

WAYNE

So my name's Wayne Griffenberg. Please. Come on in. So I've worked in the manufacturing

GRIFFENBERG:

industry probably since 1998. Yeah, since '98. I joined a company in Hatboro, Pennsylvania, where we made incubators and warmers for babies. It was a small company.

And the company was bought about five times since I was there. And each company that came in brought with them their own design of incubators and brought with them their own CAD packages. We originally used AutoCAD. They brought with them ProE we used. CATIA. We used the Solid Edge and Solidworks.

And I set out to try to learn as much as I could back then. And I was in a friendly competition with a friend of mine to become the go-to guru of CAD. And I got pretty good at it. And we had a good competition.

And I think I became the guru, the go-to guy. But if Leo was here, he'd probably have a different story. I don't think he's here. So I was pretty good.

And so my boss around the time, he was wearing a lot of hats and doing a lot in small companies. And one of his hats was programming in the CNC room where we made all the acrylic hoods, the flat patterns to bend the hoods. And I think it was becoming an old hat for him.

So he said, you know, you're pretty good with computers and CAD. You want to take over? In other words, it's yours. You're running it. You're the guy.

And so I got pretty good at that, too. We used Mastercam. And I was pretty good at AutoCAD and I would bring all of the DXF files from all those other CAD packages into AutoCAD to be able to make my own nests to bring in to Mastercam. And then for Mastercam, I put the tool paths on it and I would put that in, bring them over to the machines.

And it was a bit of a bear to do that, especially when we changed one thing in CATIA. And I got to do that whole process over again, re-nest them, put them back in. And back then, I thought it would be awesome just to have one system. One CAD system, one CAM system to be able to do that.

And I'm proud to talk to you today and being part of that company right now who's doing that. That's one of the drives from Carl, is to make that a smooth work from CAD to CAM. So we

have that today in Inventor. I'm going to show you that in turning today.

We're going to go through a brief overview, a process overview, of turning. I know some of you are new to machining and turning. So I'm going to walk through that process with you and some of the things we'll see on the most standard lathes And we're going to walk through.

So half the class will be that. We're going to talk about the CAM overview or the CAM process of what we do when we work in CAM to be able to get those good tool paths to the machines. And then the last half of the class, we're going to walk through actually a part.

We're going to program it. We're going to walk through it together. If you have any questions, don't. I'm just kidding. You can ask them. Feel free.

So as we get started. So we're going to talk about these things. We're going to talk about the coordinate system on the lathe and how that's different from the mill. You're going to get to know that. We're going to talk about the common work holding, how we're going to hold the part, where the tooling is going to be able to run to work and cut the tool paths on the part.

Some commonly used tools. There's a lot of custom tools out there. We're going to talk about a lot of the common ones, like the insert tools that we see on most common lathes. I'm going to explain to you, we're going to walk through those turning tool paths. We'll do that in Inventor in HSM.

I'll walk through with you that process. And then I'm going to explain to you-- well, you guys are going to see the post process, what it is, and getting code into the machine, what that means as far as code.

So the turning process overview. So in milling, we have a tool, a cylindrical tool, that rotates and spins and throws the chips off the side as it cuts through. Generally, that tool, in most cases-- not all machines, but in most cases-- that tool is along the z-axis.

In turning, we have the part that rotates like a cylindrical part, usually. That would be your stock. The part itself rotates. And as it rotates, we have a tool-- this is shown upside down, by the way, but some tools are like this. But generally we see them on the top on most lathes.

But the tool is going to move into the part and remove the chips that way. This is an insert tool that we're showing here. On a CNC machine, as I had mentioned, the z-axis we normally set up our part to match our machine. The way the part's going to be sitting on the machine is

where we set it up in CAM.

So we have our x, y, and z where z is our tool direction, or axis, and x and y is if you're looking down on the machine and that's where you would see your part. In turning-- going the wrong way-- the z-axis is along the center axis of the turning part itself, of the stock. Generally perpendicular to that.

When you're looking down at the machine, the x-axis is perpendicular to that pointing up. This is a standard slant machine. So the axes will be a little bit on an angle. But as you're looking down at the work part, you're still working in that x and z. So this is an important thing to remember when you're working in turning, that this is going to be your axis of orientation. This is going to be your axis that the tool's going to be turning about.

Just to give you an idea of a part. This is what it looks like. You have a turret. The turret holds the tools. It's basically a tool holder.

And we have a spindle. On the spindle, we have a chuck. Chuck jaws. Let me explain this. And the part, this is the stock part. Just to show you as you're looking down in the machine, the x-axis and the z-axis. And most machines generally, when you're working with them, will turn counter clockwise on the spindle, looking down the z.

Just to give you an idea of what this would look like. When I put this, when you're working HSM and when we work on turning, we generally like to see that datum point-- that's going to be our zero point for all of our tool paths-- right on the face of the part generally. Not always.

You can put this on the stock. But generally, we like to see that right on the face of the part, the finished part that you're going to be working with. That will be our zero right along the center line of z. This is our G54 in the code, in the machine. And that's our part datum, that's our part zero.

Give you an idea of this is just turning a face operation. So I just like to show you that running. This is doing a turning operation. The turret comes over. Now it's facing off the front of the part. And it moves away.

That's a really fast machine. It got that whole profile there really super fast. I want that machine. But that's basically how it would approach the tool. Your tools are in the turret. We're going to look at that the next slide, and it would approach the part that way.

So what you saw was the chuck. This is the spindle. This is the work holding side of the lathe. The chuck, you have chuck jaws which tighten down the whole of the part to hold your work piece tight. So there's a lot of forces going on there from the tools. Cutting chips away, you want to make sure that that's held tightly.

We have a turret. The turret is basically a tool station. And on the tool station, you have blocks, tooling blocks. And then those blocks hold the tools for each of the operations you're doing. We do have tools where we can actually have live tooling.

