CURT CHAN: My name is Curt Chan. I'm one of the technical evangelists here at Autodesk. I spend a lot of time working with customers. Specifically, John is one of our customers that I'll introduce in a second. Background, I'm a mechanical engineer as well, so I spent a lot of time in the defense industry.

So I worked for Northrop Grumman, Lockheed Martin, and then I did some time as a college professor teaching CAD for about six years, and then went to go work for a small company I'm sure you guys heard of called SolidWorks, and then found my way over here to a wonderful company like Autodesk.

And it's great to see the growth in our direction, and as we get into this, I want to keep it at a very open dialogue, and really show you guys this is a introductory class. So I think the key here is to educate you guys on what we can do with Fusion as well as the growth of where we're going. Because we have some of our other product specialists here too. We have the product manager here, as well some people from our development team. And with that said, we'll dive in.

So as we go through introductions really quick, I just want to introduce Mr. Al Whatmough. He is the product manager for Autodesk CAM products, and he is a guy that when you guys have complaints, they go straight to him, so he knows all about them. But it's great, because it's that communication from what I hear in the field and what he hears, and we collaborate to see what we can do to improve the product and take that customer feedback. We are really key on forums, and I'll talk a lot more about that in a little bit.

Also, too, we have people like Rene Fonseca who leads our development team back there as well. So if you have any questions in regards to when it gets to like posts and other things on that other high level, that's the guy I send everything to as well. And if--

[INAUDIBLE]

Yeah, and it goes that way. And then last but not least, Mr. Carl Bass. It's great to have him here and hear a little bit about what we're doing with Fusion. So there he is.

But with that said, I want to introduce our guest speaker today, before we get into all of the nitty-gritty clicks and picks, is Mr. NYC CNC. Have you guys heard of this gentleman? His

name is John Saunders. John? Hello.

JOHN SAUNDERS: Hello.

CURT CHAN: Hello. And I thought, you can come and hear me talk, and I can give you all the fluff, and make it look all sexy, and take your money at the end of the day, but the goal, really, is to really hear about it from a user, and how a user is using our products today. I think that's the value added, and what John has done with our products is that he's actually not only put them to the test, he's given us praises as well as given us great constructive criticism that we sent to the development team to make this product better.

And I thought, instead of me going through and making everything look all beautiful, let's have John talk about how he actually uses Fusion 360 for lathes specifically. He is a Tormach owner, so he has a Tormach lathe. We have Tormach here as well, so if you have any questions on any of their products too, and you can visit them down in the exhibit hall. We have them here, too, that can talk a little more about their products later on after the demonstration. But I think this is key.

Also, too, John has-- this is not like where we're paying him to go do all this stuff. This is out of his own will to where he's tested all these different products, from SolidWorks to Inventor to Fusion. And Fusion really fits that build of the entrepreneur, the guy who has the small job shop, the guy who can't really afford a $10,000 CAM or CAD package, they look to Fusion and they look to where we're growing the product.

So definitely take a look at him on YouTube. He has over 80,000 subscribers. He creates numerous videos a week, one that-- I don't know if you guys have heard of-- called the "Wednesday Widget," which goes through different parts of, hey, I'm going to make this a little widget on my Tormach 1100 or on the lathe, and the reason why I wanted to show the part of the lathe is that he's done multiple videos on this. So it's really key you guys dig through and take a look at what he has to offer.

Before I hand it over to John, since this is an introductory class, I like to give themes to a lot of the things I teach about, and being a ex-teacher I look at what really relays the message. And three topics I like to look at is educate, collaborate, and CAM. These are the focuses I'm going to look at.

But my first question here is, knowing that Fusion is a dual-platform software, so it works on a

Windows and works on a Mac, who here uses a Mac primarily? OK, so we got a good about 35%. Everybody else on Windows primarily? Any Linux people? So one guy. Always one guy. Ain't coming to Linux anytime soon.

But it's nice to know that Fusion is dual-platform. And it's interesting, because when you start going to a lot of the education conferences or the colleges, you're starting to realize that this next generation moving in to have the ability to use-- number one, I don't know how they're affording MacBook Pros as a college student-- but they have these machines to run Fusion or hop back and forth, and we are the first CAD and keyword CAM platform to run that.

Speaking of that, let's just talk about our packaging and what we have to offer. So today, within Fusion, you're going to get 2 and 1/2 axis, you're going to get 3 axis, you're going to get, as well as turning, and, of course, 3+2. I'm going to be doing a session tomorrow going through deep dive of clicks and picks on 3 axis machine and beyond.

So looking at different types of tool packs for 3D strategies as well as 3+2 machining, and so forth. So definitely take a look at them. We'll have another guest speaker as well, Jeff Hooper. If you guys don't know, you might have seen some of his bike parts, but he'll be speaking about how he's used that as well for just a short bit.

