

**ANDREW GREVSTAD:** Hi, guys. Thanks for coming to this presentation. My name's Andy. I'm with Tormach. I have a mechanical engineering background.

And what I'd like to talk about today is a prototype machining for product designers using the Autodesk 360 product. Who's familiar with Tormach and what we do? A few of you? Some guys are brand new?

We do affordable CNC machines-- mills and lathes, three-axis and four-axis machines. And it ends up being a nice pair with the Autodesk Fusion 360 product because that's also a really affordable and very capable CAD/CAM package. It's really a neat time to be interested in this stuff and using these tools for product design because this type of technology is really a new level of affordability that just has not been there before.

Who's a Fusion user? OK, good. Who wants to be a Fusion user? A couple people learning about it? All right.

So I think here's how this is going to go. I guess we've got until about 5:00 to speak. I don't know if it will go entirely that long. I'm going to present a couple case studies that have been done with actually the tools that we've discussed-- our mill in particular and affordable CAD/CAM-- and everything that could be done with the new Fusion 360 products, though some of the projects go back a few years before that was available.

And talk about some of what I consider the important CNC milling concepts for product designers and people who are new to using this tool for project design. If you're not coming from a traditional manufacturing background, you may not know about things like tool selection and workholding, and where do you even get started with that stuff. So that's really what I want to focus on.

We're going to run through in that context a little kind of show and tell on a project we set up in Fusion 360 to make this electronic enclosure case for an Arduino microcontroller. It's a three-axis mill project made from aluminium.

And then what I'd like to invite everybody to do is, when the exhibit hall is opened back up at 6:00, to come visit us down at the exhibit halls. We'll be running this demo live. We'll be happy to answer any questions you have about how the project was set up, how the machine tool is run, how we set up the machine tool to coordinate with the project files in Autodesk, and just

any questions you have about, how do I use these tools to get started in prototype designing with CNC tools? Does that sound like a plan? Good.

Here's kind of what I'm hoping you guys will get out of this-- the advantages, kind of learn by example, some other successful product design, product development case studies, so you can see all the different things you can do with CNC technology. Talk about how the CAD/CAM workflow works for designing and machining, how it iterates a little bit from design.

Then we look at specially designed for manufacturing. Even before we start laying tools paths down in the part, start thinking a little bit about, here's my design. How am I going to make it? How am I going to hold it in the machine, et cetera. Because that stuff really has to start kind of at the very concept phase of what you're doing.

Again, we'll touch on some of your basic CNC concepts for those who are brand new to how a CNC machine works. And then we'll go through some simple tool path strategies that are available in Autodesk Fusion 360.

So this is our PCNC 440 machine. This is the smallest machine we make. You can see that machine downstairs in action.

It's about 450 pounds, about the size of a table saw, basically. As far as footprint, it's a four-axis capable CNC mill that starts at under \$5,000 base price. The machine you're going to see downstairs with the options installed is probably around \$7,000 or \$8,000.

So really, if you know anything about these tools, really kind of a new level of affordability. And what we try and do at Tormach is affordable, accessible. So this is a machine that's meant to be at the ready for product designers, for prototypers, for bootstrap manufacturing. We want it to be a machine that's simple enough to use, that's simple enough and safe enough to use, that somebody could walk up to it with very limited exposure to CNC and be successful with this product relatively quickly.

It's accessible. It's a machine that you can put in your garage, in your bedroom. I've had people put it in a New York City apartment.

And it makes real parts. So there are a lot of interest in desktop 3D printers and desktop routers and stuff. With this machine, you can cut any material you want-- aluminum, steel, stainless, granite, composite. Anything you can cut with a rotary tool, you can cut with this

machine.

It's 3/4 horsepower spindle, dovetail way construction. It'll repeat plus or minus 1/1,000 inch. And again, all that starts at under \$5,000.

We also make two other mills. There's a 770 mill, which is about twice that size. And then our PCNC 1100 mill, which goes-- it's about the size of a manual Bridgeport machine. If anybody's seen a manual machine from high school shop class or maybe university shop class, the PCNC 1100 is a CNC machine that's the size of a Bridgeport.

All those machines run on single phase power. It could be installed in your garage, or anywhere you want. And we also have a lathe in a similar scale.

So the advantages of in-house CNC machining and the projects I'm going to show you-- the 440's a brand new machine for us. These projects were, for the majority, made on the 770 or the 1100 machine. But certainly, a lot of them could be transitioned to the small machine as well.

And I'm going to talk about a lot of the different advantages through examples. One of the big advantages of CNC for prototyping and mechanical design is functional prototypes. You're able to use the real materials, the same materials you might make production parts out of.

