

LARS If any of you guys are into social media, I got some stickers up here. We're running a little fun
CHRISTENSEN: thing, because the Star Wars movie is coming out. You probably maybe saw the storm troopers running around. I got some stickers here. After the session, you can come up and get one.

But there hashtag underneath. So if you put it Tweet or a Facebook out and you do #jedieengineer and then some funny kind of thing, right, then you will have the chance to win a \$700 watch before you're going home today. And the reason that it's so important I say this is because when I was down for lunch, I walked over and the only ones who have done this so far were the people who was in my class yesterday.

So there's only, like, a handful of people in this contest for a \$700 watch. So it might be something to consider. Of course, if you have a \$1,000 watch, maybe you don't care.

[LAUGHTER]

I don't know. All right, so just a quick agenda here of the roadmap of what we're going to do today, because that's always good to know where we're going. So just going to do a couple of minutes about who I am.

Like I said yesterday, in my presentation yesterday, I would much rather go around and do two minutes about you guys, because I'm selfish and I'm more interested in learning about what you guys are doing, what you're doing with software and things like that. So definitely, I know we're getting to the end of AU-- I can't wait to sleep in my own bed-- but I will be down in the manufacturing area, advanced manufacturing area down there, in the exhibit hall.

I would love to just, you know, have a two minute conversation with you guys, just love to connect with you-- social media, into all that. Definitely love to connect with you on LinkedIn, and Twitter, and all that stuff.

Then this session here is a beginner session. And I think that's important that I say that. And it's on Inventor HSM. And it's really-- I want to try to do more the why I than the how. .

So to me, how is mouse button clicks, right? And there's a ton of those. And you've probably seen a ton of these sessions through here in this. Hopefully, you all have had awesome presentations to be at. But if you are new to CAM and you have not really had a chance to

kind of like go through the steps, you can never, ever, ever really learn something on the how before you have done the why. Why the heck are we doing things?

So that's what I'm kinda going to try to do over the next hour. So it is going to be basic. Now of course, you guys who are more experienced, you always can sit here and laugh at me if I'm doing something wrong or something else like that. So this is kind of like what we're going to do. We're going to talk a little about the flow of toolpath creation inside of an Inventor HSM, kind of like how it kind of works. Then we're going go in and look at some of the different toolpath.

I always have a give-away. So even if you think that I'm boring to listen to or the material is just way too easy, there is a give-away. It's easy. So just know that.

And then in end, in the third period, we're going to talk about creating Setup and then of course questions. So I'm all for that.

Now through this presentation here, if there are any specific things you want to bring up, you know, don't sit on until the end. I might just ask you to wait and remember it. You can use your brain cells to remember it. Or maybe we answer it right there, right, for like, hey, go back or something.

All right, so my name is Lars Christensen. And there is my email address. That's always good to have somebody at Autodesk email address. I'm good at forwarding things to other people who have more brains than I do.

I got 20 plus years in design and manufacturing industry, going all the way back to long, long time ago, a mold [INAUDIBLE] trade, have run a machine shop, a shop floor, three shifts, and all that stuff. And when people ask me what I like to do, I like to help people who just hate struggling with their design a manufacturing software. Because I think that that's the worst, right?

Because one of the things that I believe is that your design and your manufacturing package is just a dang tool. Especially when it comes to machining, CAM is just another tool in your tool box to get that part made. Because getting that part made is what it's all about, OK? And the journey there should be as easy as possible.

So that's what I try to do, use a lot of YouTube videos and things like that. And if there's

anything that is missing, then it's on me to make sure that some of that content is created.

If you are thinking about the accent, I live in the United States. I've been over here for 16 years. And I was born and raised in Denmark. I made that in red just because something about Denmark might come up later. So if you want to do a little Google thing with a give-away or something, to read up on Denmark might be a good idea.

And then also I do have a blog called cadcamstuff.com And that has been around for like eight years now. I try to post something once a week, twice a week. But there's a lot of different content out there too-- looking for tips and tricks, some CAD ideas, and some different things, reviews on 3-D mouse and all that kind of stuff. So definitely [INAUDIBLE] have you to check that out.

All right, so by the way, I got to give credit-- though that he's not here, and he'll probably never hear this recording. My colleague Jay is the guy who made this drawing here. I just love when they do that, take like a 3-D model and bring it out of a 2-D drawing.

All right, so this is kind of like the four steps-- like I said, this was basic. This is kind of like the four steps to go through and make a part Inventor HSM, OK? So you start out with your CAD, right? And what we love Inventor HSM is it's working right inside of Inventor, right? So you're already inside of the environment.

And you can model your parts up in there. You can model your fixtures. That was one of the things I talked a little about yesterday. So, you know, fixture design was really for me personally, when I got involved with 3-D CAD, was where I found the benefits.

So I was working in the time in a machine shop. And we got CAD in. And I was a little bit like, all right, this 3-D looks pretty cool-- until I realized that you can really utilizing it for fixture building to hold all your parts for work holding.

Then of course, we're going into CAM. And that is where we apply our toolpath. So the way it really just works inside of Inventor is we get another ribbon bar up here. Have a little tab, and you will see that. We have a little feature tree, operations tree up here. So it's just lie inside Inventor. We're building up the different toolpaths as we're going through.

And the way that the CAM software works is that when you're applying your toolpath, you are applying the toolpath right to that 3-D model. And there are some advantages of that. It actually attaches itself to the face of the model.

So when you are creating a 3D model, every face has a face ID. This is happening inside of a vendor in the background. Every face has an ID. And what the CAM software does is it attaches itself to that ID. What is cool about that its a couple of things.