So from here, I find this really keen for the guy-- who here uses Fusion today? So we got about a good 60% of the people, which is great. Now, this is very key, OK? Because this face is not tech support. Now I'm totally, totally kidding.

I'm here to help you guys out, and more than happy to give out my business cards. But sometimes I'm here doing presentations, or on main stage, or doing certain things and you can't get ahold of me, so what I did is I put together this support reference document.

We have all this data out there that's really key for customers, like you guys, so I said, well, let me create this aggregated document that has the best tips and tricks, the best links of specific areas you want to focus on to our sales teams, to our support team, to our post team. This is all key, you have one aggregated area to go to. So I have no problem sending you this document, and we also will have it posted on the forums, but it's just an aggregated area of all the data that already exists out there.

Another other key thing, too, is that these are a lot of websites that I always recommend, and then, too, the forums. I find this is really key. Now, who here goes to the forums? Number one.

So it's only now about 20% of the people, right?

So forums is really key, because it's not only us as experts, but it's our users that reply, as well as our CEO replies back to our users about, hey, what are the troubles, what's going on, as well as what can we do to improve the product? So that's why I kind of made this really important here specifically on the forums.

Now, since this is a introduction to lathe, I say, let's have some fun, and talk about the fundamentals, and understand, well, really what is a lathe? And for the people that can answer this correctly, or the person that can answer this correctly, we got $100 gift cards to Nordstrom's, OK? No, I was totally kidding, man. $5 to Starbucks, all right?

So if someone can you tell me what a lathe is-- a customer, please-- shoot away, raise your hand. Does anybody know what a lathe is here? Please, somebody. I know, like, these guys know what a lathe. Yes, go for it.

AUDIENCE: A mechanism to create a circular geometry on.

CURT CHAN: Yeah, totally. There you go, boss. Thank you, man. Thank you for participating.

So this is what Google tells me what a lathe is, and this is pretty straightforward, right? Has anybody here heard of Pier 9, one of our shops that we have? So the majority of the people, right? And we have for manual to automated to all different types of machinery there, and it's really key just to bring it back and take a step back to, really, what is a lathe, but then we take it another step forward, right? What are the different types of lathes out there? So can someone-- we'll go with three people. Someone name one type of lathe out there for me.

AUDIENCE: [INAUDIBLE]

CURT CHAN: That's one right there. There you go, sir. Another one, please. Anybody?

AUDIENCE: [INAUDIBLE].

CURT CHAN: There you go, man. Thank you. One more.

AUDIENCE: [INAUDIBLE].

CURT CHAN: There you go. Thank you. Perfect So there is a big mixture of all these different types of lathe, and we want to focus on just the fundamentals, right? And I think this is key, because our goal

is to show you, how do you go from start to finish? And that's what Fusion is all about is, how quickly can I create and design a part, and then manufacture a part at the end of the day?

So just to give you a little more education, because John's going to dive into it, and the great thing is it since he's a Tormach owner of a lathe, he created all these great parts that we'll pass around and he'll talk about, but just from the basics is-- I'm not a machinist by trade, I'm a engineer. And Al, working with Al, who is a machinist by trade, this is where the blend of design and fabrication comes in together, so we can understand where software meets-- where the rubber meets the road at the end of the day.

And when I started diving into a lot of what lathe can do, these are all really key parts, and this image that I love, I use as a resource. If you guys went to the high speed machining session that talked about the helical. Helical guys talked a little bit about different cutters and three fluid, four fluid, all that kind of great stuff, right?

I like to use-- there's a couple apps I love, and Sandvik Tooling-- I don't know if you guys have heard of them, but they have a lot of great apps where you can actually go through and understand, well, what type of insert do I want to use, and I can plug it on here, and what type of material, what I want to do with grooving, do I want to do a parting face, whatever it is, those are great things to help you go through.

So I just wanted to point this out. It's some great educational information out there too. So it's a lot of reading, but I love digging through, and it really educates you on, like, well, what is parting off a part? If you don't know about that kind of stuff, since this is an introductory class, this is great material to go through. You can download all their PDFs. I'm going through all the different types of inserts they use, all the different types of end mills, and so forth. So I really wanted to share this with you guys as well.

AUDIENCE: Sandvik has a great class too.

CURT CHAN: Yeah, Sandvik has a great class. Speaking of that, their class is actually free. Free. So they feed you and they teach you for free. So if you go to their website, it's Sandvik.Coromant.com, and they have it all over the US. So from, I think, their headquarters out in New Jersey to one in-- I'm based in Southern California-- there's one out in Cerritos, California. So that's another great resource for you guys to check out as well.

The next portion is collaborate. And this is very, very key, because, as users, we want to hear

what's going on, because we like to know how you guys are using our products, and that's how we got John here to talk a little bit about it. He's going to dive into this in a second. But we love to hear about all the things you do. So if you're on Instagram, or Facebook, or Twitter, hashtag us, tweet us, show us what you're doing. And then, too, we even would love to take it to another level to where we would love to have you on our customer page.

