JEREMY MALAN: All right. I'm going to go out and begin. Hopefully, we have everyone in here that was planning to attend. My name is Jeremy Malan.

And I'm responsible for the support team in North America that supports all of the former Delcam products consisting of FeatureCAM, PowerMill, PowerShape, PowerInspect and ArtCAM. I worked for Delcam for 10 years before joining the Autodesk team in August of this year. Prior to that, I spent 11 years in machine shops as a machinist, CNC programmer, and a manager where I gained experience in production machining, job shops, aerospace, as well as moldmaking.

This class will teach you how to program using Autodesk's FeatureCAM software. It will focus on using FeatureCAM's automation to help you program your parts faster, more efficiently, and more consistently. FeatureCAM is a feature-based CAM system, which means that it uses common industry terminology during the programming process. This helps keep you from learning specialized terms just to program your parts.

I hope by the end of my class that you'll be able to see how automated CNC programming enables you to program parts quickly. I intend to show you how FeatureCAM can program a wide variety of machines while using recognizable language used extensively in the industry and how this approach is used consistently throughout the product range to ensure complete and consistent programs each and every time. I know many of you are new to FeatureCAM.

So before we begin, I'd like to make sure you know FeatureCAM can support all of your machines in the shop from two and three axis milling all the way up to five axis milling as well as turning, turn-mill, and Swiss. That includes a full support for multiple turrets, multiple spindles, and b axis. We also support wire EDM. FeatureCAM focuses on ease of use and speed of programming while trying to give you a balance of control required to complete most everyday part programs. If you're a production shop or a job shop which requires fast production or programming turnarounds, FeatureCAM may be a good fit for you.

In this session, I will show you some basic 2.5D milling parts, as well as a couple turn-mill parts, and a very basic five axis part.

If you're interested in going into more details about the full range of machines FeatureCAM supports, I would encourage you to sign up for Aaron Williford's session happening tomorrow

afternoon. If you'd like more information on his session, come see me at the end of this one.

So what do I mean by program using recognizable language used extensively in the industry? FeatureCAM uses common shop terms. Milling features are defined as features such as holes, grooves, pockets, bosses, sides, maybe faces. Turning features are defined as features such as turn, bore, hole, grooves.

Oftentimes, a feature will consist of multiple operations required to machine the feature. As we will see, FeatureCAM does most of this work for us ensuring that we do not forget any steps and ensuring we program consistently each and every time. In FeatureCAM, you draw or important your part. You identify the features, simulate, and generate NC code.

FeatureCAM automatically determines the rough and finish operations. It selects all of your tools, calculates your feeds and speeds, and determines the step overs and step downs for each tool. Let's take a look at this in action.

In this demo, I'll create a very common looking part you might see in any machine shop. The point of the demo is not the part, but rather to show you exactly how FeatureCAM works. I have to work magic to switch now from PowerPoint to FeatureCAM.

All right. So I'm going to go ahead, and I'm going to create a new part. And, again, FeatureCAM is wizard driven. So it's going to kind of guide me through the process as I do this.

First thing I do is I select the type of part I want to create. I want to do a milling set up. I say, OK. And it's going to ask me what's the dimensions of my stock.

So I get to define what my stock is going to look like. I want to do a block. I'm going to do 4 inches by 4 inches and 1 inch thick. I'm going to choose aluminum for my material, tell it whether or not I want to do three, four, or five axis programming, and then give my fixture ID, which is going to be my work offset for the machine.

The next thing I want to do is I want to align my G54 to the block of material here. I'm going to choose the lower left corner. And then we'll finish that.

I'm now presented with the stock properties. If I wanted to make any changes here, I could. In this case, I'm happy with what I created. I'll say, OK.

I'm going to go ahead and turn on some grid points. So I'm just going to make some basic geometry. So following these steps here, I've got my stock. The next step is geometry.

I'm going to create a circle. And I'm just going to eyeball one in the center here, a couple of circles over here. The next step is I want to create some lines that connect these circles.

Going down my steps, I can see I now need curves. I'm going to create a closed curve and use that to make a pocket feature. So I'll come down surfaces, solids. And I'm not working with those right now. So we go straight down to features.

I can say pocket. I've already got my curves selected. And I'm going to enter some basic dimensions for this part. In this case, I want a 1/2 inch depth. And I'm going to leave everything else alone for now.

I'm going to do a rough and a finish. And I'm just going to say Finish. Over here on the right, I can see that FeatureCAM's created a rough and finish operation.