First of all, it's cool that you are not doing any translation going from your CAD to your CAM. You literally are applying those tool paths right to the same model as you have modeled up, if you are the designer or engineer, or if you've gotten it from somebody else. The other thing is that if anything changes on that model, if a pocket becomes bigger or a hole moves, well, since it's associated with that, it's going to come up and say, hey, I want to move with it.

And in my past life, I used to get a 3D model from a customer. And I would sit down in my CAM software. And I would program it and get everything set up. I would go out to the machine. You get all set up. And you're ready to hit the green button on the machine. And then suddenly, the phone rings. Anybody ever, ever, ever not had a design change?

I cannot ever remember working on a part where at some point, somebody is like, oh, we want to do this different, right? Now, if you are broken between your CAD and your CAM, and you're getting a phone call where they say we changed A, B, and C, as a programmer, as a CNC programmer, you don't trust what they're saying, right?

Because if they forgot about D, then there's going to be two guys pointing fingers at each other. And the engineers always win over the CNC guys for some reason, right, though the CNC guys are normally strong.

So one of the things that's really neat about this is that now where we are in the same environment, they make a change, everything updates, right? So that's how that works. When we re-plan our tool path inside of our CAM, we can then also go ahead and verify it.

Now, you might hear some people call this simulation. That's up there. I call it verify. And the reason I call it verify-- oh, sometimes maybe I'll just say simulation. But to me, it's verification. It's not simulation. Simulation is like it's true you can see what's happening here. But for a CNC programmer, for a machinist, you're verifying that things are not going to go crazy out on your machine, because when things go wrong out of the machine, it's a lot of noise. It costs a lot of money. Jobs are going to get lost. It's just terrible. So I call it verify.

So you have an opportunity to actually look inside in the software and see what's going to

happen out on the machine. And then the last thing we do is we post out the G code. And this is really all it's about. All the other stuff can be pretty. It can be nice looking. You can do all kinds of neat things. But this NC code is really all that matters, because this is what the machine gets, right? This is what's going to feed into the machine.

So that's the end of it. All the other stuff before it is just fancy tools. Empty slides means that I should stop talking to PowerPoint, because that's boring and go inside of Inventor instead. All right, so up in the exhibit hall, there is three motor vehicles over in the advanced manufacturing area, and a motorcycle. But there's three vehicles.

There's only one of those vehicles that is on my Christmas wish list. And that is the BAC Mono - the open one. It's street legal. You can literally drive that out of your garage-- drive it all the way down to the track, drive around, drive all the way back again. How cool is that?

This is the pedal assembly out of the BAC Mono. So if you go over there, and you look down, that's what you will see in there. And for me, there is only one part that is really worth concentrating on. And that's the throttle pedal, right? My wife would maybe think the brake pedal when she drives with me. But I would only concentrate on the throttle pedal.

Now, so the neat things about being inside of Inventor, like I said before, is that we can do a lot of things in here. You can build these assemblies up. And you can check everything in here. A lot of great things inside of Inventor-- really fancy. Now, when it comes to CAM, like I said, there is a CAM tab up here, right? So you can just switch over.

And one of the things that is a little bit unique about our software when it comes-- because there's also other software that is integrated inside of all the CAD packages. But one of things about our software is that it's the same if you are in assembly mode or if you are in part mode. There is actually some of the other CAM products out there that when you go into assembly, you are getting another interface.

And that is kind of defeated the purpose of what we're trying to do on the HSM side, because what we're trying to do is we're trying to make a CAM package that is easy to use. That's the main thing in there. If you want something that is really advanced and can do really crazy things, go and talk to the Delcam guys, Powermill, or something like that. That's what they're good at, making crazy, fancy things. We want to make easy software.

So it is actually so easy that write instead of Inventor, we can just do any part and go in here.

And we can open up this part by itself-- this throttle pedal here. There's a throttle pedal. And we can actually just go ahead right now. And we can start throwing our tool path on it right now, right? There's no really disconnect from going from the CAD to the CAM.

I like to actually, if I had the time-- it depends a bit on time-- but I actually like to put mine inside of an assembly. And like I said, again, it depends a bit about the time. But it is nice to have, in this case, you have a vise on the table. Is that really distorted? Or that looks pretty good? That looks pretty good. It's just when I'm standing at an angle. It will give you a headache.

So putting it inside of a vise, now you can reassemble inside of your CAD what is happening out of the machine. And if you're doing anything like custom fixture plates to hold your parts and things like that, I would take the extra little bit of time and model up inside of Inventor exactly as you have it out in the machine, because it can just save you so much time that you confidently can sit down in Inventor with Inventor HSM, program the part, and you know that's exactly what you have out of the machine.

So I used to do a lot of tombstone machining where you could fix the plates on four sides. And that was what I would do. I would draw it up exactly inside of the software. Now, there's a trust, because it's all about trust in the end. And when they came in and says, oh, we want to put a new fixture plate on that machine out there, I was like, fine, man. I'll just rip it off out there, because I know that I know where everything is in here.

So that's just maybe a little bit of advice to tie these two together. I did a presentation yesterday about better CAD for CAM where it was trying to-- there is some advantages by trying to connect CAD and CAM, not just creating a tool path in there. But you can actually do some tips and tricks. So if you didn't have a chance yesterday, it's recorded. And you can see it there.

All right, so what I just quickly want to show you guys here is the workflow to get through quickly steps like I outlined them before. So I've already created a set up on this part. I will get back to set up on a machine-- the second side on this part. But I just want to show the step through.