So if you're saying, you know what? I'm doing something really cool, would you guys want to come take a look at it? Look at how I'm using your products. We'd love to capture another story about that. And you can go to our YouTube channel to look at all the different customer videos that we've done on how people have used our products. Backhand Bikes is one, Jeff Hooper, that will be talking tomorrow in one of my classes too. You heard about John. But this is great. So we are all about social media. Keep that in mind if you guys dig into that too.

And then, lastly, is CAM, and we're going to dive right in. The goal of today is really one thing and it's focus is lathe. John is going to walk through one of the parts he created for one of his "Wednesday Widgets," some of the things that he's learned, lessons learned, as well things that made him more productive in the software, and I think this is definitely of value to you guys.

But lastly, I want to talk about what else is going on this week. Of course, the best is CAM classes, but too, after this later on today at 6:30, we are going to be doing a design slam. Anybody been to AU last year? Was anyone here at AU last year? Cool.

So if you guys were out in the partner pavilion or the exhibit hall, we did a design slam of people using Fusion 360. So we say, hey, design this, and they have 15 minutes to design something. So it's a really cool event that goes from 6:30 to 7:30. Of course, I think there's free booze, and food, and all that kind of good stuff. So definitely check it out if you have some time.

And then tomorrow, again, we have the deep dive into 3 axis machines. So I think that'll be of value to the guys that are interested in using CAM and 3 axis. Anybody using 3 axis machine in here? So a majority of the people here. Great, great. Perfect.

So with that said, I'm going to hand it over to the YouTube sensation, Mr. John Saunders. John, thank you again for being here. Go ahead and dive right in.

JOHN SAUNDERS: Thank you, Curtis. So my approach to all this, I think is a little different, because, originally, I

didn't care about what it took to get the product made, I just wanted things made. So I had fallen in love with the machining, and I love machining, but for me it was about the art of making and less-- I'm not an engineer by trade.

The name of our YouTube channel, NYC CNC, comes from the fact that I was trying to bring a product to market while I was actually working a day job on Wall Street, living in Manhattan. And I had spent a lot of money outsourcing this idea to a engineer and then to machine shops, and I wanted to bring it in-house. But I literally didn't know what an end mill was, a Bridgeport.

And so I ended up buying this benchtop Taig machine, put it next to the twin bed in my one-bedroom apartment, and just, I mean, I fell in love with it. I convinced my wife to move out of Manhattan to the suburbs, specifically to buy a Tormach, and best move I ever made. We just moved back to Ohio, where I grew up, and we're growing. I love it.

But again, I care less about the features, I care more about how they help folks make parts. So I got excited about Fusion, because I was using SolidWorks, because we are now a big enough company where we could afford to write the check to buy SolidWorks. The viewers on our channel can't do that, they don't want to do that, so that's what I think is phenomenal about this product.

From a lathe standpoint, the first thing I wanted to start off with, this is the sort of part or widget we'll make. But if we take a look-- let's see here-- if we go back and edit this sketch-- I have, like, major gaps and holes in how I function as a engineer and machinist, because I'm self-taught. I don't really care, because I end up figuring out how to get things done most of the time. That's why I love YouTube. That's why I love the world we live in.

Like, a year ago, I would have made this part with multiple planes, and offsets, and lofts, or whatever, and then I finally realized, wait a minute here, there's, like, the Revolve feature. This is so much better. So you take a relatively simple 2D sketch. It's maybe a little bit harder to visualize if you don't think like that, but that's going to come out and make a part like this. So all we have to do is do a Create, Revolve that you can see right there, and since we've already done it, we can stop the sketch, and that gives us our part.

Everybody good with that? You guys get that? Obviously very helpful too, because when you want to go to adjust things, it's a lot easier to do that, it's a much cleaner CAD standpoint.

We have had a Tormach lathe for about a year. I didn't use it a ton, because I, honestly, am a

mill guy. I just have always been a mill guy. We brought our first employee on board about a month ago, full-time, and he has been on the mill, so I kind of lost my mill, and so we've ended up spending a bunch of time on the lathe. So we're going to walk through how we're going to make this part.

We swap into CAM and we've got everything already set-up. Let's see here, Command-G. We're just going to go ahead and duplicate this setup. Perfect. We'll delete everything.

Who here has used Fusion 360 CAM and actually posted out to a machine? OK, I'm going to go quick, because I think it sounds like most of you understand a lot of this.

In the setup, the few things that matter are, obviously, we've got our z-axis pointed forward. One of the things I don't love in Fusion 360 is you do need to-- it usually defaults to milling. As a person, I kind of prefer that, but you got to make sure to swap over to Turn or Mill-Turn. And then stock, we've got a relative size cylinder here, 40 thou, over the widest part of the part.

CURT CHAN: John, do you mind if I add something to that real quick? I think that's great that you talked about that piece, because those of you that had picked milling, you can still walk through the entire workflow. Until you get to the simulation side, and you go click on it, then the model disappears, you're, like, well, my model disappeared, but why isn't anything simulating?