It's that whole idea of function versus form. What are you really after? 3D printers are great at kind of these very sophisticated form designs. But when you're looking for mechanical integrity, structural integrity, anything like that, you're going to want to use real materials-- steel, aluminum, stainless, titanium, engineering plastics, composites, wood.

The example you're seeing there, I think, is a really nice hybrid of the two technologies. You see a 3D printed cowling. And then the mechanical and the watertight parts are milled out of, I believe, aluminum in this case. That's a prototype for an underwater UAV submarine machined on our Tormach 1100.

And then the round part, obviously, was made on a lathe. Any time you see round geometry like that, it's going to be made on a lathe. So that's one example-- impellers, things like that.

Another example of functional prototypes, mechanical assemblies. There's a lot of parts in the world that need to withstand some type of force loading or something like that. So you've got frames, structural components, gears, cogs, pulleys, threaded connections.

CNC mills can thread. They can both internal and external threads. Shafts and rotors, bearing blocks, motor mounts-- anything that needs any type of mechanical tolerancing. Heat sinks is another one that I see a lot of, and valve bodies. All this stuff can be made with a CNC mill. And all of this stuff, as a product design, needs to be designed and iterated on and prototyped.

The example here is a fairly complex mechanical assembly from a humanoid robot made by a laboratory at Drexel University. Actually, that's the leg. They've actually made the entire robot out of milled components and shared all the files on the internet. And it's a really cool project.

And what they've actually been able to do with a lot of the new collaboration online sharing tools is share that with other universities. So there's a group at Virginia Tech which downloads the same files, has the same machines and the same software, and can create the same robot. And basically, both do research in concert with the same robot and the same test rigs and stuff.

Really cool. The robot's, say, about yea tall. He's been on *Good Morning America*. He's really neat.

And then a third class of functional prototypes are important because a lot of the new additive technologies and materials are not characterized adequately for design standards. Certain optical properties, certain mechanical, structural, layered composites have directional loading characteristics. This is a microfluidics device made again at a university in Toronto that has some specific optical qualities that they needed a milled material so they could use a bulk clear plastic out of to make it first.

And you can get an idea of the size of that part. That's a nickel there. So you can make really small, detailed features with CNC milling technique. Here's another example where I've seen CNC milled prototypes really flourish. This is what I call product quality prototypes.

So this is a really neat story. The New York Toy Fair is the biggest toy event in the world for toy manufacturers. Does anybody do any of that or been to that for that? OK.

So this was a new product concept from a company called Marshmallow Fun. It's a marshmallow blaster. It's a marshmallow gun.

And what their design challenge was is to go from concept to product quality prototype in, I believe, 60 to 70 days. So this was a product they conceptualized in November and then were

able to show in January, the following January-- really compressed the design cycle. And they did that through a couple of neat CNC techniques.

One is the molds were urethane molds made out of aluminum for the soft plastic parts. And then the rigid plastic parts, like the handle there, were milled directly on the machine. And those two processes together were able to create a prototype that looked like something they could then market to toy buyers before they had to invest in any production tooling at all. So a really great method to create something, be ready to take some orders, and then ramp up the production afterwards.

So if you've got those kind of time sensitive timelines, having a CNC machine at the ready that you can just walk up to, start milling some stuff, snapping some things together instead of having to go out through contract, and back, and contract, and back, especially if you've ever tried to get any prototyping work done with a mill. With any milled prototype work done, sometimes if you're only getting one unit, you're going to pay through the teeth to get it quick.

Or you're going to wait, and wait, and wait in line until he's got some downtime on that machine to make it for you. And it may come back wrong. And then you've lost those six, eight weeks, however long it took to make it.

One of the other things you can do is-- maybe you're doing something like a Kickstarter approach, or really want to make a few products, sell them, and see what the response is-- you can bootstrap this with a CNC machine. Because the same techniques you use to prototype this machine, you can just run them again and start creating products that's ready to sell.

This is a neat case study from the manufacturing tooling world. This was a product that a designer in New Zealand created. It's a yogurt cup that you don't need a spoon to eat the yogurt with. So it actually crinkles and crushes, and the yogurt comes out. It's kind of a neat design. Ended up licensing that technology to Danon and a number of the other large yogurt manufacturers.

But what he had to do before he could even get to that point is prove to them that it was worth the investment in production tooling. If anybody sees that middle picture there, you can imagine how much that costs. It's a really expensive part.

What he was able to do is, on our PCNC 1100, go through a number of design iterations. I

counted on that table-- I think there's 22 or 23 different molds, all different, that he went through to refine this process before he got to something that he was comfortable with to invest in a really expensive piece of tooling that you don't want to get wrong.

You can also see the same approach for things like molds for vacuum forming material, vacuum forming process, or layups for carbon fibers, or anything like that. So don't just think about things that are milled and made. You can think about things like tooling.

