AARON WILLIFORD: Right, so welcome to Programming All of Your CNC Machines Using Autodesk FeatureCAM. This class is designed to teach you a little bit about the wide variety of machines that are available today. In today's machine shop, you could come across anything from three-axis mills, four-axis mills.

That index may be around x-axis or y-axis such as horizontal machine or even z-axis and the table-mounted z-axis indexers and onto complex, full five-axis machines. You can also come in contact with simple two-axis lathes, do some very complex, multi-tasking turn/mill centers complete with multiple turrets, B-axis heads, maybe sub spindles, that sort of thing as well as some high-production, small space Swiss machining centers. In many shops, you'll also find wire ETD machines.

So you'll see that FeatureCAM can handle all of these machine types and make it simple to program all of these machines. Hopefully, by the end of this 90-minute session, you'll have seen how FeatureCAM is able to program a wide range of CNC machines. You'll understand how feature-based CAM allows for consistent programming across all machine types. You'll easily be able to adapt your existing FeatureCAM programs for use on other machines that you may have in your shop. And you'll see how FeatureCAM makes optimizing your programs easy.

So before we get into FeatureCAM, a little bit about me. My name is Aaron Williford. I'm a Technical Consulting Manager for Autodesk. I look after support and training for all the products that used to be Delcam, so specifically PowerMill, FeatureCAM, PowerShape, PowerInspect, and ArtCAM for the North America region. I came to Autodesk through the Delcam acquisition having been at Delcam for four years.

Before I came to Delcam, I was an instructor of manufacturing technology at a local community college in Colorado. And before that, I worked in the shop for 10 years as a machinist, setup programmer, process designer. I pretty much did everything that there is to do in a machine shop. I used everything from three-, four-, and five-axis mills onto lathes and terminal machines, some with B-axis and sub spindles, and even did a little CNC grinding.

So let's take a look at the different types of CNC machines that are available so we've got a base understanding of what FeatureCAM is capable of. Three-axis mills are probably considered the simplest machines of today. Three-axis mills have three axes of movement as the name indicates, x-, y-, and z-axis. There are many manufacturers that offer three-axis mills. Three-axis mills can be found doing a variety of works including moldmaking, aerospace work, automotive work, and general machine work.

Moving on to a bit more complex machine, we have four-axis mills. These machines have four axes of movement, so your standard x-, y-, and z-axis. But they incorporate another rotary axis for a fourth axis. Some of these rotary axes can be positional, which we typically refer to as three plus one type movement, which essentially allows the rotary axis to simply rotate to a fixed location and machine with the other three axes. Others may be fully simultaneous, allowing for movement in all four axes at the same time.

So if you take a look at the pictures, the two pictures I've got there, the one on the upper right is a purpose-built four-axis machine and it's a horizontal machine. The one on the lower is a bolt-on style rotary indexer or fourth axis. It could be indexing around x- or y-axis.

So to add to the complexity of what may be considered already complex machines, we have five-axis machines. Five-axis machines, of course, allow for five degrees of freedom and movement in those five axes. So these axes are typically x, y, and z, just like three-axis machines. But they add two rotary axes that can be labeled differently depending on the machine configuration, which we'll talk about next.

It's important to note that these machines may be fully simultaneous, allowing for cutting movement in all five axes at the same time. Or they could be positioned on three plus two, which is a little less common now. Of course, three plus two, or positional, means that you will position the two rotary axes to a specific angular location and then cut with the other three axes.

So five-axis machines really have a whole host of benefits. One of those benefits is certainly reducing the number of setups required to fully machine a part which, in turn, creates reduced production time and ultimately more money. Five-axis machines come in three basic configurations. Table-Table machines, sometimes you'll hear them referred to as ANC stacked rotary tables since the typical access tags are A-axis for the trunnion movement and C-axis for the table-mounted rotary motion.

You've got Head-Table machines which typically have a rotary table for a C-axis. And the spindle is usually attached to a B-axis head. Then you've got Head-Head machines which have no rotary motion on the table. So both of the rotary motions are up in the head area or the spindle area.

So the axis tags that I defined in here are really just general. The machine manufacturer can determine what those axis tags are. So depending on the configuration, the machines can be purpose-built five-axis machines or they could be three-axis machines with bolt-on five-axis configurations. Some of these machines can also include basic turning capabilities where they can actually spin the table at a low RPM and come in contact with a turning tool to do some basic turning profiles and facing.

Turning centers, or lathes, as we typically call them, move in two axes, often simultaneous. But, really, that depends on the profile that you're trying to cut. The axes are typically x and z.