This means that they actually turn like a mill and they're able to cut away as you have a turn part inside of the chuck. You can have them either radial mounted or axial mounted so they can cut on the face of the part.

In most standard tools, you'll have tool holders that have inserts. Tool holders with inserts are most common. This is just a basic shape of the inserts. So you have a cut direction.

Well, in this case your cutting angles are really important when you're working in turning. Does everybody here-- I want to make sure that we're talking the same language-- have you ever worked on a turning machine on a lathe? Very cool.

So here, when we're working on the part, we have our side cutting angle. I just want to make sure the people who haven't worked with turning before have a good idea of how it's working. So here, we have our cutting angle. It's really important to make sure that you're having good tool path to remove those chips most efficiently out of the work piece.

These cutting angles are really important. So the cut angle, as you're looking at it on the side of the tool, so you have side cutting and end cutting angles, are really important because it gives clearance for those chips. It's also important to have the right angle. This way, you don't have as much friction as it's cutting through the part.

And you're going to save a lot longer tool life and you'll have more efficient tool paths that way. Relief angle. On the side of the tool as you turn it, or the side of the insert. Relief angle, this keeps it as it rotates, as the tool is going into the part as it's rotating, keeps the back edge of the insert rubbing on the material so you don't burnish. So you don't wear out the tool.

The rake angle is an important angle to look at as you're selecting your tools as well. The rake angle is set by the tool holder itself. It's the engagement. It's actually setting the chip removal,

the angle that the chip's going to be removed out of the material.

It's also setting how much of that pressure is needed to remove the chip. Depending on material, depending on the type of insert you have. Just to let you know as you're working in HSM, if you're putting tool paths on, there's always working off this imaginary tool tip.

Because we work in the x and y plane on our parts. So we're putting our tool paths within an x and y plane. The tool itself has the angle, it has the end angle and the side angle. And it also has a radius. Generally, they have a radius in the front for rigidity of a tool, as well as to be able to remove those chips efficiently.

So we generally program from the tool tip. So there's an imaginary tool tip that we program to. So sometimes when you're putting your tool paths on, you can see it. You say, well, I want to be 90 degrees like this, but my tool path is making this big arc. So that's why, because it's programmed from that imaginary tip. Just something important to remember when you're working with it.

Inserts. There's a general insert guide that you can find. I'm not going to walk all the way through these. But just to let you know that you're going to see these numbers come up in HSM. You're going to see them come up in your lathe catalogs or your tooling catalogs. This is just a standard number that you'd find with a lot of your tooling manufacturers.

So these numbers that you're looking at here define a lot of the tools when you go to like Sandvik. And you can actually find a lot of this information on our website. If you go to Autodesk, or actually go to the Knowledge Network website. Autodesk Knowledge Network. Type in HSM turning. You'll find all of this information in there.

And I can also give this presentation to you afterward. So these are a lot of the parameters that go to make up specific inserts. Now, there's a lot to think about. Now, here's the different types of tools that you'll find insert tools.

But there's a lot of things. I'm just going to jump back real quick. A lot of these things that you're looking in here take account for what you're trying to get the best finish, the most efficient chip removal. So you're going to try to find the right insert for the material you're working with. You're trying to find the right insert for the diameter of the geometry.

Mainly, it's driven for your speeds and feeds. And we'll talk about that in a few minutes. So

these different types of tools that we would do with operations on the turning machine. So the most basic, you have face and turn. So facing, cutting off that face of the part that you saw earlier. And just basically doing roughing, trying to remove as much chips off the side of the profile of the tool that you're trying to get.

For grooving-- sorry, keep hitting the wrong button-- if you want to put grooves on your part, there's general groove tools. There's a lot of custom tools like this that you also find in the industry. But these are general ones that have inserts.

Bore. If you're doing any inside grooving. If you're going to groove out of a hole inside of a part, you're going to use a boring bar and a bar. So we also have threading. You can make threads inside and outside. The inside is usually very thin.

And we have cut off tools, to cut it off of the stock. Speeds and feeds, I'm not going to get into. But as we get into the calculation, you can find this information-- really, what your speeds and feeds, you're really trying to find--

So your part is turning at a certain RPM. It's turning at a speed. And you're also feeding into the material with the tool. So you're trying to find that optimum speed and feed that's going to give you the most chip removal, the most efficient way, the best tool path that you can get for the finish at the end.

So you're really trying to dial that in. There's no one number. It depends. There's a lot of dependencies on the material that you're using, the diameter of the part, the type of insert that you're using. And also what you're trying to accomplish with that tool path.

You might be doing a roughing tool path. You may be doing a finishing tool path. So there's a lot of things that determine this. But if you go and look into a lot of Sandvik, if you go into a lot of different tooling websites and look at the different tools and options you have, you'll find that as you start to work down and you answer those questions if you look at a lot of the angles that we talked about, the geometry of the tool, the geometry of the part you're trying to make, and the material.

When you work through that, you'll find at the end a speed and feed calculator to get you started. And then from there, it's really up to you. As you're working with your machine, you're trying to find that optimal chip removal. You're trying to find the best tool path.

You can listen to your machine. You can hear if it's getting a good tool path or if your speeds

are too slow or your feed's too fast in the material. See that in the chip. You also will wear out your tool quicker if you're going too fast in the material.

But the websites do a really good job of getting you started. So that's pretty much walking through just a real quick overview of what we see in turning, the type of tooling, how it's held in the machine, using turrets, using tool blocks. A lot of this information, this is new to a lot of you?

OK. I hope that's helpful. So the CAM process real quick, we're going to walk through. And I'll give us a lot of time to be able to put some tool paths on the part in a little bit. So the CAM process, the first thing we do is we either create geometry, we bring the geometry in from another CAD package.

We can work with STL models. You can bring any type of model you need in. So that's our first part, is being able to create or open the part in Inventor. The next thing we do is we define our job set up.