You can think about things like jigs, and fixtures, and test brackets, and sensor holders, and all these sorts of things that may help you further along in your product design. And it's really having a affordable CNC just sitting there in the shop that you can take an afternoon and make something. Really is an excellent tool.

One final example is customization and personalisation. These are sunglasses. This company is called Individual Eyewear. And they make eyeglasses for people who do not fit into your mass-marketed eyewear. So think hockey players and people like that.

What he's able to do with parametric CAD is it has his base design and stretch things, shrink things. The tool paths will adjust. Make it for a really personalized product with very little effort, regenerating the manufacturing and the instructions again.

Some interesting places I've seen this done-- serialization. Industries like the natural gas and oil have requirements for parts to be serialized and stuff. So you've got a lot of parts that you make the same. And then you need to put a new serial number on each one of those parts. And then again, different sizes or different slightly stretches and modifications of things. That's another good example of a material that's not easily 3D printed, that kind of tortoise shell acetate plastic, which is really attractive and desirable.

Oh, I lied. I have one more. It's product variance and on-demand manufacturing. And these are spinal implants, both titanium screws and engineered plastic spacers and stuff. And you imagine people come in all sorts of different sizes, and they need all sorts of different spinal kits.

And so what they're able to do, as the company called Eisertech in San Diego-- and what they're able to do is essentially, as the kits are ordered by the doctors, make the parts to assemble the kit together. So they can reduce overhead. They don't have a whole warehouse full of inventory. And they can really make the products as they sell them. So it works really

nice for kind of especially niche manufacturing application like that.

So those are some examples of some of the things that just to get your mind going, what you may be able to do with CAD/CAM and CNC technology. Talk about the workflow-- and what I really want to focus a lot on today is design for manufacturing. Who's designed parts that have been machined before?

So a few of you guys know where this is coming from. And maybe you've had some iterations with a machinist or an estimator or something. And he says, well, this is going to cost a lot of money because of this. And he may or may not tell you why exactly. Depending on how good your estimator is, he may or may not tell you why exactly that part costs a lot of money.

So most of your estimators and your machine shop are coming from a machinist background, obviously. And here's how they think. Thinking about minimum setups-- how many times does my operator have to handle that part to make it?

How many times does he have to flip it? How many different machines does he have to use to make it? And minimum tools-- how many different tool changes? All this stuff adds into the cost.

And then the one thing, when you're looking at brand new designers, is don't design things that can't be machined. I think there's a few machinists in the crowd here. And yeah, right, guys?

So I think, as a product designer, especially if you're going to be designing machine parts using CNC tools, having an understanding of what you can and cannot do with a CNC machine gives you some neat perspective on designing, especially if you're trying to design to a price point or a price target.

And then they also think about sequence of operations. What do I mean about sequence of operations? I'm going to pass around some of these parts starting out. Here you go. I guess I'll start passing them.

**AUDIENCE:** --online resources and those types of things?

**ANDREW GREVSTAD:** You know, there is some good stuff. I may have some of the guys in the crowd say what they enjoy. There should be more. There really should.

We've got some good information starting on our website, but there's some other stuff. John over here has a YouTube channel in which he talks a lot about this stuff. Stand up, John. And he's giving a presentation on Thursday, right, about the lathe work?

**JOHN:** Wednesday.

**ANDREW GREVSTAD:** Wednesday, about lathe work. But sequence of operations, so how are we going to make this part? Start with the block. And you've got to think about this. And this particular part, this is for a camera. And there's actually one, two, three, four, five. And if I'm being honest, there's really six operations, I think, in this part.

I don't have my laser pointer handy. But obviously, your first part to machine-- all those little four discrete pieces in the top of that block, I call that a flip and deck method. You might have some other ways you describe that.

Flip that over. Then we're making that. Then there's some hole drilling on the end there. And then those final slots are actually done with a slitting saw at the end, too.

So there's actually a lot of different setups that this part has to go through just to make it. So it's not an easy part. And it's a part that you might actually be surprised how much it costs if you got the part quoted.

At the end of the day, this all starts with workholding. And I think that's somewhere, when I'm talking with guys who are primarily product designers, don't have a whole lot of experience and exposure. Just like, what goes into CNC workholding? So I want to just give a couple little examples and show you how something might be made, and then try to relate it into a CAM project.

So workholding, flip and deck-- this is by far the easiest way to make something. If your part can fit into this type of workflow, it's great. The machinist vises are just designed for making parts this way.

So you're going to start. And then those parts I'm passing around right now were made with this strategy. So we start with that piece of aluminum. That's about a \$2 piece of aluminum, maybe. And then we mill most of it out on the top.