So the key is, if you have it as default, it's when you get to the very end of visual aspect of doing the entire simulation will not show you anything. So it's very key when you do turning, you have to pick turning specifically, OK? Any questions on that? Good to hear.

JOHN SAUNDERS: I was giving Chris some credit, because one of the things I found that Fusion 360 does really well, that a lot of software packages don't, are actually meaningful-- well, here I hover over and I'm not getting any, but--

CURT CHAN: [INAUDIBLE].

JOHN SAUNDERS: What's that?

CURT CHAN: [INAUDIBLE]

JOHN SAUNDERS: Again, for me, I want to feel like a guy who watches our channel or who is in his own basement can do this and doesn't need to be an expert. They can sort of work through this stuff, and to hover over menus-- or whatever those are actually called-- are actually really, really helpful

and I think pretty well done in the Fusion 360 product. We'll see if we can find a better example, because these are not really what I mean.

So the first thing you're going to do, hopefully, is face off the part. It's obviously a lot easier on a lathe if you can create your sort of Z zero by having a little bit of extra stock. So you click Turning Face, and honestly, nowadays, I basically just click OK. You need to make sure you have your tool selected. You'll notice here-- yeah, so this second blue bar would be the sample turning tools. The first one is Lathe Southwest Flight to Las Vegas, which was me practicing for this class.

With the tool type filters turned on, it limited it down to a CNMT, but if you uncheck those, you can see-- and I actually find this is really helpful, because I know the first tools there, 1, 2, 3, 6, 9 13, were all tools I used in another part, and I do this a ton. I have a little library of mill and lathe parts that I have kept that I know are my go-to parts. I still really struggle in the milling 2D adaptive with open-faced pockets.

It's not intuitive to me the way you select the work piece geometry versus the stock geometry, whether or not you have machine pockets selected. So I just keep a file there that I can go to in 20 seconds and I don't stress about it. So I guess my point is, I'm not even an expert. I make my YouTube videos sometimes so I can go re-watch them and forget what I did.

So in this instance, we're going to use a CNMT, which is tool number 2 here. We're-- sorry, it's-- there we go. That one. There we go. Yeah.

Comes down, faces across it, pretty simple. I am a big believer in simulation. I wish you could leave Stock checked as a default. I also find that wall paint is a much better material to look at when you're trying to simulate. I don't need that glassy whatever it comes with, I don't even know what the default is, but--

I was talking to a bunch of high school students the other day in central Ohio, and I was sort of emphasizing, look, you guys got to realize you're going to be working with expensive raw material on expensive machines, and you have to know that the code is good, and the one thing I will say is, I don't think I've ever once found any sort of a material difference between the simulation and what actually happened, which is, I think, pretty cool, yeah.

So after that, we're going to go Turning, Profile. I'm going to use the same tool, which Fusion 360 picks up from the last op, and same thing. A lot of times what I'll do is just click OK and

see what we get.

So anybody notice things you don't like? Obviously, all the stuff on the front side is no good. It's dipping down into the groove, we'll fix that, and then we're going to go ahead and run it off the backside a little. I think, as a general rule, I'd rather put a parting tool into a turn face rather than a stock face, especially if it's something like hot rolled.

So let's go into-- oh, good example. We didn't activate the new setup. So we click this little radio button here and that activates-- you guys know what I'm talking about-- the new job. We can drag-- I think we can drag this into here. Yeah, reorder that. Yeah, there we go.

OK, so then all we've got to do is rest machine. I think we'll get rid of the front face. So you guys see all the extra steps went away, because it knows we already faced it off, which is great. To fix the grooving is a Allow Grooving somewhere.

AUDIENCE: It's under--

AUDIENCE: [INAUDIBLE].

JOHN SAUNDERS: Where is it? Just help me out. Oh, right there. Yeah, great. So that gets rid of the dive in. One of the things that I've noticed in a couple of other operations is, I think it comes up in the boring, is that Fusion 360 will treat the ID hole as a possible grooving op, which, to me, I don't agree with. That doesn't make sense.

AUDIENCE: [INAUDIBLE].

JOHN SAUNDERS: What's that?

AUDIENCE: There are some changes that were made.

JOHN SAUNDERS: Oh, OK. But see, that's the tough-- I don't always know what to think is when I'm wrong or whether it's just what's the methodology behind how this should work. And then we can run it off the backside pretty easily with confinement. Again, a misnomer, but backside apart will go 0.2 inches, and you can see we'll get the code. We'll get the lines running off the backside.

Next thing we'll do is run a groove into this thread relief pocket right here. Turning Groove, it's actually pretty cool, it finds the Grooving tool. We're not going to get into it at this class at all, but we actually posted a video on Monday on intro to CNC lathe tooling.