So you want to try to match as close as you can to that actual piece of stock in your machine. You're trying to set it up in this 3D world. We do have a nice powerful 3D package that we're working with. Why not use it to that advantage to be able to set up?

You can actually draw up your chuck in there. You can draw your turret if you wanted to. It'll be helpful for you. You can actually see when you move things around, if you have another tool that may be on deck that may hit your part. It's a good idea to try to bring that in.

And there's a lot of places where you can download these. I know HAAS offers a lot of their models that you can download to bring into Inventor. So it's a good idea. So anyway, for defining your job set up, this step is really trying to define what you have going on in the machine with all the parameters and the places where your actual part's going to be.

We're going to define, in CAM, the operations. The operations you need to get the David inside of the marble. You're going to remove the chips away to end up with your part inside.

Well, these are just a few of them, but there's different operations you're going to use using specific tooling to be able to do that efficiently. So that's the next thing we do, is define those operations. We can simulate right inside of Inventor. We can simulate what's going to happen on the machine before you go to put any code in the machine.

And at the end, we post the code. So the post processor is basically a decoder ring. If you think of it that way. So you have generic information. You have the basic data that's inside the tool path parameters that are inside of HSM.

And the post processor, it's a JavaScript processor. And what it's really doing is just decoding that information and reforming it into language that your machine control can understand and read and interpret to be able to move the tool, the turret in, to be able to remove those chips the way you did virtually. So the post processor is an important piece to get the good code to your machine.

And that's an example of what some code would look like. That's actually the code that came out from the generic FANUC post from that part we were just looking at. And just to let you know, there's a lot of different machines. There's HAAS machines. There's different controls as well.

This is just showing you some diversity, but there's a lot of different controls. Siemens controls, FANUC controls. So there's a lot of different controls that have a lot of different language. So a lot of you would have different machines. So you'd need a different post, and you can tune that post in to get the exact code you need for your machine to do it most efficiently.

Are there any questions up to this point? Cool. So let's get right into it. The next part is we're actually going to do some Inventor HSM. So I'm going to bring up a part. It's a basic part.

We're going to put some of those operations we looked at. Any time as I walk through, please feel free to ask if you have any questions and something doesn't seem right. Please feel free, as I walk through. So let me do this.

So I opened up this part before. This is that part we were looking at. In Inventor, this is basically what you'd see in Inventor. When you open up a model or you're working with a model, it has the History Tree in the browser. You also have the ribbon bar up top with all the different applications, all the different features you're trying to do.

As you hover over a lot of those-- you know this in Inventor-- it brings you that in context help that you're used to. Now, I'm going to work in the CAM side. So right now, I have Model and Features in the browser. But if I go bring up the CAM, then this is going to be embedded.

It's actually, when you open it up and you install it, it's an add-in. But it's completely integrated inside of Inventor.

AUDIENCE: Which version are you running here?

WAYNE This one? This should be the latest released version.

GRIFFENBERG:

AUDIENCE: But I mean, HSM, HSM Express?

WAYNE Ah, OK. This is HSM Pro. HSM Pro. Actually, no. You know what? I do have the multi-axis. So

GRIFFENBERG: we're actually looking at the Pro version.

So just like we do in mill, to be able to make the tool paths we do in mill, we also have that in turning. Now, you don't need the Pro to have turning. You can have the basic HSM, and you'll have mill and turning. There's no separate module for turning. You'll be able to do all those turning tool paths in the basic version.

AUDIENCE: When did that happen?

WAYNE For turning?

GRIFFENBERG:

AUDIENCE: Yeah.

WAYNE The beginning of the last version, probably one in the update of 2015 in April. I believe it was

GRIFFENBERG: April.

AUDIENCE: Got you.

WAYNE So just like you saw in the model view, you can also hover over and you can see that there's
GRIFFENBERG: the in context help completely integrated. So I'm going to click on Set Up. And when I do, you see the browser will change. And right inside the browser, I have all the parameters for the tool path that I'm working with.

So here, automatically it recognized that I have a cylindrical part. And it determined whether I'm doing milling or turning. So it went into the turning or mill turn mode. If I hover over, you'll see it has that same in content help to help you.

So I have the turning mode as I'm working through. It's also picking up the z-axis. If it wasn't

the correct z-axis, I would just click down here and I would choose a feature on the part that's along that same z-axis. It'll put the z-axis in that position.

If I wanted the z-axis on the other side, I would just simply flip the z-axis. And now I'm working on the other side of the spindle. But I wouldn't really want to do that here because I won't be able to cut anything. So I'm going to flip that z this way.

If you notice it, if you look, it's going to have the x this direction and z this direction, which is going to be the same way it's set up in my machine. If I look here at model, if I have any features-- if you have any features when you're working on it. Maybe a hole through the side. Maybe you have these cut a little bit.

If you have any features that cross through the extent of the part, you can do this. You can select the model and you can say I want to do a Spun Profile. And what that'll do, it'll look at the extents all through the part. It will take that extents and spin it through and it'll use that for your profile.

And then your tool paths will use that outside profile. Any features you have inside, you can select those features directly and it'll machine those features. So it's a good thing to know that's in there. So you can select the model and do a Spun Profile.

The next tab over is my stock. So I can define the stock. It came in and it recognized automatically where the extents of the part is and it gave me a stock that size of the extent of the part. Does this in mill as well.

I can choose to do that. If I want to say I want this to be three inch diameter. Notice it gives me update right in the spring, right inside of Inventor. When I made that change, it showed me live right now what it's going to look like.

So I can change that back and I can make that 2.5. What I want to do is I don't really want to use a set size. My stock can come in bigger, smaller sometimes. What I like to do is I want to determine I want to be able to set my stock up to always be a tenth bigger than my part.

So I can go into here and I can say Relative Size Cylinder. And for my radio offset, I'm going to make that a tenth. It'll be a tenth bigger no matter how big the part gets. Front side offset. So if I look at this from the front, I do want to be able to do a face tool path because I want that front of the part, the front of the stock, to be nice and smooth.