And we're able to flip this part over because we've got parallel sides that will fit nicely in the vise when we clamp it on the second side. And then what I call the deck and you take off all

the back side of that, and you end up with four discrete parts. A lot of your parts made in a machine shop are made this way.

The advantages-- just going back. The advantages of this are you can walk right up to the machine. You can crank down your vise. Your vise is presumably already dialed in and straight and parallel to the motion of the machine. And you're ready to go. So there's no extra ways to think about, how am I going to bolt something onto the table or et cetera like that.

The second technique I'll point out is picture frame. And you can see kind of in the bottom of that picture, they're cut out like that. It's a really common way to make complicated parts. And what ends up happening is the parts are cut off except for some tabs. And Fusion has a really neat way to leave those tabs in there.

But the parts are cut out except for a few tabs, just to hold them in the frame. Then you're going to remove those tabs with a secondary operation, either a saw a grinding wheel or something like that. And then what you need to do is-- or a secondary finishing operation. That's what's going on in this case where those parts are then bolted onto a fixture plate and traced around the outside to remove the tabs.

He's actually stacking those parts, I think, about seven or eight high and going around that way. So he's really trying to economize. You can make a bunch of parts in one big setup there.

So this is a great example where somebody's designed a part. This is a throttle handle for an aftermarket cruise control for motorcycles. And he went through the whole product design and development phase making one part, and then was able to tool this whole thing up so he could make a bunch of them at once on the same machine.

And then I touched on it briefly-- fixture plates. And fixtures are great on a lot of parts, especially for large parts and thin parts that need some support from the bottom. The problem with fixture plates is they have to be manufactured and engineered and designed prior to making the parts that you want to make. So there is some extra effort that goes into making those things.

One of the things you have to think about with fixture plates is, how are you going to hold the things to it? You can use table clamps. But then if you want to machine completely around the outside of the part, you've got a clamp in the way. So you either have to pause the program, reposition clamps.

Another way that I might encourage you to think about is, are there holes in my part that I can use for fixturing these things that are part of the design already? If not, where can I add things, holes and stuff like that, for fixturing purposes that don't interfere with the function of the design?

Here's another example down at the bottom here-- a bunch of parts lined up on a fixture plate that's actually held in a vise. So you can use some of these-- so those are really tiny parts. But you can use some of these techniques in concert with each other.

Fusion allows you to obviously model the fixture plates and the vises, the workholding, all of that. It's a really neat tool. And you can simulate all around that to make sure you're not colliding or hitting any of that stuff.

And a final technique that some people may not be familiar with-- how do you hold round things? They actually make what they call collet fixtures for a lot of common parts-- round, square, and hex parts. These things can be clamped in a vise, bolted down on a table.

That's a trumpet mouth piece where he has to do some end machining on it. It's mostly turned, but then he does end machining on it that customizes the bell to specific embouchure of the player.

And realize there is a way to hold most anything in the tool. You just have to think about it. And having seen some pictures and some workholding techniques, you can start thinking about, how would I hold it so it's rigid, so it's simple, so I don't have to flip it any more than I don't want to? And if I wanted to make another one, that the whole thing is repeatable?

OK, so now, we're going to jump into a little demo with Fusion where we'll speak through the process with the Arduino case. This file is based on an open source project that was put out on a Creative Commons license by SparkFun. So this is what you call a 2.5D machining. So everything can be made from the top. It's all sorts of what you'd call terraced levels and stuff. It's a really easy project to get started with with CNC machining.

And we make this out of aluminum. I passed all the parts around. And did they all make it back? A few of them did, at least.

So we started out with the block here. And our first operation, we actually hold it in the vise. My jaws go like this, like this. And we're going to trace all the way around. And this is actually the

back of this part. And I'll show you how that tool path's looking in Fusion. Give me a minute to switch over here.

OK, so this is the Fusion interface. Again, some people are familiar, have been playing around with Fusion a little bit. Anybody played around with the CAM module yet? OK, OK. So you guys have--

So when you loaded it, there's a model. And then by clicking down on this, you can get into some different rendering animation. And CAM is kind of the manufacturing area that we want to get into. OK, and pointing that out, I can get to my tree. And what we've done is we've set up a number of options.

And what we want to do with this part is cut out. And because this is kind of a prototyping operation, one of the goals with this is we wanted to use the fewest number of tools as possible. So we milled the majority out of this with a 3/8 inch end mill. Not necessarily the most efficient way to do it. If you're looking for efficiency, you'd use larger diameter tools.

But 3/8 end mill was a good choice for this because when you look at the fillets here and the radiuses, these are all 3/8. And that's really, when you're talking about design for machining and how do you reduce cost in machining designs, look at things like inside corners. Because those radiuses, you can only cut with CNC operations round corners.