Lathes can cut a variety of parts. But typically they're going to be cylindrical in nature. Keep in mind though that even simple lathes can become more complex with the addition of multiple turrets, sub spindles, tail stocks, and maybe steady rests.

Turn/mill machines, as the name would indicate, really bring together both the milling world and the turning world, allowing for the milling functionality in a turning set up, if you will. Again, there are many machine configurations for turn/mill machines. But the one thing that really makes them a turn/mill machine is the addition of live tooling rather than just supporting those stationary stick-type tooling that we're used to seeing on lathes. These machines can become very complex with the addition of y-axis movement and B-axis heads, which really ultimately gives the machine five-axis capability.

Wire electrical discharge machining, typically called wire EDM, allows us to machine harder materials than what are machined using the traditional machining techniques. These machines can cut both inside and out, so outside profiles using a wire or an electric current. Along with cutting harder materials, wire EDM machines allow us to cut smaller radius cuts on profiles.

So now that we've got our manufacturing lesson out of the way for the day, let's move on to feature-based CAM. So I guess the big question is, what is a feature? And why do we want to use it in CAM? Or why should we use it in CAM?

The short answer to that is a feature is an entity that is used to describe an individual section of a part or a product. So every part is made up of at least one, typically several, features. So, for example, on the mill part that we have here, we could have a pocket, a boss, and maybe some holes. On the turning part that we see, or turn/mill part that we see on the other side, we could have a side feature, grooves, bores, and turn features.

So using features in CAM really allows you to program using those common shop terms that you're already familiar with like the ones that we just talked about, pockets, bosses, holes, those sorts of things. And, essentially, what you do is you define the end result of what you're trying to get. And then the software guides you in getting that result.

So to explain this a little bit better, I'm going to explain how FeatureCAM actually goes about using feature-based CAM. So in FeatureCAM, you're going to import a solid model. You're going to use feature recognition to find or identify the features. You'll simulate to verify that the result is what you want. And you'll generate your NC code. During that process, FeatureCAM has automatically created all of the operations necessary to fully machine the features, selected all of the proper tools from you from the tool crib that you have active, produce the feeds, speeds, step-overs, step-downs and generated the tool path.

If we compare that to some of the more, let's call them traditional CAM approaches, for example, process-based CAM, you would need to import the solid model, identify a feature, select the desired machining process, simulate to verify that the result is what you want, and then ultimately generate the NC code. But is that all? The answer is yes and no, really. You should go back and verify that you have all of the required operations, that you're using the correct size tools, and that you are using your desired feed, speed, step-overs, and step-downs.

If we take that a bit further and compare that to operation-based CAM, you would import your solid model, create a set up, select the type of operation, choose or create a boundary if a boundary is necessary-- it's not always going to be necessary-- choose the tool path that you want, choose the tool that you want to use, input your feed, speed, step-overs, and step-downs, simulate to verify that the result is what you want, and generate your NC code. I should note, though, that you have to go through that same process of choosing the operation, the boundary, the tool path, the tool, defining your feeds and speeds, step-overs, and step-downs for each subsequent operation necessary to fully machine that part.

So let's get into the fun stuff. We'll take a look at programming a part on each of the many different machine types that we mentioned before. We'll start with a three-axis mill. And this example will look at how feature-based CAM can speed up the programming time significantly.

We'll explore the difference between FeatureCAM to feature recognition types, automatic feature recognition and interactive feature recognition. And we'll take a look at where FeatureCAM gets its smart information or its intelligence from.

So let me switch over to FeatureCAM and we'll program a part. So what we're going to do is just import a solid model. We're going to start a milling setup. And FeatureCAM is wizard-based. So what that means is that FeatureCAM will try to guide you through the process of machining, of setting up, of whatever it is that you're trying to do in FeatureCAM, guide you through that process using wizard.

So in this case, we're going to use a wizard to establish our initial set up location and stock size. And then I've checked this box here to launch AFR after finish. So what that means is, after we've established all of the setup information and the stock size, we'll just jump right into recognizing features.

So we'll define our z-axis and just align it perpendicular to this surface here. And we'll rotate our x three times so that, if we're looking at this in a top view, this is how the part's actually going to sit-in the machine. We'll define our stock based on a simple block. And in this case, it's going to be a bounding block around the imported geometry.

I've added some extra material here including some material on the top for a face pass. We'll set our setup over here in the upper left corner on that hard jaw location. And we'll bring our z down 50,000. It's that extra material that we left on the top for a face pass.

We'll finish that. FeatureCAM jumps us right into automatic feature recognition. There's a whole host of options for automatic feature recognition that you can use to help determine what features you're going to get from automatic feature recognition. In this case, we'll use the defaults and we'll just continue through this.