So what I have is I have defined here a stock around this part. So the way you would do this is if you are manufacturing this part-- this throttle pedal or pretty much any other any other model, is that you've got order some stock. You can't get away from that, right? You've got to

go and find something on the shelf out in the back.

But what you would pretty much do is you would go in and say, all right, I am going to leave it little bit-- 100,000ths on each side. So it needs to be this wide-- 100,000ths of this side. So you made it a little bit bigger. And then you cut it or you order it. And then you can put it in the vise.

So pretty much what I did when I put that part in was I was doing the same thing. I made it inside of Inventor just with 100,000ths distance from the jaws. So it's assembling itself.

I'm going to come back to the set up a little bit later and show the different options in there on what you can do to stock. But that was really just how I threw this one in just using the standard mating inside of a Inventor assembly to place at 100,000ths, 100,000ths, and that's it.

Now, the thing we maybe would do-- the first thing we would do when it comes to machining is we would do a facing operation. And this is where I'm starting to talk more about why instead of how. A facing operation is pretty much just clearing off the top. So think about have like a piece of raw material that had been banged around, shipped, somebody kicked it on the shop floor, whatever.

And now, you put it in the machine. You're holding it in the vice. And you just want to take a nice, clean cut over the top. So that's a facing operation to do that just simply just making a planar surface that many times would become your datum, or your zero, or something like that.

We do have the facing tool sitting right up here. And by the way, all the tools that I'm going to be using through this presentation is in the free package. So Autodesk is giving away for free the 2.5 access for both Inventor HSM-- what we are going to do here, and also if you are on SolidWorks, HSM Works.

And that comes with post. It's not a 30-day trial. It's literally just because that Autodesk when they entered into this manufacturing was like, a lot of the competitors have made way too much money on selling simple CAM-- Just give it away And it's awesome.

So I'm going to select this facing operation here. And one of the things about Inventor HSM that I really like is, like I said before, it's easy to use. So when you're looking at the menus over here, these five tabs up here are always the same. They're always in the same order. They

always do the same thing. The first thing you always are asked to do is pick a tool.

So this is one of the neat things about this. You can literally go through this presentation. You can go back home. And when if you haven't already installed the software, if you install it or play around with it, you will pretty quickly catch up again, because it's the same thing no matter what you're doing. No matter what kind of tool path you are using, you will always see five tabs. And they all do the same thing.

So the first thing we're going to do is we're going to select a tool. So I am going to select the tool here. And we get a tool menu showing up here. Let me just give you the quick little tour about what you can do here. So this is the tool window to pick your tools. So if we start down on the bottom, down here you can actually go in. And you can create a new mill tool.

So let's say you have a custom cutter that you want to use or something like that. That's where you can do that. I think everybody can read what the next one does. It will create your new mill holder. So you can actually model up both the tool and the mill holder. If you do pay our paid package, you also get access to create turning tools. So we also turning inside of the product. That's not part of the free version.

Then over here, we can edit and select the tools as we added them in here in the workspace. Now, Inventor HSM already comes with a bunch of different tools libraries and different things over here that you can play through. So I'm just going to go in here. And I'm going to select a tutorial library.

And you will see the first thing here is actually a 50 millimeter face end mill. Now, what I can do is I can actually go down. And I can click Edit. And then it opens up another kind of menu here. And the easiest thing is just going through the different tabs. But the second one, the Cutter tab, is probably one of my favorite ones. I don't know of any other CAM software out there that has such a nice interface when it comes to edit your tool.

So what you're seeing here is actually what a face mill looks like if you're not familiar with a face mill. I actually also put it in the PowerPoint here. So this the look of a face mill. So this here is the side view. So the idea behind a face mill is you want to just plan it off the plate. But you want to have a big cutter, because if you have a small cutter-- you can use a small cutter. But if you have a small cutter, it's going to take a lot of small steps. If you have a big cutter, then you can take bigger steps.

So that's the idea about one of these face mills or shell mills, as they are also called. Now, we like to use carbide in this industry. But if you were going to make a carbide end mill that was about this big in diameter, you probably would have a hard time lifting it, first of all, because it's heavy. And if you dropped it, it would be very expensive, right? If you dropped it, you'd be shot. It would be very, very expensive. So what you use, is you're using this little-- it's like a steel holder.

And then you put these carbide inserts on that are just screwed in. And they actually have different faces on them. So I found a real one here. This is kind of like looking from down at the bottom. This is probably how the stock sees the end mill coming towards them when it comes down.

So I don't know if you can see this, but they actually have different shapes. This one has 1, 2, 3, 4, 5, 6 sides. And I think this one only has 4. So you can buy these different inserts in different sizes and shapes, depending on what holder you get. You don't need to have a face mill. So if you were new starting up with a shop, this was not be the first tool I bought. I would just use a standard end mill and go maybe a little bit more back and forth.

These inserts-- they are carbide. Now, you see that they are yellow, or gold. That's because they got a coating on them that should make them last a little bit longer, run a little smoother, less different types of coatings in there. This looks like some kind of tin coating or something like that. Generally speaking, inserts last for about 20 minutes before you got to rotate them-- 20 minutes and cut. And then it seems like that they're pretty much dead. So that's what that tool looks like-- I'd just like one or two to show that.

Now, what is really, really nice about this interface here is, of course, you can now change things. So if you don't have a 15 millimeter-- just because you're buying Inventor HSM, oh, now we got to go out and buy a 50 millimeter? That would be silly, right? So of course, you can edit all these in here.

We're going to come back in a little bit in a second. So you always select your tool. So I'm going to hit Select here for this face mill. OK? Now, you see that there is four more tabs up here. But one of the things that is neat about being inside of a 3D environment, inside of a solid environment, is that the software knows that there's something there, right?