So I'm going to put a tenth rate in the front of the part like that. And now on the back of it, as you saw the chuck, I'm going to have a chuck clamped on here. So I want to be able to part this off later because I need room to hold on to the chuck. So I'm going to add to the back side an inch.

So now I can hold on to the back side. I have at least an inch in there to play when I go to part off the part. And then the last tab over here, this is my post processing information if I want to give it a program number when I go to post it out. I can also give it a name, if I can spell. There we go.

And I'm all done with stock set up. If I hover over here, I can see my stock is all set up. Notice it's going to match what I have in the machine. Now I want to face this off.

So I'm going to go to my face tool path. So I go up to Face. It gives me that profile of the part. It's looking at the profile, the extent. It's looking at the difference between the profile of the part and where the stock extent is. So it knows how much material to remove from there.

One thing I love about our CAM, it's always the same. If you've seen this in mill, you'll have five tabs in the tool path. The same is in the turning. So you'll have the same five tool paths. You're going to work across. You're going to have the same work flow going from left to right. Your tool, your geometry.

Your heights are now radii, because your radii is from the center of the part out. You're also going to have your passes. The passes are the parameters that you can change as the tool is entering, or as it's cutting in the material. And then you have your linking, which is the moves that the tool path is going to take to get to the part and away from the part.

So it's always going to be the same interface no matter what tool path you're working with. So working from left to right. I got a tool. Go select my tool.

Right now, it opens up the library. And right now, it doesn't have any tools. But when you work in HSM, and if you open up a part later, it'll remember all those tools you worked with before. In this case, it's the first time I opened it.

When you're working with a library, it's a pretty good library. What the developers did is they found a lot of the standard tools out there with the inserts and holders. And they filled the library with some pretty good speeds and feeds to get you started. A lot of them are steel-

based.

But you can go in here and change these. If you're doing aluminum, you can change the speeds and feeds for aluminum. So they did a good job of loading up the tool library.

So what I'd like to do is I'm going to find a tool that I'm going to use for facing this. So I'm going to go into Tool by Type. Going to scroll down close to the bottom. A lot of these are mill tools. But if I go down close to the bottom, I see now I have turning tools down here.

If I go into General Turning-- and some of these should start to look familiar from that code we looked at a little bit earlier. So some of these are predefined, but you can always go back in here and re-edit the tool if you need to. I'm going to grab a roughing tool, general turning tool right there.

Going to select it. It brings me it into the graphic environment so I can see it. It's hidden so I can see the insert behind, and where the insert's going to be, because that's the most important. That's where my cutter is.

So it gives me an idea of the size, where it's going to be in my design. It also pre put in the face tool path already. So I don't have to select a new geometry. It knows the extent of the stock. It knows where the front of the part is. And it put that tool path in automatically.

So I don't have to do anything else. I just say OK. And now I have this face tool path right there. It's going to remove the material from the face. And we can simulate that right here, right inside of Inventor.

So if I hover over this or I go up the Set Up, I go to Simulate. It brings the tool in. I can also show that with the stock material. Again, this is right inside the part, right inside of Inventor. Completely integrated.

I don't have to leave this environment. It doesn't have to copy this out as an STL or anything. It's right in that environment. And I can hit Play and I can go through and I can see that material being removed. If it clashed with the stock or if it clashed with the part itself, I'd see a little red tick down here and the tool would turn red.

That's the face tool path. I'm going to go in. I'm going to do the profile of this part. Let me go up to the side here. I'm going to zoom in a little bit. I want to do the profile of this part with a roughing tool.

I can use the same tool. So I go up the Profile. Again, if I hover over, it gives me that in context help to show me what that tool path is going to do. I can use the same tool. The same roughing tool will work.

So I'm going to go. I'm going to leave the tool alone. I'm going to go into my geometry. I don't really have to select any geometry because I did this spun part, plus it recognizes the extent of the part and the extent of where the stock is. So I don't have to select any geometry.

So I'm just going to say OK and leave everything else out of the box. And I get a tool path. Two clicks. There's a good tool path that we can run in the machine.

You can change any of these parameters. If I right click in there and I go to Profile and Edit. If I wanted those step downs to be finer, I would go to the Passes tab.

And I can change that maximum step down to be something like that. Say OK. It'll change, update real quick. Recalculates. And now I have a new step down into the part.

AUDIENCE: You can change the number of pins. For example, this one would have 10 heads in order to get [INAUDIBLE].

WAYNE GRIFFENBERG: Sure. Yep. You can change the passes. If you did a number of roughing passes and I go into here. And I say roughing overlap. I can do finishing passes, roughing passes.

You're saying can I go by number and not by distance? Right now, we don't really have that in there to say I want this many passes, calculate it. Right now it's set up to say I want each step down to be this much.

And by default, this is a roughing tool path. As soon as I put it in here, it's the first profile I did. So it turned on Stock to Leave. So right now it's leaving 500,000 is on there. If I want to change that, I can change that here. It'll change.

Let's say I did change that and I want to leave on, let's say, I'm going to put, oh, 80. Oh, eight. So 800,000. And every single time I do a tool path in this material using that tool, I want that to always be. If I right click in the field, I can say Make Default. And every time I do a profile, it's going to leave 800,000 on there. Every time.

So let's leave 800,000. So we're going to say OK. So I'm able to go back in and change

parameters after I put the tool path on pretty easily.

I like the way that's set up. And what I'm going to do is I'm going to do a finishing tool path based on that. So I have a couple different ways I can do that. But right now what I want to do is I'm going to highlight this. I'm going to say this is my roughing pass.

So I'm going to highlight it and soft click with my left mouse button. And I'm going to call this rough. That's my rough pass. I'm going to go and right click, Copy, Paste it. And this is going to be my finishing pass.