If you need to use a little tiny tool to get in there, it's going to take a lot more time to cut it out. So--

**AUDIENCE:** Your filling should be for your drill bit?

**ANDREW GREVSTAD:** Yeah, yeah. Yeah, your radius or your drill bit, is that's really-- or you could use a tool that's smaller than that. But that dictates the maximum sized tool you can use, to cut the feature all the way to shape.

Now, you can use a 2-step process, and we actually do a roughing and finishing profile on this. And I'll show you these here and talk about the reasons why you'd do that. And you can use a large tool to rough, and then a smaller tool to come in and do the finish and the detail work.

But if you had to cut this whole thing out of like an 1/8 inch tool, it would take forever. So you want to try and maximize. Because you don't want to be spending all day doing it. So that's one thing with kind of design for machining if you're able to think about that ahead of time.

One of the things I do like about Fusion as a parametric CAD/CAM tool is integrated CAD/CAM is I can drop this part. And then if I want to make that change later on, I say, oh gosh. I've got 3 in my corners here. These are all 3/8. I need a 3/8 tool to make all these inside corners here.

But then for whatever reason, these were a quarter inch or something like that. And then it's like, well, do they have to be quarter inch? Could I open those things up so I can make it with the same tool?

You don't have the same problem with outside corners. You can make square outside corners. It's inside pocket corners is where you run into a lot of issues. And sometimes, people don't think about that if you haven't thought about how the machine actually makes the parts.

So what we've done is we've set this project up for two ops. So op one, I'm putting my block in. And let me see if I can get my stock size going here so you can see-- oops. So I've actually-- and you can model your stock here. And so that's my stock.

And actually with op one, let me flip it around so you can see kind of the orientation of where we're at. So if the top of my screen is the back of the milling machine and the front is-- actually, I've got that wrong. Hang on. It's right-- there it is.

So that's op one. So I'm going to put this block in the vise. One thing about these blocks, if you've-- this is extruded aluminum bars. It's got two really good edges and then two saw-cut edges. You want to use your good edges to hold in the vise.

So we're clamping this one like this, and we're milling out the back side, as you can see. And I'll run the simulation of the tool paths. So play along with that. We'll open that up.

OK, so this is a roughing tool pass. We're using a 3/8 tool to go around the corner, going around the outside to kind of cut this part to shape here like this. Let me just crank her up.

We're about to select those faces in Fusion. And if I adjust those faces-- if I decided I want to make this thing wider, I make an adjustment in the model. And the tool paths will expand out as I need it. It's a really neat feature. Let's bring it up a little bit more.

All right, so that's what happens there. And now here-- oh, it's still going on that. This is what they call the 2D adaptive, is the strategy. So there's a couple little parts on the end where it grabs and gets it. That pass right there is the one I wanted to show you. Hang on. OK.

That second pass there is a finish pass. It's a 2D contour finish pass. And the idea with the finish pass is I'm taking a relatively light cut-- you guys may or may not know about finishing-- that minimizes effective tool deflection.

So with a roughing pass, I'm trying to remove the material as fast as possible. My tool, though it's carbide, is still relatively thin diameter. It's going to flex a little bit. So with a finish pass, I lighten the loads on the cutter. It's going to cut more truer to shape of what I intended. So that's one of the main reasons why you do a finish pass.

Let's see here. I may miss it, again, huh?

**AUDIENCE:** Andrew, you can just click on the green bar at the bottom.

**ANDREW** Oh, yep. OK.

**GREVSTAD:**

**AUDIENCE:** It accelerates it.

**ANDREW** Right--

**GREVSTAD:**

**AUDIENCE:** Yeah, just the bottom. Right there. A little further down.

**ANDREW** OK, right there?

**GREVSTAD:**

**AUDIENCE:** Click there.

**ANDREW** Oh, I learned something there, huh?

**GREVSTAD:**

**AUDIENCE:** There's a machine type for each operation as well as tooling and--

**ANDREW** Yeah, that's cool. OK, I learned something there. OK, that's your finish pass right there, I believe. But in the final pass, we're going to do with that same tool as the facing to make it flat here. And when I pass this around, you might have seen the pattern on there. The pattern's a little bit different than we have in the simulation, but--

In the final pass, there was a Chamfer tool, just an edge break. So we selected our surface

contour there. We're using the Chamfer tool, which is a 45 degree tool like that. And that's op one, OK? So that's gotten us to this part.

Now, what we do is we flip it over to do the second op, where a lot of the material removal happens. I want to talk about just a couple things in that second op. And we'll simulate here. There we go. We'll turn you up.

OK, so second op, we're flipping over. And when you run the thing, you'll see the tool goes from the bottom. So we'll flip the part around here virtually and run it.

And this is the decking thing that I talked about on the vise. So here, we flipped it around to this orientation and we're going to take off all this top deck here. And it'll crawl along.