So what 3D really does is it makes what is called watertight models. And the computer knows that it's in there. That's the difference between a surface model. If you ever brought something

and it was all surfaces, because there's a little crack in the model somewhere. You can heal it sometimes.

But if you have a solid model like we have here, the software actually knows what is there. So I actually don't know if I just want to do a facing operation over this part. I don't even have to select any geometry or anything. I can just hit OK, because the software knows where everything is. And you will see that I get a facing operation that shows up with a little bit of a blue and green line here, going across the part.

Now, we can go in and verify or simulate in here to see what we actually got. And that's why I call it verification. And not, I think, simulation is kind of cute, because you will do that as you're creating a different tool path, you're constantly going in and simulate it to see if that's really what you want, because we have all kinds of different settings we can edit and stuff like that. And we'll get to that.

So if I go up here and hit Simulate, and I go down and hit Play, you will now see that we get a visual representation on. We get verification on that we are facing off the part. And we say, all right, that looks pretty good. We can also turn on stock displays. That's like a check box right over here. So now we actually get this little green box here. And now we can actually see a step over too.

So that would give you a little bit indication of how much material I have in there.

AUDIENCE: One thing I had a question about this-- your program zero is at the very top left-hand corner of the part. Is that common for you? Most often, I like working with parts with x marked with y. So I put my zero at the farmost corner [INAUDIBLE].

LARS That's good. We're going to get to setups. So we can talk a little bit about it more when we get
CHRISTENSEN: there. But I like the way you're thinking. Yeah, so we're going to talk about that to when we flip the part around it if I don't run out of time. Any other questions?

I have asked our development when we do verification with stock, I want to see chips come flying off. And they just shook their head at me.

AUDIENCE: And coolant [INAUDIBLE].

LARS What?

CHRISTENSEN:

AUDIENCE: And coolant?

LARS And coolant. I want the coolant. they just shook their head. They didn't even answer.

CHRISTENSEN:

AUDIENCE: And you could add some sound effects too.

LARS Do you also want to crash? Do you want to see sparks coming and stuff like that? Yeah, now,

CHRISTENSEN: we're talking.

AUDIENCE: Verification of the [INAUDIBLE].

LARS Verification. You want to know what really happens. Yeah, so that's really how easy it is to

CHRISTENSEN: create an operation inside of that. What did I do? I hit the facing operation. I selected a tool. And I hit OK. And this here-- we can actually go now. We can go up and post this out. If you got an hit Post Process, this is all the post that comes with even the free version.

So Autodesk-- this is something that some people have run into nightmares about posts in the past. I should select the [INAUDIBLE], because that's what we have down in the exhibit area down there. So we have a dedicated post team on our team. And that's all they do is post. So we support post. We want to make sure that we have posts for all the machines you guys are using.

And this is the code that really matters. Like I said before, my PowerPoint, right? All the other stuff is just a tool. This is what we got to get out to the machine. Now, this is a fairly short code just because we don't have a facing operation at this point. But if any of you are familiar with G code, you can see we're using tool number 1 in making a tool change. There's a speed in there. G54 is the pick up coolant on.

And many times when you're programming things, you don't have to program the entire part before you start machining. I would many times just do a cover of the first roughing operations, post it out, go out to the machine, get the machine running, right? Now, the owner comes out. The machine is running. You're sitting in front of your computer drinking coffee, eating a donut, programming on in the part. And he's like, oh, look at this guy, man. He's awesome, right?

So sometimes you just take small cuts. And it takes a little bit-- no, you shouldn't do that. But

no, but that's a thing to keep in mind though. You don't have to sit there and program everything. Get your roughing operations going, man. And then you can go in and play around with some of the other things along the way.

All right, from this point on, we are just going to keep on putting on some tool path on this part here. So let's go in and add a 2D adaptive tool path here. So I'm going to click on the 2D Adaptive. And again, I'm going to go and select a tool. This case here, I'm going to go down. And I'm going to select a 16 millimeter flat end mill. I'm just going to go into edit to show you the picture. So now you can see that the diagram in here looks a little bit different.

Now, I just wanted to talk a little bit about tooling, because this is a thing that I get asked quite a bit. There is so many different types of tools out there-- end mills-- so many different variations. And if you're new to machining, then you also when you're talking with CNC machines, then you also start talking about formulas to figuring out feeds and speeds and stuff.

And I remember personally when I got involved with CNC milling, I found that a little bit tricky, man. You throw these formulas at me at that point in my life. And I was a little bit like, I thought I was done in school, right? I can't get it in my little calculator on my iPhone.

So that becomes a little bit tricky. And this is a show stopper for many people. It's frustration. What do I do? How do I come by this?

I know a couple of people-- John, who is one of the guys who's here with NYC CNC, who has a YouTube channel. Him and I chatted yesterday. He said he had some kind of a rule that 300 divided by 2 or something else. But really, my thing tool vendors are your friends, OK?

So I think it's fine when you're going out and buying things that you shop for the cheapest, and try to get a good deal, and things like that. If you're new to CAM, just find it a tool vendor. And when you call him up to order some tools, say to him, dude, I'm kind of new in this.

And I need some hand-holding. And if you can't give me some hand-holding, then I'm going to call somebody else, OK? Because what they can do is they can give you some starting points for your feeds and speeds.

And one of the neat things about it is, and I have done this many, many times, I would get their feeds and speeds, and the tool would break. Well, then you'd just call them up, and be like, sorry, Joe, it broke. And they send you another one without charging you, right? If you're

standing there, and you're trying to suddenly pull numbers out of the air or something like that, then they're not going to refund you a \$30, \$40, end mill when you smash it into it.