I duplicated the same exact tool path. This one's going to be my finishing. So I'm going to say finishing. And let me be consistent. I'll just call it Finish.

Now, I want to change those parameters. So I right click. I'm going to go to Edit. Now, I don't want to finish with that same tool because I'll get the same profile. I'm going to get those little edges and those little creases in there.

I'm going to use a sharper tool, a different angle tool to be able to get in and remove the chips more finely and be able to leave me with that nice profile. So I'm going to change the tool. Again, working from left to right.

Go to Tool. There's my library. I strongly suggest if you're working with HSM, create a library of your own tools that match what's on your turret. It's a great way to work because then you're changing that one tool. If you change the insert, you change it in your library, you're one to one.

Instead of always searching for a tool and then editing it. In this case, I have some tools. I have some libraries that match the machines I'm working on. So if I go into that library, I have some finishing tools that I worked with before.

I can bring that tool in. If I select it, it brings the tool in. It brings in those feeds and speeds, that information from that tool. And it brings it in. And I can change these on the fly if I need to.

If I'm getting a rough tool path on there and it's not doing what I expect, I'm going to go back in. I could change my speeds and feeds right here on the fly, post the code back out. Find I'm getting a better tool path, then I save those feeds and speeds back to the tool in the library.

But what I have here right now is only in this part. It's only in this tool path. So I brought that

tool in. Just to show you, I can go in here and easily right click if I want to make changes to that tool. I right click and I can change any of those aspects.

I can change the insert, the size, the angle. I can change the corner radius. I can change all that information. The holder itself, if I have a change to the holder. I can change that and save it right back in.

Feeds and speeds. The set up. The orientation of the tool. This, you can just right click, go in here, and edit any of that. Save it back to your library and you're good to go forward.

So I'm going to select that tool. I want to change some parameters. I want to go in. I don't want to leave any stock because I copied this from the rough. So I'm going to turn off Stock to Leave and I'm going to turn off Roughing Passes.

I want one nice smooth pass. Now, here we could change the number of step overs in the finishing. So if we're going to do finishing, we can change that. We're not doing roughing passes. Now we can change our number of step overs.

So I'm going to leave one step over for now. Say OK. I get a nice finishing tool path on there. It's going to follow that curve. So let's take a look at what this tool path looks like now.

I'm going to go to Set Up, Simulate. I'm going to walk through. Simulating, it's going to do the facing. It's doing the roughing now. I have a lot of passes in there, so I can speed this up a little bit. You could also make it go backwards, too.

If you see where it was hitting somewhere close, you can back it up and you can make it go back through. And look, it put material back on, just like it did in mill. It's one of those neat additive subtractive things.

And as it's running, let me show you in the simulation, if I click Info. The Info button's showing where the tool is, where the tool tip is at the time it's running in the simulation. I'm also getting feedback as far as what's the spindle speed, what's my feed rate at the moment.

I can slow that down and take a look at what's going on in that feed rate. It's also giving an overall time for each operation. It's giving me this information feedback, the volume, the cubic inches it's removing from the material.

AUDIENCE:

Is that new in 2016, that Info tab? Or has that been there in 2015?

WAYNE As far as I know, it's been there. But I'm not too sure if it was or not. I remember it, but I'm just trying to remember how far back I remember seeing it.

AUDIENCE: It's really useful.

WAYNE Yeah. I think it's going to be really helpful, especially if you're trying to quote and you're trying to find out how long it's going to take. Or the material, you're trying to find out the best material to use. It's a really good, helpful tool to be able to walk through.

So I'm going to get out of here. I'm going to close. Again, completely integrated in Inventor. So I'm going to go and I'm going to drill this hole through. And I'm going to profile.

I'm going to use a boring bar to bore this out in the next couple steps. So I'm want to go into Drilling. So if I go to Drill. Again, it changed a little bit here only because drilling cycles are generally just a cycle. They don't need the linking parameters or passes because you're defining all that in your cycle.

So it's going to do the GE83 cycle. It's going to go through the drilling cycle. And when it's done, then it picks up the next operation. So here, I'm going to pick a tool.

I'm not sure. If I go over here, I'm in Drill. And I'm not sure what that hole is anymore, as I've been talking. If I hover over that hole, it gives me feedback right on the screen and shows me, oh, it's a three quarter inch hole.

So I'm going to go over here. I'm going to choose the tool. I have a three quarter inch tool I was working with on my HAAS before. So I'm going to use that three quarter inch drill.

So the geometry I'm going to select would just be that hole phase. Now, the hole itself, all I have to do is just select the cylinder of the hole. The hole, the top of the hole is already into my part. And the bottom of the hole is already out of my part.

What I want to do is I want to define. I want that drill the start up here because there's still stock material there. So in the next tab is my Heights. So in the Heights tab-- and it's looking-- so what I can do is I can change where the bottom height and top height are.

So it's giving me the clearance. This is where I'm cleared to move to another place on the part. My retract height is where it's clear from that tool path. So I want the height to be at this face.

So if I go to the top height, I'm going to go to Selection. I can choose the stock top. I can choose the model top. But I'm going to go to Selection, just to show you can do this. And I'm going to select an edge on the front of the part. Let's say that edge.

I can also select the front-- oh, maybe it'll let me select it. Did I do something silly there? Oh, I select the edge. You could select the face. You could select the edge.

And it put the top. You see that light blue right there is the top. That's where the hole is going to start. And I want the hole to end down here. So I'm going to choose the bottom height. I'm going to go to Selection and I'm going to pick this face-- and why did it do that?

Did I select something wrong? It's only when I'm here showing people. Let me try that again. Bottom reference. Right there. Usually it's easier than that, guys.

OK. I guess I selected something wrong. But you could always go back in real quick and select it. You could also put in a value there, too. If I want that drill to come all the way through and I want to make sure the drill tip is there, so about left with that little concave chamber at the end, I can add.