**AUDIENCE:** Did you program that stock in? Or did it see from on one what was left?

**ANDREW** I don't believe it does the op. You have to import a second model in, right?

**GREVSTAD:**

**AUDIENCE:** You have to import a second model in.

**ANDREW** Yeah.

**GREVSTAD:**

**CURT CHAN:** [INAUDIBLE] is there's-- eventually, we'll to have that where the stock, just like with a lathe. You can do that today. But not knowing-- the cool thing, though, is-- and you can actually right-click on the op and say Save Stock Out. And then take that stock and compose that STL file.

**ANDREW** Oh, and then you can bring it back in?

**GREVSTAD:**

**CURT CHAN:** Then you can change the mesh of that STL refinement. Then import that back in to the model and you use that stock as your stock [INAUDIBLE] going forward.

**ANDREW** Yeah, that's--

**GREVSTAD:**

**CURT CHAN:** It is cool. Works on Windows today, doesn't work on Macs, because I've tried it. [INAUDIBLE]. So long term, we're going to add that little [INAUDIBLE].

**ANDREW** Yeah, that's cool. This is Curt Chan. He's the tech evangelist for the Fusion 360 product. And  
**GREVSTAD:** you're talking--

**CURT CHAN:** We have a talk tomorrow.

**ANDREW** Tomorrow.

**GREVSTAD:**

**CURT CHAN:** And Wednesday.

**ANDREW** Yeah, and you guys are doing hands-on lab things, so if people really want to play around  
**GREVSTAD:** with--

**CURT CHAN:** Lab is tomorrow morning, I believe, downstairs-- hands-on with [INAUDIBLE]. And then John and I will do the lathe tomorrow. And then Thursday is [INAUDIBLE].

**ANDREW** Yeah, so don't miss that stuff if you're looking for more details and stuff. That's a good thing to  
**GREVSTAD:** have, bringing that STL model. A lot of stuff fits into rectangle shapes. But sometimes, you get something that doesn't, and you want to know, how do I bring that thing in? It's nice to know how to do that.

So stepping back a little bit here, what we've done with this is we've done that decking thing. And then we've done a rough pocket using that 2D adaptive. And I love that adaptive strategy in Fusion because it does a constant chip engagement, makes it a really nice programming feature that you actually didn't have in a lot of CAM unless you were willing to spend about \$20,000 or \$25,000 a few years ago. So it's really nice to see it in Fusion.

And if you want to see what that strategy actually sounds like on a machine tool, that's what we're running downstairs. So I do encourage you to come at 6:00. So you've selected the cavity of that. And let's run it here.

One thing I wanted to point out with that is the difference in floor finish. And one of the reasons we actually do a floor finish pattern-- I don't know where the two parts go. I'll pass them around again. If you could pass those around again so you just can't tell the two parts up, you can see the difference. This is after the adaptive strategy, and this is with the second finishing operation here. And if cosmetics are important to you, you can see what the difference is.

So adaptive is a really nice way to remove material out of a cavity and stuff. But then going for

that final patterning in the bottom gave us the really nice kind of-- the machine marks that everybody wants.

So let's see if we can show the-- what's that?

**AUDIENCE:** They want the machine marks?

**ANDREW** Yeah, a lot of people want the machine marks.

**GREVSTAD:**

**AUDIENCE:** [INAUDIBLE].

**ANDREW** Well, we take a look at the parts, and you'll see what the-- so here's this adaptive thing where  
**GREVSTAD:** we actually move in here. And what's neat about the way the software works as it figures out all that math for you. And really, it's an incredibly complex tool path.

**AUDIENCE:** Is there a reason why you wanted like a small stepover in this example?

**ANDREW** Yeah, because we wanted to go full depth at cut on this part. Our spindle on the 440 machine  
**GREVSTAD:** is 600 watts [INAUDIBLE]. It's a small spindle on that machine. So we wanted to take a light chip load.

The horse power required for the cut is going to scale with a few things, and one of them is the depth of cut that you're taking. So we can take a light depth of cut. But in this case, we wanted to take a full depth of cut, full length depth of cut, to actually remove any tool marks on the wall finish as well. So that's why we did that.

**AUDIENCE:** So you went full depth but 10% the size of the diameter for the stepover? Or what was that stepover?

**ANDREW** Do you remember, Matt?

**GREVSTAD:**

**MATT:** It's like [INAUDIBLE].

**ANDREW** Yeah. And that one right there was that floor finishing, and it goes fast. We'll go right there.

**GREVSTAD:** There it is right there. You can see that, the pattern that it made there.

So it's a combination of kind of efficient material removal strategies, and then some finish passes where we wanted to have a little more control with what the tool path did to get the

desired features of the part. Then we finish it up with a very common spot and multi-stage drill sequence. So any time you're machining something, if you're going to bring a little thin twist-drill in there, you want to spot those holes first so the drill doesn't move around or skate on you.