One of the reasons that feeds and speeds are so tricky is because there are so many factors into it. Of course, what material are you cutting? But also, if you go down on the floor down in the education area, we have a VF2 making these small, cool cars. Then you can walk over in the other area where we have these small tabletop machines that runs. That's like running a semi versus a little go cart. And the tools are going to be run different in that.

How are you holding on to the part? What kind of tool holding are you using? So there's are many factors. So tool vendors are your friends. If you're starting out, just rely on them to feed the speeds. And you'll start picking it up. I always carry a little notebook. I'll always write down, oh, so for that half ins end mill, he said this and this. I would always go back through it.

There is so many out there. There's some rules by thumb. This is mine. Don't worry about high-speed cutting for standard cutters. We're only using carbide in this industry. If you're using standard end mills-- carbide. Don't go with high speed. That was ages ago.

Don't worry about coatings unless your tool vendor says, well, you have a little bit of special application. You're doing production. You're trying to kick out-- you're trying to take 10 seconds of a cycle time. And we can extend your tool life by 10% by adding a coating. Don't worry about it.

Fall through to standard-- OK. If I had to go out and buy right now. If you came to me and says, all right, let's start out. You're new to this. What would I go? I would say, just give me a 4 flute. And you can look at these 4 flutes here. See this one here? Can you see how these two are connected, but these two are not? This is the same thing over on this one.

So this is center cutting. They can actually cut straight down. But when I'm talking about 4 flute, it's depending on how many flutes are on here. And it's really about how big of a chip you can take. So if you're machining anything that is hardened that had been heat treated, then you maybe have more flutes.

But if you're cutting something like plastic and aluminum, you might go down to 3 or 2 flutes, because you can have a bigger chip between the flutes. But if I had to go out and buy-- just give me 4 flute. That's out of the gate. And this is the three different sizes I would order, depending on if you want inch or metric.

There's a billion different sizes in between. But this year, for most people's standard work, this here will get the job done. And if I didn't have money to buy a facing mill, I would just face mill with a half inch end mill and just over it a little bit more. OK? But really, my biggest thing I always tell people is talk to your tool vendors. They are more than happy to help you and get you going. And sometimes they also have free T-shirts and stuff like that.

AUDIENCE: It's a rule of thumb not to [INAUDIBLE].

LARS Is that a rule of thumb? You can, right?

CHRISTENSEN:

AUDIENCE: It depends on the cutter.

LARS If it's center cutting like this one, you can actually plunge down with it. Yes All right, so I'm

CHRISTENSEN: going to use this adaptive here. Now, when it comes to the adaptive tool paths, I'm going to select this 16 millimeter end mill. I'm going to go to the second tab. So the second set we have in here is all about selecting geometry.

So the first tab is the tool. The next thing is about selecting geometry. Now, for a facing operation, the software knew where the stock was. And we didn't have to select anything. But now where we're doing the 2D adaptive, we have to select something. So I'm just going to go down and select this bottom etch down here.

And when I select that bottom etch, you will see that we get a verification of the area that we're going to be machining. There's a little arrow right there. I get so excited, I talk fast. And I can flip the tool path back and forth with that. You can see how we're losing the blue shadow, depending on what side we want to be on. That's actually all we have to do for the adaptive. But we can click on that. And we now generate a tool path. It looks something like that.

Now, there is more settings for this tool path that we can mess with. But just to show you, we really don't have to do those two tabs for this tool path here. And if I go in and verify this, I'm just going to select the set up, hit Simulate. I can turn all those yellow and blue lines off if I want to.

And you will now see that, again, we have our facing operation that is already running out of the machine. And then you will hear, see, the adaptive tool running in here. Now, the adaptive tool is a little bit different. It's called of roughing tool path. I don't like that, because if you had

come to me when I used to be on the floor at [INAUDIBLE]. And you came and said, oh, we got this fantastic roughing tool path. I would have been like, I don't rough much. But adaptive clearing is actually a bit different than just a standard roughing tool path.

If we go over here and take a quick look, the way the software works is this is standard. A standard pocketing operation will just take this out of shape and see you offsetting it in. But what adaptive does is it keeps a constant chip load on your cutter. So the software looks at what the stock is, what the part is, and it calculates so it keeps a constant load on the cutter

That means that we can cut a lot faster. For some people it's great. Some people don't care about. But the other thing about it is, and this is what sells me on it, is it gives you a lot less tool wear, because the tool is not going. And suddenly, it hits a lot of material, and then little bit less, and then more and less.

And it's actually also good for your machine what any owner of course will love, because it keeps a constant load on that. So what you will see is this is slow motion. You can actually go full depth with the cutter. And you get these beautiful chips flying right off here.

And if you go to any machinist and you show them that. They think that's the best thing. If you are new to machining, just tell you-- just use the adaptive clearing. If you are an old timer who's been around for a while, I tell you, man, it's the best tool path that I have ever used.

Now, the next thing I'm going to do-- so the adaptive did leave a little bit of stock on our part here. I am afraid that I'm going too slow. So I'm going to go in to finish the outside. I'm going to use a 2D contour for that. Now, a 2D contour is pretty simple. What you do is, again, hit 2D contour. Of course, select the tool.