I'm going to say because I'm going from top down, I'm going to say negative 0.25. It'll stretch the bottom down that far. So now I have the drill going through a quarter inch further. My cycles. You have the same cycles that you'd find.

You can do chip breaking, breakthrough. You can do boring cycles if you wanted to. I'm going to leave it as standard drilling and say OK. I have a drill path. And then what I want to do is I want to use a boring bar to bore this face out.

So I'm going to go into My Profile. My Profile sees a tool like that. How am I going to get a tool like that inside of there? Well, the first thing I'm going to do is I'm going to go here. Instead of doing Outside Profiling, I'm going to do Inside Profiling.

You set this at the Tool page. Inside Profile is now going to change and it's going to look at features on the inside of the part. So now if I go to change my tool, I'm going to change the tool to be an inside boring bar. If I go to my lathe inch, and I'm going to go and choose this turning boring bar.

You see it sideways. If I bring this up, I'm going to select that boring bar. It gives me the view

of what the insert would look like in the head of the boring bar. If I want to look at this, I'm going to Edit real quick. I just want to show you.

When that's set up, I set it up as 90 degrees because it's sitting in my turret in the block at 90 degrees with the insert up. So this is where I would set that up in the tool library at 90 degrees. So a lot of people ask that question, is I don't know how to get the tool to be in the inside of the part, or how do I get it to face the right way.

And they change it in the tool path. But you want to do it at the tool itself. And you do it right in that parameter. So there's my boring bar.

I'm going to select that. Here's an important part, and it's a tricky part. Well, this isn't it. I'll jump into here real quick. Let's do that. Let's talk about the tricky part.

If you've seen HSM for mill, HSM for mill has your heights and has the heights from the top of the part out. In turning, your heights are coming from the center line and working its way out in radii. So the center line is what we would call the bottom of the part in mill.

So when we're working from the bottom up, it's a good idea to work from the bottom up. Start with your clearance, because your clearance is going to be the inside most of the part. You don't want the tip of the tool to go down past the radius because now we're doing part of the diameter. We want that to always stay above the radius.

So my clearance, as you can see, is at zero inches. That's right on the center line. So I want to change that. So I'm just going to go into my clearance right now. I'm going to leave it where it is at the stock ID.

I'm going to make it a quarter inch. Ah, let's make it an eighth. Because I want to make sure I don't touch. That's actually where the tip of the tool is allowed to go in relationship with the center of the part. So the tip of the tool, I don't want it to go past the center of the radius. I want it to stay above.

So I'm going to give it some distance above. And it makes sense. Your part, make sure it's at a place that your tool can actually retract to without hitting the back of the part. So working from the bottom up.

My inner radius-- I know it's a little confusing-- but my inner radius, I'm going to select that. That's actually going to be where I want this thing to start. That inner radius, if I go to Inner

Radius Reference, I'm going to select that drill hole. That's where I want the tool to go and start and work its way.

My outer radius, if I go up to my outer radius, my stock ID, outer radius is going to be-- I'm going to select that outer radius there. I don't know why I did that. But going to select that outer radius there.

So my clearance is the innermost where my tool can go inside the part. My bottom, if you will, my inner radius, is the place where I want that tool to start into the material. And then my outer radius is where I want the tool to end in the material. If I set that outer radius all the way above the part, it would try to look above the part to continue.

Now I'm going to jump back a tab because I went into Geometry. And there's no geometry to select, specific geometry. What we have is Confinement. We're going to tell it how far into the part do you want that tool path to go.

I want to confine it from this face to that face. Because I don't want it to bore inside the drilled hole already. You can, if you have a long enough boring bar. You can dot hat. No problem.

But in this case, I don't want it to go past that. And I don't want a backside stock offset. I should have showed you that earlier, but that's OK. We'll go back. But a backside stock offset simply means it's going to leave room behind the part here. And I don't want that to happen here.

But it'll leave room for you to be able to part it off or be able to work in the back of the part. But here, I don't want any of that. So I'm going to turn that off. And I think I'm good. If I go into my passes, I can go in here and I can set how many roughing passes, how many finishing passes if I want to leave stock.

In this case, I do because I'll come back and finish it later. Just going to say OK. And I get this internal profile using my boring bar. So that's the important part to remember when you're working with internal tooling or you're working with an internal profile, work from the inside out using the radii.

It's a little tricky, but it's the way we work with it in HSM. So I can come back here and I can get a nice sharper tool. I can copy this down and make that a nice tool.

I'm going to leave it the way it is. I realize I forgot to do any grooving in here. So I want to groove that. So if we do this, I go look at the set up real quick and I simulate it, we'll see what we did so far. I was excited to do the internal stuff. I forgot to groove the outside.

So we'll go back and do that. Could speed this up a little bit. And you could also do this. You can also jump to the next tool path. See the little button right there? I could jump to the next one.

There's my finishing. I can jump to the next one. There's my drilling. And I can jump to the next one. And there, it's doing that internal profile right there with the boring bar. So far, you guys have any questions? Yeah?

AUDIENCE: I notice you're orbiting the model while you're simulating. I see you doing a 3D mouse up there. Can you do that with a normal mouse, too?

WAYNE You can. Yep. You can do that.

GRIFFENBERG:

AUDIENCE: Orbit while you're simulating?

WAYNE Sure. Sure, absolutely.

GRIFFENBERG:

AUDIENCE: Does the mill simulation do that, too?

WAYNE Yep.

GRIFFENBERG:

AUDIENCE: I notice it paused or something while I orbit while simulating.

WAYNE That's interesting. It may be a graphics card.

GRIFFENBERG:

AUDIENCE: [INAUDIBLE] when you're rotating the model.

AUDIENCE: Yeah. I like being able to orbit the model while [INAUDIBLE].