Tooling absolutely matters, and unlike mill, you know, by and large, in a mill, you can get by with the nominal diameter of the tool and you get code that posts, but lathe it matters a lot more. For instance, I treat-- does anybody know what a-- what's it called? A contact point on a lathe tool? That the right term? Does anybody not know what a contact point is on a lathe tool?

AUDIENCE: [INAUDIBLE].

JOHN SAUNDERS: So lathe tools have radii to them, generally, and the point that's driving the G-code path of the little blue line-- this actually came up-- I only learned this a week ago, because-- or a month ago, because I-- let's see what we can find it the profile.

AUDIENCE: [INAUDIBLE].

JOHN SAUNDERS: Yeah, so it's what Al said. I saw this blue line going through my solid model, and I was like, oh my god, we're going to crash, I can't run this code, and I can't really reach out to you or somebody, and somebody sort of walked me, talked me down, and made me realize that no, no, no, it's because it's the theoretical point of a tool that doesn't exist in the real world, but it's what drives the tool path. That make sense?

AUDIENCE: It has to do with how do you pick up the tool. You don't actually pick up the radius of the tool, you pick up--

JOHN SAUNDERS: Right.

AUDIENCE: --X and then Z, so the point you pick up on the machine is virtually everywhere there's [INAUDIBLE].

JOHN SAUNDERS: And so the video that we just posted was walking through how it's actually a lot easier than I think you may think it is to go through and create a CNMT empty tool. Shoot.

AUDIENCE: Yeah, I'm a total beginner, but I'm just curious, would you do mockups to make sure the code is--

JOHN SAUNDERS: A mock up of?

AUDIENCE: Just if you're going to-- before you go for the real thing.

JOHN SAUNDERS: I will-- on a mill, I never do anymore. When I turn a part, hmm, you know, I did run-- I cut air.

I'll just put the Z 2 inches out and run it dry, but honestly, now, I would be fine. I would go-- yeah, I use-- go ahead.

AUDIENCE: I'm just thinking about your design process, if you're trying to enter it at all, does that enter into your workflow?

JOHN SAUNDERS: In terms of what? In terms of--

AUDIENCE: Just getting the physical object made and reacting to that, and then going back to the model.

JOHN SAUNDERS: I don't-- yeah, it's a good question. I don't actually create a ton of models from scratch, and I actually, I'm glad you said this, because I wanted to mention, we do run a job shop. I run CAM on, with rare exception, everything we quote. I tend to fly.

I just go really fast when I do all this stuff. And that's one of the reasons why I love this software, especially when you pull in templates, is a customer can email me a solid model, I can have tool paths that I'm comfortable quoting on in, I'm going to say, 90 seconds. So I can get them back a price in five minutes.

Because for me, I just want it off my plate. If the job comes in, great, and if it doesn't, it doesn't. I'm not a big job shop. I'm not worried about booking efficiency of machine times over the next three weeks. So I put zero stress into the decision-making process that gets me an idea of the cycle time, and then I can go in and refine those if I'm actually going to make chips.

Oh, so anyway, my point was that with lathe, other than unlike mill, where it's sort of like the nominal diameter of the tool, with lathe, the tooling matters a lot. We're not going to get into that in this class, but for a grooving tool-- go back in here and it picked up the grooving tool, which I think is pretty cool. Let's not click anything and see-- it's actually interesting. You get not a terrible tool path. We obviously only want to groove that area, so Confinement. I'll just click these two faces, click OK.

Great point where simulation really comes in, and actually, Al, maybe you can help me understand this. This is actually not a complete tool path right now. I have to turn on Rest Machining to finish the width. Is that me or is that--

AUDIENCE: [INAUDIBLE].

JOHN SAUNDERS: So if we take a look. Simulation, and leave Stock off, and run Play. You're going to see it

leaves Material on, definitely on the Z positive face, and actually, it looks like it's going to leave on the--

AUDIENCE: [INAUDIBLE].

JOHN SAUNDERS: Yeah, and so if you take a look, keep an eye on what that tool path looks like. If we go ahead and click Rest Machining, that tool path's going to expand.

AUDIENCE: Yes, there's actually some tolerance issues that we've identified that we're [INAUDIBLE].

JOHN SAUNDERS: Still, normally, you shouldn't have to do Rest Machining?

AUDIENCE: You should've got the same result both ways, but the tolerance issue [INAUDIBLE].

JOHN SAUNDERS: Got it. My point is, I don't really-- it's not that I don't care, I just want it to do what I want it to do, and it's-- with between simulating it, it shows--

AUDIENCE: So to that point, not to say we've got bad software, but first [? round ?] performing [? the forums ?] were not always perfect. The best thing you can do is talk to us, and it could be a problem with us, it could be something you're doing wrong, but there's no problem putting something [INAUDIBLE] we got wrong. We want to fix it. [INAUDIBLE].