Sometimes, you will use the same tools to rough and finish. Sometimes you really can't just because the rough will wear a little bit more. I'm going to go one down in size here for 10 millimeter. And so that's easy. So you're going to select your tool. What I'm going to machine-- I'm going to select the same etch as I did before to machine this. And many times, what I do when I'm programming Inventor HSM,

I really don't waste a lot of time going through all the different tabs. It's sometimes a lot easier just to hit OK and see what you get. Just see what you get, and be like, all right, that is pretty cool. Now, if you ever want to change anything, it's super easy. So what I'm looking at here is this end mill is going to come in over here.

If I wanted to come in over here instead, there could be a typical example where I hit OK. And I'm like, oh, dang it, I wanted it over there. All you do to get back into the tool path is you just go over and you right click. And you hit Edit. And you're right back in where you were before.

Now, I can go over-- the next tab over here is the Heights tab. We are going to come back to that in a little bit. And then there is the Passes tab that has everything to do with the cutter engaged in the material. So the cutter is inside of steel. That's all the different settings in there. And you can see there's a lot of settings, right? Oh, my goodness. Look at all these check boxes, right?

So you have all the power you need to do all kinds of crazy things. But you have seen already you don't have to turn them all on to do symbols standing machining. The last one is the Linking tab, but has everything to do with the cutter not engaged in the material. So me entering the part is on this last tab, because it's not engaged in the material yet.

So I can actually go down here and hit the entry point down here in the bottom. And I can literally just go over and select right on that 3D model right there. And when I hit OK, you will see there are now just chains that lead in. That's how simple it is if you want to make changes in them. It doesn't have to be so difficult to do, OK?

Up here, we have a little bit of a shelf. See the little [INAUDIBLE] over there. The Contour tool path might be pretty good for that too. So I'm just going to go in. And I'm going to hit it again, Contour. I'm going to leave it at the same tool, 10 millimeters, that works out.

This time, I'm just going to go ahead and select this edge right there. And then I'm going to go over and hit OK-- see what I get. There's no really reason to go through all the steps and sit there and think about it too much. See what you get up on the screen, because remember, we have this really cool simulation of Verify where we can go in and see what the steps are as our tool goes through, right?

We got our facing. We got our roughing operation. And you can just see how the bar is running down here? You can actually just click anywhere on that one and use fast forwarding right over to that point. So there's it going around. And there we can see we're doing our Contour operation. Cool?

But now you see that we actually have a little bit of a sliver right there. This is why I call it verify. Every time you're applying a tool path, you're going in. You'd be like, oh, that was the

last tool pass I did. So I'm just going to go in here. And I want to right click and hit Edit. And where am I. Back right in where I'm setting all my settings, right?

So I need to do something with that little sliver. Well, that is when my cutter is in the material what will be as always default tab for passes. So I'm going to go over there. And then I can go down and say, you know what? I want some multiple step over finishes. So I'm just going to go and say, I practice. So I know four steps will be good. But in real life, you could, of course, just test it out, right? 100 step over. Click OK.

And now, you will see that I got those step overs there. That means that we now go in and simulate it. We now know, I know, if you trust me-- you don't have to. But now that sliver is gone.

That is really the basic steps of machining through here and see what is going on. Another thing that I really like to show that we can do inside the simulation is we can show transparent. Now, we are actually showing the part behind it. So now we see the model between the-- that's pretty cool. And we can compare the two together. That's kind of nice too.

Now, we can see everything in blue. It's not finished. Anything in gray are finished. That's neat, right? So now you don't have-- many times when you're machining things, you will use a big cutter to machine something like a pocket. And then you have to pick out the corners with a smaller cutter. And I can tell you how many times I've been standing out in a machine and be like, oh, I forgot that.

Well here you can see it right in your face, right? We're missing something. Now, the other thing, of course, we can do in here is we can drill some holes. So I'm going to go up here and hit the drill operation. And in here, we only have four tabs just to confuse the enemy. We don't need more.

Now, since this is steel, we will many times go in. And we would actually spot first, because a standard drill is 118 angle. The tip point of a standard jobber drill is 118 degrees. And what happens is if you don't spot first, then when you try to drill, the drill will want to wander. And you don't have a hole where you want.

You can actually-- they may get actually now 120 degrees. Then they actually start centering themselves a little better. But this is just a little spot drill. I should actually just hit the edit so you can see the image. So this is a little spot drill. You can also many times use these to

chamfer with. Now, to select my geometry for the drilling operation, I want to show this. Many times it's the simple things.

There's a check box saying, Select Same Diameter. What that means is that when I select my hole in here, it picks all three. I love being lazy when it comes to CAD and CAM. That's just my motto, man, right? But I can still add more holes. If I were going over here, I can actually add that right there. And it just adds to it, OK?

The third tab we have is the Heights tab. Now, what the Height tab is-- this is where we're controlling things for different heights in here-- so how the machine is moving around inside of our environment. So when the machine is moving around in here, it's going to clear the part sitting in the workflow.

And then I wanted a real picture. And I spent probably about 45 minutes this morning sitting down in the exhibit hall looking on Google to find a good image of how it looks inside the machine. And then I went over to the [INAUDIBLE] booth and snapped a picture with my phone. It was stupid.

So what you can see is, right? Isn't this the life. That's just my life, man. So you have your tool here. And what the clearance page does is making sure that we can clear all the things inside of our machining area. That's really what it does. Or how it's approaching things in here. So you will see there's a clearance height way. How is it going to move away when it's got to go to the tool changes and re-track between different operations?

What I can do in here-- so right now, you will see the bottom height for this drill. Oh, and by the way, of course, when you hover over, you get a lot of explanation. When I hover over here, you will see that right now, it's looking for the bottom of the hole. What would be great if I was drilling all the way through. But I'm just spotting. I'm just trying to make a little mark in the steel for my drill to catch.