It's picked my tool, my feeds, and my speeds. If I do a 3D simulation and slow that down a little bit, we can see we went down into the material. We rough it out, and then we do the finish.

Now, let's say after I've created this pocket, engineering comes to me. They say, I'm sorry, we've made a mistake. This top radius here, rather than being a 1/2 inch radius, that should have been a 1/4 inch.

I can come in here. I can change that to a 1/4 inch radius. As soon as I hit modify, I can see my tools change. And the way that I go about cutting that pocket changes.

We now rough this out with a 1/2 inch end mill. It comes in with a smaller tool that can clean out that corner. It does kind of a semi-finish in the corner, and then does a finish tool path.

I want to add to this a little bit more. I'm going to go in here. And I'm going to create a 0.0010 chamfer. If I was in the shop, I would probably really want to edge break this. And, again, we can see it automatically creates those for me.

We simulate it. It's already moved them. It knows my chamfer has to come after my rough and my finish. And let's put that in place.

Let's take this a little bit farther. Let's add a 7 degree draft angle. And let's do a 16 inch bottom radius. When I apply this, again, we're going to see all the tools change. We're going to see

the shape of the feature change here.

All of my feeds and speeds are calculated for me. For the last step on this part, I want to go in. And I want to add a couple of holes on here. I could draw my holes out if I wanted to. Or, I can just create those holes from dimensions.

So I'm going to go hole. I'm going to make a pattern of it. I'm just going to do a standard 1/4 inch diameter hole. I'm going to go through.

We can see here if I wanted to I can create a whole list of different types of whole features here. I'm going to make a rectangular pattern, put these 3 and 1/2 inches apart, and then offset that up. I can choose here whether I want to do a spot drill, a pilot drill, a drill, reem, bore.

In this case, I'm going to accept the defaults. I can see I've got my spot drill, my drill. I'm just going to go ahead and say Finish to that. And we'll simulate that one last time.

One of the most important things that I forgot to point out on this is in every step along the way here I'm generating my NC code as I'm going through and doing this. So in this case, I'm running a Haas post. I have the NC code right here. This is ready to go out to the machine.

Here we go. All right. So using the part that we just did as an example, in a traditional operation-based CAM system, I would have had to create multiple rough operations for my pocket. This would be to account for the taper and the radius. I would then have needed to create a finish operation for the pocket as well as a chamfer operation.

Next, I would have needed a spot drill operation and, last, a drill operation. For each of these operations, I would have had to select a type of operation, a boundary, type a tool path, and a tool. I would then have had to set the step over, step down, the feeds and speeds for that tool. And most of all, I would have had to have repeated that step for each one of the operations in that part.

Process-based CAM system uses predefined processes and applies them to your current part. This would have made this parts simpler than an operation-based CAM system. However a process-based system lacks the intelligence the FeatureCAM has, meaning that you would still need to analyze each process and verify that it will, in fact, work in this particular case.

What do I mean by FeatureCAM has intelligence? FeatureCAM does not use templates.

Instead, it uses a list of common parameters.

These parameters have defaults set out of the box to allow you to get up and running right away. But they're also completely customizable, so you can modify them based on your preferences. By using this approach, we're able to adapt to each feature ensuring the best practice every time.

We've learned about features and what it means to program using features. Next, we'll see how we're able to expand on this concept using feature recognition. Feature recognition is simply the ability of FeatureCAM to recognize these normal machining features from surface and solid models.

This coupled with FeatureCAM's automation allows for faster programming times than can be achieved with traditional CAM programs. There are two types of feature recognition available in FeatureCAM, automatic Feature recognition and interactive feature recognition. Automatic feature recognition is an option in FeatureCAM that allows a user to automatically recognize features from a solid model.

Automatic feature recognition it greatly reduces programming time. This allows the part to get to the machine faster, which reduces the overall cost of producing the part. Interactive feature recognition is FeatureCAM's tool that allows users to interactively recognize features from surfaces or solid models during the programming process.

Rather than just allowing FeatureCAM to fully recognize-- sorry-- and create features on its own. The user assists FeatureCAM in programming by defining the type of features they want to cut and the surfaces they want to make up these features. FeatureCAM then uses that information to create the feature including boundaries, tool selections, feeds and speeds, step over, step downs, et cetera, necessary for successful machining of the feature.

I am going to take the same part. And I'm going to go through, and I'm going to program this with both AFR and IFR to show the differences. So looking at this part here, I'm going to turn off my grid points.