JOHN SAUNDERS: No, that's what-- seriously, that's what gets me excited about Fusion 360 is the fact that-- it's funny, because the internet is full trolls. I haven't found a single person who doesn't appreciate what it is. Seriously, I was talking to-- I swung out to a shop in Vegas when I landed, and was talking to a guy who was switching over to Fusion 360, and I was, like, you know you, as a hobbyist, you can just use it for free. He's, like, you know, I'd rather pay. He's, like, it's so happy to have the quality, especially on the mill side, for the tool paths. So I get excited about it, because, well, yeah, I do.

CURT CHAN: Chris? Question?

AUDIENCE: Do you come down with the curving tool and turn, and then come back out, because [INAUDIBLE] get the cut and you don't need it.

JOHN SAUNDERS: Yeah, I think it's actually--

AUDIENCE: [INTERPOSING VOICES].

JOHN SAUNDERS: I think it's actually doing that. Let's take a look. Did I just-- so if we turn back on Stock. Oh, hold

on.

AUDIENCE: Another quick point. The way he's walking through it is the best way to use the software too, but in these settings you change initially before you go to tool path [INAUDIBLE] if you want to, [INAUDIBLE]. See what you get, make a change, see what you get, make a change. The tool path is calculated fast. So the way you're teaching is perfect, John, because you're walking through [INAUDIBLE].

JOHN SAUNDERS: So see that last finish pass, which is pretty conventional. I think the only thing I've seen different is sometimes it'll, with a grooving tool, come in from one side, go to the middle, wrap it out, and then come in from the other side and wrap.

AUDIENCE: Can you get rid of all of the roughing paths as you [INAUDIBLE].

JOHN SAUNDERS: For sure.

AUDIENCE: --groove that [INAUDIBLE].

JOHN SAUNDERS: Yeah, I bet we can do that right now.

AUDIENCE: [INAUDIBLE] bottom.

AUDIENCE: Yeah, bottom.

JOHN SAUNDERS: Oh, yeah. Boom, there you go. If we do-- so the only reason it did that, I think, is because of the max. We still have an overriding factor here of the-- where'd I see that?

AUDIENCE: [INAUDIBLE].

JOHN SAUNDERS: Is that-- there we go.

AUDIENCE: [INAUDIBLE].

JOHN SAUNDERS: Yeah, so if you did 0.5, which is something really high, it should get rid of that, like so. Cool.

The point of that groove was a thread relief groove. So next step, turning thread, it picks up a threading tool. Another great point-- can you guys see the bottom right, the black point? This is a controversial subject, because some people put the contact point or threading tool on the actual tip of the cutting tool. Some people like to put it on the leading physical face, because it's a lot easier to avoid crashing a threading tool if you have the max point, physical point, but

that's not what that says right there. What that says-- so I'm sorry. That is putting it on there. Pay attention to it.

Click OK. Geometry, Threading Faces, click the face. And Thread Pitch, let's say we were doing a 1 by-- sorry, that's Thread Depth. Thread Pitch, if it was a 32 pitch thread, 1 divided by 32. There are a lot of features here that we can get more detail with, but click OK, and boom, you get thread depth.

Would it ever make sense-- I don't know this. I don't have a lot of experience with CAM software threading-- but could you have a dropdown menu? Because what we'd have to do here is go pull up the thread depth that you're wanting--

AUDIENCE: Yes.

JOHN SAUNDERS: Yeah.

AUDIENCE: It makes sense.

[INTERPOSING VOICES].

JOHN SAUNDERS: Right, no, but it's super easy to look that up.

AUDIENCE: Sure. It's printed in the catalog the last hundred years.

AUDIENCE: You need to go further and see.

JOHN SAUNDERS: Yeah, so then what you can control, obviously what-- is it Chris? Yeah, so what he just noticed is great, which is that we're not actually going to cut far enough, and that can be Backside Offset, 0.1, and I literally just a lot of times, just visualize this to see what I get. And you're not going to-- actually, I think you do get simulation on this, which is crazy. Simulate, Wall Paint.

AUDIENCE: [INAUDIBLE].

JOHN SAUNDERS: Yeah, isn't that awesome?

AUDIENCE: Yeah.

JOHN SAUNDERS: So obviously you can adjust number of passes and specific things. I actually meant to ask you guys this. How would you adjust, if it was too tight and you needed to decrease it, you could obviously go change the solid model, how would you override the depth? Is it just-- yeah, I

guess it would just be inner radius selection of this, and then put a negative offset in?

AUDIENCE: [INAUDIBLE]

JOHN SAUNDERS: If you wanted to cut your threads deeper.

AUDIENCE: Offset in the machine.

JOHN SAUNDERS: What's that?

AUDIENCE: Offset the tool in the machine.

JOHN SAUNDERS: I don't do that.

AUDIENCE: That's the right place to do it.

JOHN SAUNDERS: Is it really?

AUDIENCE: Yeah.

JOHN SAUNDERS: OK, fair enough.

AUDIENCE: Yeah, the tools have an offset number and you can offset that down [INAUDIBLE].