So that's the sequence with the tool paths. One of the things I kind of wanted to preach about this whole thing is even before we even talk about feeds, speeds, depth of cut, widths of cut, any of that stuff, we're getting the geometry right first. So we actually set up all the processes, set up the ways we want to make the things, and make sure the shape looks the way that we want it. And then we'll go back into here and adjust our cutting parameters.

Your cutting parameters is going to be different depending on what tools you're using, what materials you're cutting, what the available machine horsepower is. But all that can be figured out in equations and stuff-- equations or lookup tables, et cetera. And HSM has some actually built-in with compatibles.

But the important thing is, get the geometry right first. Then you can go around and fiddle with the details.

**AUDIENCE:** How long does that part take?

**ANDREW GREVSTAD:** Cycle time on this part on our 440 machine is 35 minutes.

**AUDIENCE:** It's like end of the first job, ready for the second job.

**ANDREW GREVSTAD:** Yeah, or 40 minutes. And my guess is we're removing probably 90% of this block-- 85%, 90%.

**ANDREW GREVSTAD:** Let's go back to my PowerPoint here. So here.

So kind of to recap a few things here, evaluate the design for manufacturing. What tools am I going to use? And most importantly, how am I going to hold this thing? There's not one right answer to how to hold a part, but have a plan before you're going to do it.

And think about, is going to suit my needs? Is it going to be simple? Is it going to be rigid? Is it going to be repeatable? Can I set up vice stops? Can I set up table clamps? Can I set up other things? So if I want to make the part again, I can just load a part in and just hit go.

Figure out the sequence. How am I going to make it? Which operations? One, two-- maybe

I've got three, four, five different operations. But how many different strategies? And maybe there are-- like I talked about some specific reasons why you might machine something full depth.

**AUDIENCE:** Are you going to set up just [INAUDIBLE] when you're doing this?

**ANDREW** No. I'm with Tormach. I'm not with Autodesk.

**GREVSTAD:**

**AUDIENCE:** Oh.

**ANDREW** Yeah, so this is done on our Tormach machine. Yeah. And set up the project in CAM.

**GREVSTAD:** Orientation, work off-sets-- so we didn't talk about that. I should go back and show that real quick.

**AUDIENCE:** Which is the set of tools to use?

**ANDREW** The tools? What you can do is you can click into these fellas here and-- oh, I've got my

**GREVSTAD:** simulation open here. Hang on. Tools. How do I stop that thing? There is Escape, OK.

So I can bring it in there. I can load up these tools. I've set my geometry. This is the tool. And I select the tools over here. And you can select these tools.

And actually, Fusion's got some really neat tool libraries already built into it and stuff. So you can select tools. You can populate these tool libraries with your own cutting tools yourself. Here's pick which color do you have. What's the shape-- all that stuff. So all that's done in this thing.

And how you're going to use this in practice is you're going to set this up for the tools you have. And then save that stuff. And then you can use these really as quick templates. So once you've spent a little bit of time setting this up once, those tools are available to you and ready to go.

**AUDIENCE:** Sorry, does it automatically do the feed and speed depending on the material that you're dealing with?

**ANDREW** Say that again.

**GREVSTAD:**

**AUDIENCE:** Does it automatically do feed rate speed?

**ANDREW** There's some suggested feeds and speeds.

**GREVSTAD:**

**AUDIENCE:** On the material.

**ANDREW** Yeah.

**GREVSTAD:**

**CURT CHAN:** Not today.

**ANDREW** Not today.

**GREVSTAD:**

**CURT CHAN:** Long term, so when you take the material, filter out all of your tools [INAUDIBLE].

**ANDREW** HSM will do it. HSM has a built-in library, right?

**GREVSTAD:**

**AUDIENCE:** It has a built-in library. But it doesn't adjust the feed and speed based on the material.

**ANDREW** Yeah, mhm.

**GREVSTAD:**

**AUDIENCE:** [INAUDIBLE].

**ANDREW** Yeah, it's kind of a three-legged balance. You've got spindle horsepower, what the spindle can actually do. You've got just your material physics and stuff. And then you've got your cutting tool parameters, too

**GREVSTAD:** So there's lookup tables. There's guidelines. There's software lookup programs, that you can use all this stuff.

**AUDIENCE:** On just a regular feature. The materials makes a big difference.

**ANDREW** In reality-- and what you do is you run a couple test passes with the cutter and the material you want to cut. And you say, that's where I want to be. And then you can program those in there for your regular thing. So it's really kind of-- lookup tables and stuff will get you close. But then you can really dial it in based on what are you hearing with the cut and stuff.