I have my stock already created. I've got my set up there. And I have a model that I've imported that is the part. Again, going back to my steps, I can then skip down.

Because I'm using AFR, I don't need geometry. I don't need curves. I can go straight down here to AFR. AFR does have different options on how I want to create this.

In this case, I'm going to keep all the defaults. We can see it's identified a bunch of features. I could pick and choose which ones I want. And I'm going to accept all of them and just say Finish.

I program this or run the simulation here. I'm going to see FeatureCAM has programmed this part complete. I go over to my NC code. I can see that I have the code there. And this is ready to go out to the machine.

Now, if I had one of these parts, I'd probably be perfectly happy with this. If I'm going to do 1,000 of these parts, I may want to try to customize this and go in and program it exactly to my liking. What I want to point out here is look at the machining time. Currently, this took 21 minutes 50 seconds.

You guys remember that for me, because I am likely to forget. All right. So I'm going to go through, and I'm going to undo this. OK. So I've undone that. I have no operations here.

And I'm going to go through, and I'm going to run IFR and program this exactly how I want. First thing I'm going to do is I'm going to create a face feature. I'm going to go ahead and say Finish and Create More, which will keep me in the wizard here.

Next, I'd probably go in and create this pocket. I'm going to do that. I'm going to extract it with feature recognition. I'm going to use automatic recognition, which will go through, analyze, and find all the pocket features. Just got the one, I'm going to select that, say Finish.

The next step is I want to create my side features and, again, extract those with feature recognition. It finds all of my features. You'll notice it doesn't find this one. That's OK. So now I still need that one feature.

I'm going to come back in there. And we have several different options. I've got five different ways that I can find these features. So this time, I'm just going to select side surfaces, grab the surfaces I want, add that in.

And now, I've got that programmed as well. The last step I need to do is I need to create all my whole features. I'm going to do a whole, extract of feature recognition. I'm going to find all of those, including my counter-bores, select those. And we'll finish this part.

Run that simulation again. I can see all my feeds and speeds have been selected. My tools

have been selected for me. And I've got this part cut. Only this time, it's cut exactly how I want it.

Now, you might notice there's one more change I want to make. My center drills, I've got center drilling going on up here, again here, and then down here. This is due to an option that I have set. So if I come over to my automatic ordering, I can see that I've got it set to higher operations first.

That means that all these drills are going to happen before these ones and these ones. If I just turn that off, everything gets reordered. Simulate that one more time. And we're going to see that now it's exactly how I want it, all of my spot drills happening at once.

The best part if I look at the details, by taking just a couple of minutes to program this exactly how I wanted, I cut that down from 21 minutes 50 seconds down to 12 minutes and 15. So we almost cut that run time in half.

The last thing that I want to do on this part is I want to verify on the machine that this is going to work. Before doing that, I need a vise. I need something holding onto this. I'm going to grab a Kurt vise. I'm going to add some parallels in there and import that and.

I get some messages, because my file is set to read-only. It brings that in. It positions my part. And now I can go through, and I can run my simulation on the machine verifying that this is, in fact, correct and this is going to cut right. It's important that I'm able to identify this now and identify I'm not going to smash into my vise before I get out there on the machine and use up that time that, you know, I'd like the machine to be running.

So as we have seen, FeatureCAM utilizes feature-based programming. Because feature-based programming uses a set of customizable parameters, the same concept can be applied to turning and turn-mill parts. FeatureCAM has the ability to recognize features from surfaces and solids in turn and turn-mill just as it does in milling.

This allows you to program your parts the same way every time, the same expected results, since you will be using the same smart settings every time you program. To demonstrate this, we're going to look at two different turn-mill parts. As I program these parts, notice how your turning features will be made the same way and give you the same expected results. Also, notice that when we program the milling features for use on a turn-mill application, those milling features will give you the same expected results that you've been seeing in your milling

examples.

I'm going to close a couple of these parts here. We're going to look at this one right here. Now, in this part, I've got two different set ups. I'm going to start here on the first set up. And we're going to turn this profile. We're going to face this, do the milling stuff.

And then on the backside, we're going to flip it over, face it, groove this. And then I'm going to break all the edges on it. so start, and let's set up one. I'm going to create a turning feature and do the face first.

It's going to find my outer diameter based off of my stock material. And, again, I will just finish that. For the next step, I want to create my turning. I'm going to do from a curve turning.