JOHN SAUNDERS: Got it.

AUDIENCE: [INAUDIBLE].

JOHN SAUNDERS: OK.

AUDIENCE: [INAUDIBLE].

AUDIENCE: Because later down the road--

JOHN SAUNDERS: That's never-- that's never happened to me, by the way, I just was asking for a friend.

AUDIENCE: If you rerun that and you adjust it for some goofy number, and then six months down the road, you go to remake that and the tool's not exactly the same [INAUDIBLE] cuts, super heavy, and you're stuck with that part.

JOHN SAUNDERS: Right, but I feel like a lot of time--

AUDIENCE: As opposed to doing it all on the machine. [INAUDIBLE].

JOHN SAUNDERS: OK, so dealing with the ID of the tool, we're going to cheat here and just create a-- I think I may have already-- I deleted it but-- this threw me for second. You go back to the same drilling menu from mill for lathe, so click on Drilling. We actually need to measure. We have a 1 and 1/2 inch ID diameter.

AUDIENCE: Were you measuring?

JOHN SAUNDERS: I was.

AUDIENCE: Here's a trick.

JOHN SAUNDERS: Shoot.

AUDIENCE: Do your [INAUDIBLE] on the job? [INAUDIBLE]

AUDIENCE: [INAUDIBLE] per diameter.

JOHN SAUNDERS: Got it.

AUDIENCE: Yeah, just like how it's centered.

JOHN SAUNDERS: So let's just assume that we did something you would never do, which is put a 1.4 inch drill in there to drill out that hole. What I want to get to-- I think everybody here can drill-- what I want to get to is ID boring. By the way, I think everyone here probably knows it, but we are big into customizing our menus, and if you hover over and click that Add to Toolbar, it pulls it up to the top, which is super useful.

AUDIENCE: Could I ask you real quick here?

JOHN SAUNDERS: Yeah.

CURT CHAN: We've added this too, as well, so you can customize all your-- drag all those [INAUDIBLE] you want. And two, if you don't know where something is, see that little Search? Just search for it. That way now. Very efficient. And it changes for each setting. If you're in CAM, or render, or model, that'll change as well.

JOHN SAUNDERS: So Turning Profile didn't make sense to me at first. Boring bar, this is like big boy work now, because you don't have all this real estate on the outside of the part to play with. So we had this-- where is it? Turn it on. I don't love the filters. So this is our boring bar, and the tool holder

and the insert definitely matter here, and this is where having them set up correctly will help you a lot with, for instance, in this part, crashing the backside of the boring bar into the other side of the part here.

So we've got our-- here, let me start over on that. I was clicking around. The Profile, Tool. There we go.

So first thing you got to do, Turning Mode, Inside Profiling, great. Geometry, select that. Actually, that didn't-- hold on. Click OK. We already get tool paths. Awesome. They're not great tool paths though. So go back and edit it. We want to do a confinement, and I'm going to confine it just to that wall here. Click OK, see what we get. Perfect.

It's not cutting out here, but what it is doing that I don't like is it's going inside of this through hole, which we didn't actually drill out here yet. We're going to fix that a different way, which is that-- the other thing that you got to pay attention to is, what are your heights? And this is all opposite normal stuff, because again, you're going to be doing your rapid in and outs in the center of the part. So what I did, outside radius was-- and, Al, anyway, hop in, because I'm relatively new to this, but I, again, got it working pretty well.

I'm clicking the profile that I want to cut. Inner radius, I'm actually going to choose that wall there, which does sort of two things. One is it pulls it off the center line, which is going to help give me a little bit of cushion, again, on the backside of that tool. It's an arbitrary distance. Also, if I've actually drilled that through, I don't need to spend time boring where there's already material gone. And clearance, I'm actually going to have clearance be the inner radius with no offset. Click OK. Much better. And we do Simulation.

AUDIENCE: Now, you drilled that. Did you just do it with rest machines? [INAUDIBLE].

JOHN SAUNDERS: Yeah, we'll come back to that.

AUDIENCE: Ah, no, sorry. Turning rest machining doesn't take into account [INAUDIBLE].

JOHN SAUNDERS: In Fusion only? It definitely does in SolidWorks HSM, and I was going--

AUDIENCE: [INAUDIBLE].

JOHN SAUNDERS: It was actually the--

AUDIENCE: [INAUDIBLE] Drilling on the center line, I can say that's the only example [INAUDIBLE].

JOHN SAUNDERS: Well, we'll try. So the ones they originally did a week or two ago, it was really cool, because we had-- I can't remember if it was a 118 or a 135 degree drill, but obviously when that drill went down, it left the difference of material, and when I chose Rest Machining, the product, that it's smart enough to realize it's got to go peck out almost like an adaptive style, like, it was almost maintaining that constant chip load, and coming in and cleaning it up.

AUDIENCE: As I started to clarify, the drilling [INAUDIBLE] rest machining, but [INAUDIBLE].