So I really don't want to go that deep. Well, if I hit the drop down, you will see that there is a lot of different options in here. And you just really just gotta find one that fits you. So in my case here, I am going to select the model top. But notice that I can just put in an offset.

So if I want to go 100,000ths down from the model top with that spot drill. That's what I've done. Click, OK. And now we have spotted that simple little thing there. I'm going to take the stock off for a second. Right? That was just spotting just like that.

Now, we can do a drilling operation. We got to drill holes the holes too. So like I said, I like to be lazy. You can actually right click on Operation. And you can go down and say, Create Derived Operation. And then you can select whatever you want to do. So we're going to just add the drilling operation. So I'm going to click on that. And it is asking me to select the tool. And I know from memory that those tools are a 10-millimeter drill.

And then I just lost my focus there. But using Derived, I don't have to select any geometry. That's anything about Derived. You pretty much taking the operation with the spotting and just copying it down. And now you can just change the things that you want to add. So in this case here, I don't want to go 100,000ths from the top anymore. That would be dumb, right? I actually want to go all the way through maybe my stock, bottom, if I want to drill all the way through the plate.

And you can even add a little bit of extra material in there if you want to make sure that you go a little bit deeper through there, you can do that. And now, we have added that operation in there-- that drilling operation. So I can see me drilling through there. Good stuff? Let's do a giveaway.

Sorry, I'm thirsty, man. This dry is just killing me. All right, I talked a little bit earlier about that I am from Denmark. That was highlighted. I know you all have been googling up and know now that with all this monarchy and all this stuff. I hope this is not too hard. We're going to figure out what flag is the Danish flag. I hope that was part of what you saw them.

AUDIENCE: [INAUDIBLE].

LARS No? Nobody's?

CHRISTENSEN:

AUDIENCE: [INAUDIBLE].

LARS I thought that was funny. See? Was that you? All right, so many people don't know that I am

CHRISTENSEN: actually living in Upstate New York. And Upstate New York actually makes some good wine. So I brought a bottle of wine from Upstate. Now, you can either drink it tonight. Or you can drag it back home. What is your shirt size?

AUDIENCE: Large.

LARS And then also there's three choices. And it is number 3. That's large. Got some advanced

CHRISTENSEN: manufacturing shirts. And of course, we have the integrated CAM inside of there. So thank you for playing along, man. Thank you for staying.

I can't remember what is in the next slide here. OK, that's fine. So the last thing I want to do here, because I wanted to have some questions. But we just talk about work set up. And I think this is important. So let's just move on here to finish this part.

So I had already to set up on this part in the beginning. And the reason I didn't want to get into the set up is, because if you're a new user, it's just maybe a little confusing. And it's just not a good way to learn. But what we got to do with this part when we machined all this stuff here is that we probably have to flip it over. We had machined all this stuff we have over here.

By the way, can you guys see how I have some red lines here? I actually have some collisions. That's true. So now we can find out where the heck-- I did something. It was just on the drilling. It's part of my offset height here. That's not a problem.

So when we have programmed this whole part-- there was a couple more features in there. But that doesn't really matter. When we are getting to this point, now we have to flip the part over on the other side and kind of like a machine off using that face mill again to maybe clean off the remaining of the stock we have in here.

So what we do inside of Inventor HSM is we creating a new set up to do that second operation. Now, there's a lot of different ways you can probably work around things. I'm just going to do it the lazy way right now, because we've already confirmed that that's kind of like the path that I'm at. Just don't tell my boss. Well, you can.

So I'm just going to hide everything here. But actually, one of the things-- so yeah, and what you maybe would have to do on a part like this is you had to make some special jaws to hold onto it. I'm actually creating-- it was brought up yesterday. I'm actually creating a creating video out of my YouTube channel where I'm going to show how to make soft jaws. So if you're interested in that, I'm going to do something about that.

But what we're going to do now is we're going to machine the backside of it. Now, one of the problems we can have is that we machine the first side. And now we flip it over. And you brought it up. I always pick it up in the upper corner-- you saw that yesterday. Maybe you've seen some videos that I made.

Actually, what I would do on this part personally, as you maybe know, I drill that hole all the hole through. I would have drilled the holes all the way through. And then I would have machined them too-- circular machined them. And this would be the hole that would be picking up as my organ. But that doesn't really-- it's just different ways you can do things.

When you go in to create a set up inside of Inventor, it's going to come in and ask me the things is in assembly. All right, so when you bring it in here the reason we are doing a setup is because we have to tell the software how things are out of the machine. You have a CNC machine. You go out to a CNC machine. That's pretty much how that thing's going to stay.

Everybody got a model where somebody modeled on the front instead of the top or the right. And it's upright and beat those guys up. You're not going to go out to the machine and set the machine up on the side, right? You have got to do it inside the software.

So that's what we're doing with this set up. It's very simple to do. The easiest thing-- anytime you are inside of a menu, I don't care if it's Inventor HSM, Inventor, Fusion, whatever. Always just start from the top and work your way down.

So this is a milling operation. So I don't have to change into a turning operation. Here we are setting the coordinate system. So this is where the G54 or where the 0, 0 point is going to be for the machine. OK?

Now, if I go to the first one, here I can select. So you see I have this nomen over here or the triad? That is displaying what is up and down-- what is x and y out of the machine. Generally speaking, the blue z-axis has got to be up. That's like coming down. That's what I gotta move in here.

Our developers are awesome. I love them. They have all these different options. But I can honestly tell you I have only used this one. I never used the other ones. I just always used to z-axis plain. So just to make it easy, just take all the other ones away. No, I'm sure people have use for them.