WAYNE True. Yeah. If I bring this back to the beginning and I go to Simulate, if I hold the Shift button,

GRIFFENBERG: this is what you would do normally in Inventor. If I hold the Shift button and the middle mouse button. Now it's going to shift a little bit only because it's waiting for my mouse to get done

moving, you see?

AUDIENCE: OK.

WAYNE Because I'm holding Shift.

GRIFFENBERG:

AUDIENCE: With the 3D mouse, did it pause or did it catch it?

WAYNE Ah, see here? It can do this. It's not pausing.

GRIFFENBERG:

AUDIENCE: So how do I get that with a regular mouse?

WAYNE I had the same exact problem, so I don't know. I don't know.

GRIFFENBERG:

AUDIENCE: Why [INAUDIBLE]? rotation [INAUDIBLE].

WAYNE Yeah. So if I hold Shift and I rotate it, you see it does pause.

GRIFFENBERG:

AUDIENCE: Right. Yeah.

WAYNE I'm Not sure how to get it to go fluidly that way. But I can look into that. I'll give you my card

GRIFFENBERG: later. Send me an email, and if I find an answer, I'll give it out to you.

Yeah. It'll go smoothly with the 3D mouse, though. I could jump to the end here. And that's what it's going to look like when it's done. I close that out.

I'm going to do grooving, because I jumped ahead. But I'm going to go put some grooves in here. So if I want to go the Groove tool path here. Add some grooves. I'm going to go to my tool.

Again, it's always the same workflow. Working from left to right, the same tabs. The same steps. It's always the same. It's easy to find where your tool paths or where your parameters are. Easy to work with.

So I'm going to do some grooving. So I can scroll down. I can do by type and I can find the grooving tool. Scroll down here and I can go down to grooving. I'll pick this grooving tool right

here. The second one looks pretty good.

I can bring that in. It brings in the feeds and speeds with it. It gives me that graphical view of what it's going to look like along with the part. It does bring it right to the zero there.

And what am I going to groove? Again, the confinement. So if I go to the Confinement, this is saying where do you want to-- it's going to try to groove everywhere that tool can go. It's looking at the part. It's looking what's left on stock. And it's going to try to put that grooving tool wherever it can go.

So what I want to do is I want to try to limit wherever it can go. You don't have to. But I want to limit between here and here. So if I go into Confinement, I'm going to select that face and that face. And it's going to confine it in between those faces.

I could choose the front. I can choose anywhere on the part. I can choose edges, too, and it'll work. So I want to confine that grooving tool path right there. My radii, if I want to add some clearance to it, I can. For passes, I can tell it whether I want to peck or not peck. In grooving, that's important so you don't overload the tool.

You have most of that surface area. You have the tool. You don't want to have the back edge hit the tool. So you can peck as you're moving down. So you can do that.

You can also do a full step down or not. You can also do sideways, where it can almost profile as a groove down. So you have those options that you can work with as you're grooving. So I'm going to leave them like that out of the box. I'm going to leave pecking on.

Roughing passes. We can leave those on, too. I'm going to say OK. And I get a groove tool path. So it's looking at what was left from stock and what's left in the profile, and it's going to groove all that out.

We can simulate that real quick. I could see I'm hitting on the back of the tool there. So I might want to go in and change my pecking parameter there. See it's not lifting all the way out, and it's hitting?

But this is a good thing, because it's showing me that. You see these little dings down here? It's showing me places I want to take a look at. You don't want that to go, because you're hitting. You're actually tapping into the side of the part.

And it's telling me I want to go back in here and I want to change that peck. I want to change the way it's doing that. So I'm going to close this out, go back into the Groove, go into my Peck here. I'm going to actually change my retract, Pecking Retract. I'm going to make that an eighth of an inch.

Now I'm going to be able to retract all the way out of the part and it shouldn't hit. So that's grooving. And now I want to part this off. I want to do a parting tool path.

So I'm want to go up to Part. It automatically recognized I had a grooving tool, and it said do you want to use that grooving tool. It brought it in automatically. It actually recognized the last tool I used. It does the same thing in mill.

So it recognized the last tool I used. I can go in. Again, the same work flow. Going to go to Tool. I'm going to go choose another grooving tool or I'm going to go into my HAAS. I'm going to use a tool that I have already used in the past for parting it off.

Select that tool. That's a good parting tool. I don't have to choose anything else. It already knows the geometry. It already knows where I want to part it off. It knows the extent of the back of the part. I don't have to choose that. I don't have to select it.

So we've got a pretty good part. We did a couple simulations there. What we're really looking for and why we're doing a lot of this is we're trying to get that good code to the machine. And we can take a look at that. I want to walk through real quick a setup sheet just to show you.

So if I highlight Setup One and I go to Setup Sheet, it's asking me where I want to save it. I'm going to say Open. And it gives me this HTML. It's a setup sheet. It gives me a graphic view of what I had on the screen.

It's showing me where my WCS is, position on the machine or on the part, where that zero is. And as I walk through, it's going to show me more information as far as all the tools being used. Now, in the mill side, right now you can see a mill tool.

They're working on this on the next update. Right now, we don't see turning tools in the setup sheet. But they're working on this, and we should see an update soon where they're going to have the turning tools in here as well. OK

So it's giving me all the tools were being used. The maximum feed rate. The maximum spindle speed throughout the entire program. As I walk through each operation, it gives me the

distance. The way it's set up, you have other information as far as the total estimated time for each operation.

So it's a pretty good setup sheet to be able to have on the floor to know which tools go in the turret and which tools are going to be used and what's the total time for each tool. It's a pretty good setup sheet out of the box.

So let's get some code. Let's post some code. So I'm going to go into Post Process. This is our interface for the post process. I have a post up here.

Our post, I think there were about 100 posts that come with HSM. And these are all generic posts. We have seven guys now, I think, on the post team-- I think there is-- who work with the post to get good feeds and speeds to be able to work and get good generic post running.

And this comes with. They're all open posts. They come with the HSM, our CAM. Let me get a generic FANUC post I can bring up here real quick. Turning post.