And I don't actually have my curve yet. So if I just hit this curve chaining, I can come in here. And I can pick what sections of this I want to cut. That's the section I want to do my turning on.

I'm going to do a rough and a finish. I'm going to say Finish and Create More. I've got all of my turning done. Then the next step is I'm going to mill the outside profile of this. And then last, I'll do the holes.

I'm Going to find a side feature. I'm going to extract this with feature recognition. And I'll just choose automatic. It finds that side there. If I'm happy with that, I select that. Say Finish.

The last part are the holes. So I'm going to go create my holes. And, again, I'll find all of those, finish that out. Let's do a machine simulation. And let's verify that this looks correct up to this point.

I've got my part in there. And I'm able to verify with all these drills that are sticking out, that they're not going to run into my jaws. They're not going to hit my chuck or sometimes even the shrouds of the machine. I can verify that right here in the software.

So up to this point, we're looking good. Let's go to our second set up. And I'm going to create the face feature. And then, like I said, I'll do this groove, and then break all the edges.

I'm going to do a turning. I can increase my thickness here, 0.00220. I want to do a rough end to finish, and then do that groove feature. Again I don't have a curve for this feature yet. So I'm going to go in. I'm going to define that while I'm in here.

FeatureCAM is going to find the tool that it thinks is the best fit for me. We'll finish that out. And

then the last part is doing the chamfer around it.

So I'm going to do that as it turn-mill feature. And I'm going to find all of the chamfers automatically in this. And it's found them all. I want to modify.

Currently, it's set to 0.0050. So I'm going to change that, make that smaller, 0.0010. And then also I want to find the holes as well. So I'm going to uncheck to exclude the small diameter stuff.

And then we'll select all of those and finish. So I believe at this point I'm done. However, if I run, I'm just going to do a 3D simulation here. If I run the 3D simulation, we're going to see that there is one thing that I overlooked.

As I come around and I face this off, I can see that in my first operation when I did that side feature, I didn't cut down deep enough. Being able to simulate both the first and second set ups at once, I was able to identify that. I'm going to go into my side feature. And on my depth, I'll just add in 0.0030. Go a little bit deeper on that. If i rerun that simulation, we should now see that I've solved that issue.

I've got a small gouge there. And there we go. And that small gouge was detected, because of the rapid withdrawal move. There's a setting I could change in there to have it not wrap it away from the part like that.

There's our finished results. Now, I want to also show I went through, and I created set up one of this using IFR. If I delete all of those features, instead I'm going to create this with AFR just to show, again, that I can create these items and get a very similar result.

So I'm just going to find set up one, select everything except for this bore. It wants to do the small hole here as a bore. And where it is small, I don't need to do that.

Let's just go ahead and turn off our second setup. We just want to focus on that first one. And I can see that running AFR, again, I'm able to get almost the same results. The only difference is the order is just slightly different than the way I programmed it.

For the next part, we're going to do this part here. And this part is going to utilize a B axis turn-mill machine. It's going to get a bit more complex in there. We're going to do a lot more different types of features. But, again, we're going to be able to see that it's done in the same manner.

So if I open the actual part itself, I have here created. Let me show my solid. OK. And on this one currently, I've already got this programmed. I want to actually run this on the machine. So you can see how I'm cutting it. And then we're going to go through, and we're going to duplicate this result.

We've got all of our features there. We transfer to the sub-spindle. And we're facing the back of that off. At that point, FeatureCAM's going to rerun the first, the main, spindle to verify that while that part is in the sub-spindle, the head of the machine isn't going to come through and crash into the back of that part. Let's go ahead and program this from the start.

So I've got one here. The only operation I have is on my second setup, I left the face feature. I deleted out everything else from this part. We're going to go through this on set up one. And we're going to program everything.

So I want to start. I want to do all of my turning operations. I'm going to do a face from dimensions. Again, it's going to find my diameter. I'm going to change my thickness here. And I'm going to do just a finish pass.

Next, I want to turn my profile. I think I actually do have a curve in this case. I do. OK. We'll go ahead and use that existing curve. And on this one, I do want to do a rough and a finish.

Let's go ahead and finish that. And let's do a quick 3D simulation. We can verify that looks correct. In a machine simulation, I can go through. See that? Verify I'm not hitting against my jaws.

Now, I want to go in, and I want to create my milling features. I'm just going to hide my stock, get that out of the way, hide some of my features. I'm going to start by doing these face features here.