But what it does when you select them is I can now go in and whatever face or etch I am selecting in here, that [INAUDIBLE] z-axis will go perpendicular to it. So I'm selecting right on that face. And you will see that my z-axis will go perpendicular to that face. Are you ready? 3, 2, 1. Pretty cool, eh?

I could go down here. And I can do things with the x-axis. If I want to misalign that one. I could

also flip the direction on it. Do, do, da, doo. Whatever you want. It's fine for what I'm doing here. Then the next thing we're doing is we're selecting how we're going to select the stock.

So if we go down here and select the start point, you will see that I get all these different [INAUDIBLE] coming up here-- in this case when I'm selecting the stock. And I can just click on any of those different areas to show where I want to place it. Now, you raised up why do I always-- not always, but why do you sometimes see doing it in the upper left?

Really, the only reason is that's where my solid jar is on my vise. OK? Down in the bottom down here that's the movable end. That can swing around a little bit. So on a vise, you have a solid job here. And the movable one here. That movable can actually twist a little bit. So you could actually have a little bit of a twist right down there if you pattern that square.

But the nice thing about doing what you're doing-- I had a customer who always-- he said the same question. He's like, why don't you put it down there? I asked him, why would you put it down there? And he said, because then my code will always be positive. When I'm looking at my output in the G code, and I see a minus, I know I have done something completely wrong, OK?

Now, if you're a mold maker, you might just see that it's going to be in there, because mold makers normally like to work from the center of a mold. That's kind of like the heart. OK? A die maker will normally select from an etch. So wherever you want to pick up the part. But that's to answer yours.

The last thing I want to show here is be aware when you go over to your stock options there's some great options in here. You can select different types. And you can even select from a solid. So you can actually model your stock inside of Inventor and use it. So this is really good if you get castings, for example. Or if you have an operation that first went to the lath and now it's coming to you that you can model up as a solid and then bring it in there.

And when you have set the settings up, and you hit OK, then you will see inside of the tree, I now have a set up. And if I click the facing operation, then I can go in here. And I can select our tool, hit Select, hit OK. And now, I've created that operation to face off that. So if I did them both, you will see that we have now machined both of them here.

I could talk for hours and hours and hours. But this was really what I was planning on saying. Before questions-- I'm just going to say it again-- up here, I have stickers. And if you're using

your social media, and you've got to do it within the next 30 minutes, and you say something funny on Twitter or Facebook, and you put #JEDIENGINEER, there's a \$700 watch. OK?

And I was just told right before I came up here that the only ones who have done this to get this \$700 watch was the people in my class yesterday. So chances are pretty good that it could be one of you guys who will get a \$700 watch shipped to you. On my PowerPoint, I have this one here. I really hope that you guys will take the time. I know it's a little bit of a pain in the ass-- in the something-- I made it so far. And now it's recording.

I know it's a little bit to do these. But this is how Autodesk decides who should come back and who should not come back. Thank you so much for coming. Just ping me up for questions. I'd like to connect with you on social down in the advanced manufacturing area. I really appreciate you guys sitting through this. I hope it was helpful. Questions? Yes.

AUDIENCE: Probes Is there any support for probes yet?

LARS They are developing it right now. I think it's like-- so we just came out with a release where we
CHRISTENSEN: added some pretty neat engraving and chamfering. Have you seen those chamfering options?

AUDIENCE: Not yet.

LARS So chamfering is like you can actually deburr your part by having a little 45 degree cutter going
CHRISTENSEN: around. So it's not sharp. Well, what we actually have done with our new update is if there's a vertical wall, it will not cut into it. It will stay away from anything where it will undercut pretty much.

Anybody who's ever tried to do deburr parts on a CNC machine, you have done this. Right? You're like, oh, I'm going to go around. And then you'd miss some area. And, eh , you cut into it. So that has just been added.

I think there's one more thing coming before a problem. They're working on it. So I would think we're going to see-- I haven't seen it in beta yet. But it's not years, it's months for sure.

AUDIENCE: That's the main reason we haven't-- we have a seat of HSM. But we don't utilize it because all of our setups have all broken apart.

LARS So what kind of software-- what machines are you using?

CHRISTENSEN:

AUDIENCE: What machines?

LARS Yeah.

CHRISTENSEN:

AUDIENCE: [INAUDIBLE] we have [INAUDIBLE].

LARS So you're using the probes to inspect a art or to set it up?

CHRISTENSEN:

AUDIENCE: Set up and then also they do throughout the program.

LARS Yeah, because I have done that in a previous job. But do you know what probe it is? A

CHRISTENSEN: Renishaw?

AUDIENCE: A Renishaw.

LARS Yeah. Because all Renishaw runs on is macros. So you can actually create a macro that-- and
CHRISTENSEN: we can get that in the post that when that macro gets triggered in the post, then it will start the probing cycle for you. So you don't really need a probe. I did this all with another CAM software where I didn't have probing either.

So macro-- you can send me an email. And I will definitely get back to you. We can chit chat a little bit about it. But there is a way around it that actually works fairly good, yeah. Any other questions? But it's coming. So maybe it's not worth your time. No?

So I was little bit nervous, because this was beginner. But was it good? Was it useful? Not too boring?

AUDIENCE: Good.

LARS OK, good, It's always interesting when you putting these together if it's something that can be
CHRISTENSEN: used. Awesome! Well, if you're on social, I would recommend you do that. I also have my card up here. So I definitely would be more than welcome to grab that.

If you have any questions, the CAM scene guys around here definitely make sure that we support you and do what you need from us, all right? Thank you so much, man. Awesome!

[APPLAUSE]