So we're going to recognize along our z-axis of our only setup that we have in here. So FeatureCAM has found all of those features. All of these blue highlights that you see on here are features that FeatureCAM found. At this point, we could go through this list of everything that FeatureCAM found and maybe pick and choose what we actually want to keep. I'm happy with what FeatureCAM has found through feature recognition. So we'll just say Finish.

So if we take a look at this part in a 3D simulation, we'll see that we've more or less fully programmed this part. And if I select my NC code tab here right away, we can get and NC code and we can take this out to the machine.

Now, why did I say more or less fully programmed this part? Does anybody see anything missing? Let me show you an option that we've got in FeatureCAM here.

I'm going to select this solid as a part compare target. So this is the solid that we actually programmed from. And we'll just go into a viewing option here and show our part compare. See the blue? FeatureCAM didn't recognize that, on the tops of those features, we had a chamfer. No big deal.

The reason it's no big deal is because, in feature-based machining, we can just go in and add the chamfer to those features. So we can just add a 0625 chamfer to that one and a 0625 chamfer this one.

Now, I don't know if you guys picked up on it. But as soon as I hit Apply, FeatureCAM added the appropriate operation for us. It also rearranged our operations over here to make sense of the machining process, to make sure that we're machining this part in what may be considered the most efficient way, OK?

So what did FeatureCAM actually do? Well, what it did is it automatically ordered these operations based on automatic ordering options. So, currently, that's just set to minimize tool changes. So we're looking at all of the operations in here. And if we have an operation or we have two operations that are using the same tool, if we can safely machine both of those operations at the same time while the tool is there, we'll order those operations together.

OK, you've got a lot of options in there as far as how you want those operations to order. In addition, you could switch to manually ordering operations and manually place these operations where you want them, OK?

So what allows us to just make simple changes like that? And where did FeatureCAM find all of its information from? Well, first, we've defined a tool crib. And FeatureCAM comes with a pre-defined tools tool crib. And that's the one that I'm using in this case.

So FeatureCAM just pulled all of those tools that appropriately fit in the features that we're trying to cut from that tool crib. And, in addition, we've got a set of customizable configurations or attributes that you can go in and teach FeatureCAM or set up FeatureCAM how you want to actually machine. So all of those variables that you would normally define, here you define it one time and FeatureCAM just pulls from those same things every time. And you can have more than one set of these defined maybe depending on machine type or material type or something like that, OK?

So as we move through these examples, we're going to move through progressively programming on more complex machines, hopefully more complex parts as well. You'll see that, with FeatureCAM, the way that we program stays consistent, making it an easy transition from simple to complex. So featureCAM four-axis consists of the ability to program four-axis positional parts on x-rotary, y-rotary, or z-rotary fourth axis. In addition, you can program for Tombstone four-axis machines that index around either x-axis for your vertical mills or y-axis for your horizontal mills.

So in the next example, we'll program on a vertical mill indexing around the x-axis. This will be our cutter body example. Then we'll move on to show you how FeatureCAM handles machining on Tombstone. In the Tombstone example, we're going to machine the same part on all four sides of the tombstone. But I should note that FeatureCAM has the ability to support machining different parts on each side of the tombstone, If that's what you typically do in your shop.

So let me load up another part file here. So in this particular part file, we're starting with nothing programmed at the moment. So this is a generalized cutter body shape. And this is actually a part that we use in training customers.

So it's a great example of how we go about programming four-axis positional cutting and FeatureCAM. So we're starting from a pre-term stock shape. That's this light blue outline that we can see here, so just so you know we're starting there. I'm going to go ahead and hide the stock so that we can actually see more of what's going on in here.

So we're going to use interactive feature recognition in this case to find all of these features. So we're going to program this first side that actually happens to be along z-axis. So we're going to select a side feature and extract with feature recognition. And, in this case, we're going to just go along the setup z-axis.

We'll let FeatureCAM automatically find the features that it can along the set of z-axis. There it's found both of those features that we want it to. So we'll select them. And we'll just say Finish and Create More.

So moving on in a counter clockwise direction to the next side, we can see that we have essentially the same thing here. So we're going to select another side feature, Open Cut, Extract with Feature Recognition. But now, because this is not along the z-axis, we're going to specify somewhere around the index axis. We can either specify a specific angle or we can just pick a surface that we want to be normal to.

So we'll go Next. And we'll let automatic recognition find what it can. So in this case, it found this feature that we do want but not this one over here. Or it found this one that we don't want over here as well. So we're going to select the one that we want and say Finish, OK?