**GREVSTAD:**

**AUDIENCE:** Questions?

**ANDREW**  
**GREVSTAD:** Yeah.

**AUDIENCE:** Once you enter that data in, can you save it as a profile to look up later?

**ANDREW**  
**GREVSTAD:** Mhm.

**AUDIENCE:** So you can set up your own personal library and your--

**ANDREW**  
**GREVSTAD:** Yeah. Curtis can explain more on that too.

**CURT CHAN:** [INAUDIBLE].

**ANDREW**  
**GREVSTAD:** Right there. OK, yeah.

**CURT CHAN:** Basically, [INAUDIBLE] libraries do that. You can save any library. You can create any [INAUDIBLE].

**AUDIENCE:** You could customize your own settings to your particular shop machine. And then go back to pull [INAUDIBLE].

**ANDREW**  
**GREVSTAD:** Yeah.

**CURT CHAN:** [INAUDIBLE].

**ANDREW**  
**GREVSTAD:** Or if you had two machines, you could set up a different profile for both machines and run that. Yeah, John?

**JOHN:** [INAUDIBLE]. And then I can just click on that tool in the bottom [INAUDIBLE].

**ANDREW**  
**GREVSTAD:** That's a real good timesaver, being able to save that stuff and just load it up, especially if you're just revisiting a project. Maybe it's one of these situations where you make the part once every few months or something like that. You can save that in there in a file and then revisit it. It's really a timesaver.

Was there another question over here? Nope. OK.

**AUDIENCE:** Is there a place online to get it later?

**ANDREW** This stuff that I presented?

**GREVSTAD:**

**AUDIENCE:** Knowing about the libraries being able to save tools.

**ANDREW** Yeah, again, if you're not familiar with Curt, he's done a lot of really neat YouTube videos on  
**GREVSTAD:** how to do this stuff. So as a testament to him, there's incredible YouTube resources that he's put together on how to do this stuff.

**AUDIENCE:** [INAUDIBLE].

**ANDREW** Yeah, actually, CAM, they've done-- this is a really nice interface. But there's things that I've  
**GREVSTAD:** got to look up, so I'll watch a YouTube video. Like, how do you do this?

And I've used a lot of different CAM systems. But like, do I do this in Fusion 360? I'll look at a YouTube video and figure it out. So they've really built in a lot of neat, kind of on-demand, support for this.

All right, so going back just finally to that prototype. Where is that here-- sorry, cheat sheet. Evaluate for manufacturing sequence, setting up the project in CAM, orientation work offsets. I guess that's where I was going to go with this, was you have to orient what you're doing in Fusion to what you actually want to do on the machine.

So I've got to set my origin. Where is my stock location? You're usually picking some point on the surface of the stock. A corner of the surface on the stock is a great place to pick.

For different reasons, you may pick the bottom of the vise job. I mean, that's what we actually did on this project where we're machining it, so when we do the flip that that part's right on the money. And we can use the same work offset for the flip. But at some point, you have to say, where is the physical location on this part that I can find, and then identify that in your program.

And then another question about how much modeling is needed, in Fusion currently, you can model fixtures. You can model tool holders. And you can model the work.

I know in HSM, you can bring in machine tool models and stuff like that. So you can actually add some more complexity. That stuff's all really cool.

Think about how much do you really need, because you can spend a lot of time getting all those things just-- they've made it really easy to do it. But still, there's a lot of rearrangement and stuff, whereas a simple three-axis part, really, that stock size is all you need for the majority of what you're doing.

Design tool paths-- like I mentioned, get the geometry correct first and then fill in those details based on what you want to do. And then simulate. And we'll take it a first step further on the machine.

You can actually do an air cut on the machine. If somebody doesn't know how to do that, I can show you how to do that on the machine. So we can actually run that tool path just above the part and kind of get that final eyeball to say, is this doing exactly what I think it's going to do?

So again, I'm going to invite everybody to come on down at 6 o'clock to see us run the part. Mention to us that you were in the class if you had any questions here. Matt and myself will be happy to answer those.

And check us out with Tormach. Feel free to connect with me personally on email, also our Facebook. Our YouTube channel is actually very active with machine tool set up techniques and all this stuff. So if you want to learn how to get started doing this stuff, check our channel out.

I mentioned the Autodesk channel. If you're learning how to set up projects in Fusion 360, really great resource. And Twitter, all that stuff. Any questions?

**AUDIENCE:** Projects?

**ANDREW GREVSTAD:** OK, so projects? Any projects? If somebody wants to talk a specific project downstairs, I love talking projects. So we can talk projects downstairs and how you might machine something. Thank you for the time. I appreciate it.

[APPLAUSE]