I'm going to do these as a face. And I'm going to extract these with feature recognition. I can easily identify which angle my tool is sitting by selecting the surface here. And then we'll add that in. Well, notice I've got 0.00850 of material thickness there to take off.

So I'm going to want to do a rough and a finish in this case. I'm happy with this 2 inch face mill. I will go ahead and accept that.

Now rather than recreating this, I'm going to go through, and I'm going to pattern it. I'm going

to create a patterned feature. I've already got that face selected. I'll just do two of these at 180 degrees and Finish.

The next step, while I'm there, I'm going to go through, and I'm going to create these pocket features. So from a curve, I'm going to do a pocket. I don't have to repick my angle. Because I already picked that on that previous one.

And it's going to go through, and it's going to find all the pockets that are at that orientation. Go ahead and select that one. And I will pattern that one as well.

I think the next approach I'll take is I'm going to come after these side features here that make these counter bores. So doing a side feature, I'm going to select a new surface now for my tool path orientation or my tool orientation. And if I select automatic recognition, we're going to see I don't get the results I want.

The reason for that, my drills come through. And they break into these surfaces. And it's confusing FeatureCAM on that. Again, no worry. I'll just select my side surfaces. I'll select the surfaces that I want to use.

I could do these one at a time. But I'm going to actually go through, and I'm going to create both of them at the same time. This allows me to, when I make any changes, I'm going to affect both of them. Where they're the same geometry, I want this to apply to all of it.

Now, once again, I will pattern those around the diameter. The next step I want to do is I want to create the side feature that goes around this hex shape. I'm going to do that from the face of the part-- there's a few different ways I could do it.

Assuming I don't have y axis travel or enough travel, I can't face these. And so, again, we'll go from the face of the part. I'm going to do a side feature. I'm going to tell it that it's along the z axis.

And if I pick automatic in this case, what I'm going to find is that it creates each one of these as individual features. While this would work, that's not the behavior I want in this case. So I'm going to use another method. I'm going to use this horizontal section.

And I'm going to go in here. And I'm going to select where I want the top of my feature to be, and then where I want to slice this model. And what this will do is it will slice my model at that location. And then it's going to project my geometry. It looks like I actually missed my slice

location.

It's going to project geometry here. And I can go through and create my curve based off of that geometry. So I have a lot of different ways that I can achieve what it is I'm looking for. Select where I want the bottom of that feature to be.

And then I'm going to make a couple of changes. In this case, there's not a lot of material to remove. I don't want to rough it. And it's going to do a semi-finish into finish. And I'm not going to ramp on that.

The next operation I have is I need to create all my pockets around the outside diameter, and then I need to do all my whole features. I'm going to create pockets, extract the feature recognition, choose my orientation. We'll go back to automatic and select that pocket.

Like we've seen, rather than recreating each one, I will just go through, and I will pattern that. I'm going to do six of them. And assuming I don't know that angle, I'll just say 360 divided by 6. I preview that. I can see that looks correct. And I'll finish that.

Before I go through and do the holes, I'm just going to look at this. I'm going to verify that everything looks correct here. What I'm going to find is I've got a gouge here.

This gouge has happened, because I have told it to minimize tool paths. And I've done these pockets here on the face before I did this side feature. So let's move these pockets up there as well, really easy for me to go through and fix that. And it's going to move my side feature up here. And now, when I simulate it, I get the results I'm after.

The last step are all the holes around the part. I'm going to go through. I'm going to create all the holes that are on the face. It picks up all three of those. I'm happy with that.

I will finish that, and then do all the holes around the diameter. I'm just going to select automatic. And it will go through, and it will find all of those holes.

And it's also finding the counter-bores. Now, in this case, I've already cut those. I just exclude holes with the diameter greater than 1/2 inch. I can control that and tell it to just find the holes.

I've got that. Let's do a final simulation. Actually before I do the machine simulation, let's look at this on a 3D simulation. And on the 3D simulation, it's going to go through. It's going to program this. And it's going to look like it's complete.

Speed that up a little bit. So everything looks good. But as I run the machine simulation, once again, we're going to find that there is one step that I have forgotten to do. Once this finishes everything on set up one, it's going to come over-- ran it too fast. And it faces the part on the second setup, which is our sub-spindle.

But I haven't put the part over there yet. So we need to get the part over there, so that we can do that face operation. I'm going to go in to turning. And FeatureCAM makes this really easy.