Same process as before. I don't have to reselect the surface. Feature CAM remembers the normal two from the last election.

So, in this case, we're actually going to use select side surfaces because it didn't find this one. And the reason that it didn't find that one is because we've purposely reversed the surfaces on this part, OK? That way we have to go in and program in a different way during our training classes. So we'll just select these surfaces here, add them into our list, and we'll switch the machining sides so that we're machining on the appropriate side and finish.

Moving around the part, we're just going to continue the same process for the remaining two sides here. So I'll go Automatic Recognition, find the feature that I want, finish, and jump back in. Then we'll select the side surfaces.

Now, this one we've got two pockets that are essentially the same, just located in different spots. That's fine. Since they are essentially the same but just located different, we can create them both at the same time. So we'll select all of those side surfaces, verify that we're machining on the correct side, and finish, OK?

One side left. Normal here, automatic, and we'll select the one that we want to keep and finish. And one last time, we'll select the side surfaces that make up this insert seat pocket, OK?

Are we done? Not quite. We still got the holes, right? So we'll start our new feature, wizard again. And we'll say hole, extract with feature recognition. This time we'll just say around the x-axis but automatically find all of the holes. FeatureCAM has found all the holes, we select them, and finish.

So notice over here, in my operations list, I've got these two little red exclamations. What that's telling me is that something is wrong here. If I actually try to run a simulation right now and generate a tool path, I'm not going to get it because there's something wrong.

Well, what is it? Now, we can see the stars right here. FeatureCAM doesn't know what tool to use to machine side six, which happens to be these pockets right here. No big deal.

We can tell FeatureCAM what we want to use. So I'm just going to use the same tool that we used on all the other pockets here and just select an override with that tool. Quick 3D simulation, and we fully machined that part.

So I'm going to load up a part that I've pre-programmed here. This is an all 3.5D part. But notice that we are actually cutting some features on this front face here as well as this left side and this right side. So this is a perfect candidate to be run on multiple faces on a tombstone machine. That way we can machine the whole part at one time and we can machine multiple of them at one time.

So what we'll do is leave this as it is right here. We don't have do anything special in this program or in this file. We'll open up a new file and do a tombstone fixture.

The first thing that we want to do is actually define our tombstone. So, in this case, we're going to index around the y-axis for a horizontal machine. I've predefined, in this case, the tombstone size. But you can set that to whatever sizes of tombstones you actually have. And then we'll load in some parts.

So I'm going to select that part that I've already got open behind this window and we'll just add it in here. When we do that, we're going to specify a few offsets. And these offsets actually come from where the setups are located or the setup, the primary setup, is located in the original part file.

So offset from our left edge, we're going to leave zero. We're going to offset from the top 14 and 3/4 inches. And we're actually going to pull out from the tombstone face four inches. If I go back to that part file, we'll see that the part is two inches thick and the fixture plate that's underneath, that is also two inches thick. So all we're doing is just pulling that out that four inches. OK?

We've loaded all of our parts in here. To get a good idea of what this actually looks, like I'm going to make some changes to simulation numbers here. So x is going to remain zero. Y is going to be the distance from the top of my tombstone to the bottom of my tombstone, which happens to be 27 inches. And z is also going to be zero. Those numbers don't play into NC code at all. They're just strictly for simulation purposes.

So we'll step into a machine simulation in this case, orient this so you can see what's happening. And what you'll see is that we-- wow, that's fast. Slow that down a little bit. FeatureCAM will automatically order the operations for all four parts that we've put in here to effectively use those tools using the automatic ordering options that we had set, which, if you remember, was just to leave the tool in the spindle as long as we can.

So there are machines that have the ability to do some simple turning by offsetting ahead with a stationary turning tool in it and essentially running the spindle at an appropriate RPM to mimic turning. These machines are typically referred to as turning heads machines. They can vary in size from really, really large, horizontal machines to small attachments that could be found on vertical or horizontal machines. Some of these attachments can even support multiple tools, kind of like that one that I show down there in the lower right-hand corner.

So we're going to take a look at an example of programming a part for a smaller vertical machine that has the attachment turning head on it that can support two turning tools in it. So I'm just going to close out these files and we'll load up this one here. So, again, a small part. But we are going to be running two turning heads on this.

And if we take a look at our machine simulation-- sorry about that. I don't have a tool for that. I'm going to go here, Select a Different Tool Crib.

So we take a look at the machine simulation, we'll just bump past some of these milling ones here, pop a hole in the center. So now you can see that we come in with our turning tool. We turn the profile. And then we come in and turn the inside profile.