JOHN SAUNDERS: Got it. Got it. Luckily, I don't have Live Tooling yet, so that's not a problem.

AUDIENCE: [INAUDIBLE].

JOHN SAUNDERS: Then parting, parting is really complicated, so everybody just-- I'll hang on for a minute here. Turning, Part, click OK, you're done. Yeah, so that's all I got. I think lathe is awesome. I went to the template class this morning, which is something that we've been excited to get more in use with.

But you can add-- most of these are going to be pretty standard operations, so you can pull them into a template, and pop in a solid model, and just apply that template. And lathe, the nature of lathing, sort of self-fixturing with a [INAUDIBLE], we can go from an idea to a part in, seriously, under 10 minutes. It's so easy, and that's what I love. Thank you.

CURT CHAN: Thank you, John. We have about 10 minutes left. We wanted to leave about 10 minutes, a 10-minute window, see if there's any questions you guys had with lathes or with Fusion in general. Anyone here a student? Yeah, so great. So I think one key thing to keep in mind is with Fusion-- have you tried, have you downloaded Fusion yet?

AUDIENCE: Yeah.

CURT CHAN: OK, perfect. So you know it's free for students as well as an enthusiasts, so great to hear.

So always keep that in mind. We're going to help. Forums, if you guys want that PDF document that I printed out, all that help information, more than happy to-- I'll probably post it through the-- I guess, the channel for this class on Autodesk University, the website, and you guys can get it from there as well. I think it would be definitely of value. I just put it together probably, like, a couple weeks ago. So some good stuff. Yes, sir.

AUDIENCE: Can you turn on the rest machining [INAUDIBLE]?

CURT CHAN: Yeah, will do.

JOHN SAUNDERS: OK, so the tool path now obviously is-- so first off, what you obviously could do is turn off Roughing Passes. With the boring bar, that can be a problem, because you could overwhelm it as it dives into the bottom there, but let's see if we turn that back on. OK, back to our old tool path, and previous operations. Double-check that. There's a couple places where the default, I think, sometimes can be not from the previous ops, but rather from--

AUDIENCE: [INAUDIBLE] from set up stock?

JOHN SAUNDERS: Yeah, right, so you obviously want to pull it from previous ops, click OK. Oh, it works. Awesome. This is what I was talking about. This is awesome. So you guys can see, again, going back to setting up your tools correctly, you don't just grab a drill and assume it's correct. Like, the actual lead angle of the drill there will matter, because it's going to drive the roughing-- the press machine.

AUDIENCE: How about driving up the faces to [INAUDIBLE].

JOHN SAUNDERS: Somebody.

AUDIENCE: [INAUDIBLE] on the list [INAUDIBLE] come in and back off and then go up to faces.

AUDIENCE: [INAUDIBLE]?

[LAUGHTER]

AUDIENCE: [INAUDIBLE] understand. [INAUDIBLE].

JOHN SAUNDERS: This is the part that we ran last week, and one of the things I'll mention is, it's funny, so this is a 875 part that just gets turned down a few thou on the OD, and we really wanted to hold-- the part called for plus or minus 2 and 1/2 thou, completely overengineered part, like, does not need that tolerance, but the customer wanted it.

And so we were running them, and it was holding a couple thou, but I know I could do better than that, and I knew the machine, the OD, like, the machine was fine, and it was amazing. We switched our order around, and we turned down the OD almost last, so it actually required an extra turret change, not a big deal, and just that additional 30 thou of wall thickness on the part changed the rigidity of this 1.7 inch part such that our taper went from, like, 6 thou along this

ID to, like, 8/10, which was awesome.

But again, it was funny, because I never would have thought that that little amount of extra material would have changed how the part actually cut.

CURT CHAN: Great, great. Just one last question. Who hasn't tried Fusion yet? Out of curiosity. So a handful of people. So all we need is just a credit card from you guys today.

But I think this is key. I do a lot of stuff on our YouTube channel, too, as well as for milling, and it's always great to hear this stuff from a customer, because you're going to hear some of the things and the challenges he goes through, and you're going to hear it firsthand today of Al telling-- we have a laundry list of stuff, but we're going to get it fixed, and I think that's the value added at the end of the day is, we listen to the voice of the customer. I know there's a handful of customers in this room that I do work with hand on hand, and I'm sure they can attest to that as well. But I think at the end of the day, we're here to listen.

Definitely give it a shot, for those three or four people that raised their hand, we're excited to help you guys and nurture you through this process.

But if there are any other questions, feel free to come up, talk to John, definitely take a look at him on his YouTube channel, look at us on our YouTube channel. We have plenty of CAM videos as well, and hopefully we will see you guys in the 3 axis and beyond class tomorrow from--

JOHN SAUNDERS: And the design slam.

CURT CHAN: And the design slam tonight as well. We'll be showing some Fusion. Not CAM, but Fusion. So all right? Thanks again, guys. Appreciate it.