So you have a list of different machines. You have HAAS machines, Tormach machines. You have-- excuse me-- Mitsuru machines. You have a lot of different machines in here to work with. And when you start working with it, try some of the generic to get started with to make sure you're getting good code.

And then we can work with you, too, on your post. You can work with a reseller as well. Let me go into-- where was I? Generic FANUC.

So these posts are not encrypted. If you know JavaScript, you can always go in here and open it up and you can edit your own post. So again, this is the decoder ring. It's a script that tells that information. It pulls the information in, the generic information from all of our tool paths, all those parameters. And it's rewriting them in a way that it's going to give us code that our machine can read.

You're going to bring that code into your machine control. It's going to interpret it and it's going to move the tools exactly where you want them to be. So I'm going to hit the Post button. There are some switches here, too.

Going to go to Post. It recognized the number I put in there. I think I used this before. I'm just going to overwrite that one. And that quick, we get code that we can actually send right now to our machine.

And it would run this. And you'd be able to get that part. So within a half an hour, we're able to get that whole part programmed, be able to get code out to the machine. Any questions about that? Sure.

AUDIENCE: [INAUDIBLE]

WAYNE Most of them. The most. So HSM is really the integration within Inventor in this case. The
GRIFFENBERG: generic information that we have in HSM, the post processor is really what it's going to interpret, that information. It's going to decode it.

It's going to output it to your particular machine. So if it doesn't cover your machine, we can work with our post group. We also have resellers that we can work with as well to work or change the code and make it work in your machine.

If you start working with it and you're posting generic code out of here, and use the generic code-- if you can get the code and edit it and be able to put it in your machine and get good code. If you can save a sample of that good code and also a sample of it in CAD or CAM and CAM, and you can get that to either your reseller or back to us, we can have our post team update that post so it always comes out to the code that matches your machine.

AUDIENCE: [INAUDIBLE]

WAYNE I couldn't hear you.

GRIFFENBERG:

AUDIENCE: How many axis of the machine? For example, small machine like [INAUDIBLE].

WAYNE You're talking about the machining axis?

GRIFFENBERG:

AUDIENCE: Correct. But [INAUDIBLE].

WAYNE Ah, gotcha. So right now, we have posts that work with mill turn. So that can give us our y-axis
GRIFFENBERG: and our x-axis We're really limited, really, to that y-axis and z-axis. But the label is y or z. But we're able to work with that code for mill turn machines.

But for multi-axis when you actually have, let's say, a b-axis, a c, and multi-axis that way, we're limited in our post to be able to do that right now. Yes?

AUDIENCE: Is there some kind of headers so you can create or put in your own tools for the geometry build?

WAYNE Yes.

GRIFFENBERG:

AUDIENCE: So you can get your actual tools.

WAYNE We can. Right now in turning, that is called a form tool, to be able to put your own geometry in.

GRIFFENBERG: We have that in the mill side. But right now, we're behind in turning to be able to do that with the insert tool.

The team's working on that. I don't know where they are in progress with that. But there should be a future update to have the form tool for turning. But right now, it's in mill, but not in turning.

AUDIENCE: OK.

WAYNE Now we get back coding. We have a few minutes left. Now, we did some code. We put some

GRIFFENBERG: tool paths on here. We're able to get a setup sheet out. We're able to get some good code.

And now we have to go and change the model. Now there's an update. Engineering's got to make a change. We have to change the model.

So what happens then? If I go into the model, and let's say we go into the sketch that defines this model as a rotated part. Let's say the overall dimension here goes from an eighth of an inch to say 1.35. That's a change.

And then it gets a little bit wider. It goes a half inch here. And you can make a lot of different changes here. We have a lot of changes in the part. So that changed. Finish the sketch.

Go in here. Go to my CAM. And notice all the tool paths turn red. They have a red X on them. Means something has changed, either a parameter or the geometry. So now what do we have to do?

Do we have to reselect all that stuff? Well, if I hover over Setup One and I go to Generate, it recognizes all the new geometry and automatically updates every one of the tool paths. So you don't have to reselect anything. It'll recognize any new geometry.

That's a powerful thing to have in CAM. It's great integration. And this will work. If you bring into an assembly, you can try this. Work with an assembly inside of Inventor. Bring a part in.

And it could be a part that's a CATIA part or something that you import, IGS part. And bring it into an assembly and put tool paths on it. And if you put tool paths that you're not really defining too much of the distances in there, put tool paths on it. And then if that part changes, bring it in again into IGS in your assembly.

Update it. It'll recognize those features because you did it as a spun profile. It'll recognize all the new features and automatically update. That's really powerful.

And Inventor has any CAD. So you can bring in the natural model from ProE. Or you could bring it in from, sorry, CATIA. You can bring it in from Solidworks, Solid Edge. So you can bring the Solidworks part in, make the changes, have it update. It'll also update your Solidworks part.

So you have that any CAD that you can bring a part in. So if you work parallel with another group who have a part in Solidworks you don't have to worry about redrawing it. Just bring in that native part. It'll put in all the tool paths and update. If they have an update, just bring it back in again. It'll update automatically. Really powerful.

So you see it updated here. If I go to post the code again-- let me go to Setup, go to Post Process. Pull up the post again, go to post it. That quick, I'm able to get those tool paths updated. And now I can send a new tool path to the machine. It'll view the new geometry.

Any questions? So I hope this information was helpful. And I walked through a little bit of the-- I usually have a summary there-- but we walked through a little bit of what the lathe, what the tool looks like in the lathe, what the turrets. And we looked into some of the tools, some information on speeds and feeds.

I hope you found it helpful, walking through the part. If there's anything in your mind that you're thinking that was questionable, please feel free to ask. After this presentation, I'm going to put this presentation and the part into the file where you can get it. So you can practice on your own. Yeah.

[APPLAUSE]

Thank you.