I just select Part Handling, tell it I want to transfer this, do a slug transfer over to the sub-spindle. Give it the distance I want to grab, how much I want to feed by. I've got control over how it's going to clamp, whether the spindles are going to orient or index or stop.

In this case, I'm going to select the default and just say Finish. And now, if we simulate this in the machine one more time, we can now see that we are cutting our entire part. And there we go. Yeah.

AUDIENCE: Just on that one, you do have the turret. And on the previous ones you had, the [INAUDIBLE] that was something that you load in [INAUDIBLE] what [INAUDIBLE] all your tools in [INAUDIBLE].

JEREMY MALAN: So that one, was a B axis machine. So it essentially has like a milling spindle for the upper turret instead of an actual turret on it. So it depends on which machine you're using on the configuration you're going to see there.

AUDIENCE: OK.

JEREMY MALAN: OK. So by now, I hope that you have a good understanding of how FeatureCAM works and what the benefits of FeatureCAM are. In addition to the standard FeatureCAM functionality, we also have an advanced programming interface that allows custom functionality and applications of the software. This can be used for many things.

Some examples of things that can be done with this are set up sheets, custom geometry creation, custom tool path features, setting parameters to specific features, and many more. In general, this functionality is often used to automate things even beyond what the software does natively. Each of the items I previously mentioned are examples of functionality that we have already done in the software. And they do ship with it.

While I don't have time to show everything, there is one more example I want to show. And

this is a five axis part. On this part, I'm going to create the part that we see up there with just a few easy clicks.

Much unlike switching from PowerPoint to FeatureCAM. OK. So on this part, I have a five axis part. The part is supposed to be coming from a cast here. So I've got a solid that represents my stock. And then I have my part program here.

What I'm looking to do here is I want to program each one of these pockets that have that dark green floors on them. So selecting one of the dark green surfaces, I'm going to use a keyboard shortcut that will bring up this select by color.

Pick all the dark green surfaces there. And then I've got a button here. And this button is going to go through when I hit that, and it's going to create a set up at each one of those orientations. And then it's going to identify those features.

What it's looking for specifically is pocket inside features. So it takes a second to calculate. It will go through. And it will create all of these. And shortly, we will see them in our operation list.

So I now have all of my features minus my holes. If I go ahead and run a machine simulation, I can look at this. And I can verify that everything looks correct.

So it's roughing each one of these. It's doing the finish passes on everything. And there's one thing once this completes that I want to show you that I'm going to go in and modify. While I have cut this whole part complete, when I was roughing all these pockets out, I'm roughing right to the finish depth of the pockets.

In some cases, that's fine. In this case, the surface finish on the floors of these pockets is important to me. And I want to make sure that I'm finishing those with a separate pass. So I'm going to go in, and I'm going to go into my machining attributes.

These are kind of the brains of FeatureCAM. This is how FeatureCAM automates everything for us. I'm going to go into my step overs. And on bottom finishing, I'm just going to tell it I want to finish the bottom.

It's going automatically modify every one of my pocket features all at once. And now when I simulate this, I'm going to see that it leaves material there.

And my finishing tool comes in and finishes the bottoms of all these pockets. So are you able

to very quickly and easily modify your entire program with just a couple of changes. There we go.

For the last step, I need to create my hole features. And as we've seen before, I'll just go in. I'll create a hole. I'll extract these with feature recognition.

I'll find all the holes along z. Now, these two here I've already done as pocket features. So I'm just going to select the ones I want, finish those, and then create all the holes that are around, non-z holes I guess. And select all of those, and Finish.

It's going to run a quick center line simulation. And I can see the NC code here for, in this case, an DMG machine. Soon as I finish, it's calculating. And there it is.

OK. So at the beginning of a class, I told you that I'd teach you these four things, learn how to automate CNC programming and how that's going to help you to program parts quickly and consistently, discover a broad range of programming capabilities from three axis mills to complex multi-axis milling and turning centers, learn how to quickly and efficiently program all of your parts, and learn how to program using recognizable language used extensively in the industry. I hope that I have done that and that you've found this class to be informative. On that note, I would love to know how I did.

I'd love to get your feedback. If you guys have the AU mobile app, you can do this right on the app. Otherwise, feel free to come speak to me after class. And let me know what I can do to improve for next year.

I also want to point out if you have any other questions about the software or any other Autodesk software to come see us at the answer bar. It's located outside of Hall C on level two and is open daily during the hours above. OK. Thank you. And any questions?

[APPLAUSE]