in any particular order, but it's logically laid out so it makes sense. You saw I moved it out of the bed and said move to bed and it brought it back in. So it does what you would expect there. And there you go. Now I'm going to preview.

So there it is. It is essentially showing me what the tool paths are going to be. It's telling me the estimated volume, the time to completion with this printer's specs in mind, and I can go and preview slice by slice. You can see the path right there. You can see that it's going to give me a double wall on the outside. That was going to give me a 15% sparse infill. I can see what it's doing. The yellow line signifies the extruder separations, so that's when the extruder stops extruding and jumps off to the other side.

I'll admit this is not the best configuration to print these parts. I will probably print them individually because I'm going to have the extruder jumping between parts at every single layer. So I'm probably going to have some dangling filaments there. And then once I'm done, I just click export and save my G code. And save for later. Any questions? Yes.

AUDIENCE: Back to Inventor. When you had the solid and then print solid, right?

DIEGO VALDES: Yep.

AUDIENCE: Are those two linked?

DIEGO VALDES: Yep.

AUDIENCE: So that you [INAUDIBLE] design change and then--

DIEGO VALDES: Absolutely. And it would actually break your edits if you make a design change, of course, that will break the edits that you made before. So if you delete a face that you were using before to position it, then you're going to have-- yeah. But yes, it is linked. So that's the advantage of-- what I would have done before-- the workaround would have been do a Save As of my part and do the edits, put them all together ready for print, but in this case they're linked. So yes. Anyone else? What's up?

AUDIENCE: What if you [INAUDIBLE]

DIEGO VALDES: Listed in Print Studio? I don't know. I think they're going to add printers as they do partnerships with other companies. I don't know that there is a way to add a printer because I think the drivers need to be talking back to Print Studio, but it's an open platform so I think if it won't be added on its own, somebody will add it or probably you can add it if you have access to the printer's API or code. So yeah. Print Studio or Spark is an open platform, so anybody can go in and edit the code.

AUDIENCE: Different colored filaments or do you have multiple filaments? How would you tell it to print either one of those different colors, or is that in this particular software?

DIEGO VALDES: No. I don't think it's in this one.

AUDIENCE: [INAUDIBLE].

DIEGO VALDES: Yeah, I didn't see any options to change filaments. I know some printers have multiple extruders with that option, but I don't know that Spark handles it. Again, I'm not a Spark spokesperson. Anybody else?

Well, if that is all-- let me see if I-- so did you learn anything? OK. I know it was a little bit basic, a little bit on the starter/beginner track. Hopefully I'll delve into more details and maybe next year I'll have a 3D printer. If you guys happen to come again.

These are the standards you've probably seen a million times if you've attended, but give me feedback. There is a prize for best presentation out there, so if you give good feedback--

[LAUGHTER]

Exactly. And yeah. If you didn't take notes or whatever, everything is on AU. The manuals are actually linked as part of this class. If you look on your Autodesk app, you can download the PDF that I've been referring to all this time, that again has more detailed instructions than what I showed in the video. Thank you very much.

[APPLAUSE]