We're going to replicate this. I'm going to go ahead and delete all of these features And we'll just start over.

So, for me, the first thing that I want to do is go ahead and put the hole in the center of this. We're going to select a new feature. We're going to say hole, not extracting it with feature recognition in this case.

The diameter of the hole that I want is going to be 16 millimeters. But I don't know the depth. So I'm going to select the top of my part and the bottom of my part. So that gives me a depth of 60 millimeters. I'm going to go through and then just finish that.

From there, I can actually jump right into the profile, the outside profile and the inside profile. To do this, we're actually using FeatureCAM's API. So I've already activated an add-in that ships with FeatureCAM that everybody has access to called Turning Heads. And so my green check shows that I've already activated that. To access the features that are in there, I'm going to start the new feature wizard, select a user tool path or a user-defined feature, select my turning head, and, in this case, it wants a curve.

So I actually already have my curve in there because I had pre-programmed the part before. But I could easily extract the curve using one of the curve wizard methods that we have available in here. So I'm going to select my OD curve, make sure that I'm on an OD tool path, turn on a finished tool path, and everything else I can leave as is here.

I do want to make sure that I'm using an appropriate tool. So I've got this 35-millimeter right head turn tool in my milling document. And in addition to that, I want to specify an end point for just my finishing tool path. I want my finishing tool path to retract to a specific location before it moves on to my next tool path. So I just pick my point out there.

And then I can go and do the same thing for the inside profile. Of course, we don't want our OD curve. We want our ID curve. We'll change the tool path type to ID and turn on a finish pass. We're also going to set the u-axis sign to negative so that FeatureCAM knows that we've switched which side the tool is pointing on, OK?

Red exclamations, FeatureCAN doesn't know what tool to use. If I actually walk through the rest of that wizard, it would guide me in the process of picking the tool. In this case, I'm just going to finish the wizard and come in and select the tool that I want. So this particular profile has a bit of an under-cut in it. So we're going to use a grooving tool to actually machine that. We'll select that same grooving tool for both the rough and the finish.

And then we want to define or let FeatureCAM know that there's already a hole in here. So we're going to set what's called a minimum radius boundary, no need to machine beyond this minimum radius in this case of 8 millimeters. I did that in the rough. It automatically comes over to the finish for me. And then, much like the outside profile, I want to select an end point for my finishing tool tool path.

So if we take a look at our tool mapping at this point, what we'll see is that we are using this dondre multi-head that allows for two tools to be in it. So we're using offset 22 and offset two, all with the same tool in the machine there, OK? So let's just step into our machine simulation again. We pop our hole in, we come in and turn the OD, and then we come in and turn the ID.

So as we work through our next example for our five axis machine, I want to point out that, up to now, we've been using the same feature types to program across all of these machines that we've programmed on so far. One thing that I didn't hit on yet is simultaneous three-axis machining.

The tool paths all show up in FeatureCAM under what's called surface milling. So as we program parts that have more complex surfaces, we have to begin to utilize these surface machining or surface milling feature types. But we're still going to maintain a consistent programming style. So, in this next example, we'll program a five-axis positional part utilizing some of those surface milling tool paths.

So this is the part that we're going to program. And you can see it's a pretty traditional part that we can easily program nearly complete on a five-axis machine. To make it a little bit more realistic, I'm actually going to go ahead and import a vise into this part file. So let's use a Kurt vise, add some parallels. Then we'll just drop it in here.

You get a couple little pop-up warnings because I've got my file set to B read only. So FeatureCAM wants to naturally save the file as it's importing this vise. That's just the way the devise import is written. No big deal there.

So we're holding this part in the vise just like this. But we've got three setups in here. And what tells FeatureCAM this is a vise is just this option here, using solid as a clamp. So we want to make sure that, since we're actually running this on a five-axis machine, these other setups are also using that same vise as the clamp. That way FeatureCAM is aware that there is more than just the part solid when it's machining from any of those setups.

So we're going to start with setup one as our primary setup. Let's rough some material off of here. We'll select our part solid. We'll go into surface milling. Sorry. We've got a whole lot of other surfaces in here that we don't want. So we'll select our part solid.

We're going to do a single operation and we'll just do a traditional z-level or waterline roughing strategy. We're going to use two different tools to do that. So we're going to select Multiple Tool Diameters and say let's go with a 3/4 inch tool and a 1/2 inch tool.

Our 3/4 inch tool, we want to use a 3/4 inch longer, high plus tool. And for our 1/2 inch tool, we actually want to come in with a ball nose tool. So we'll use a 1/2 ball nose, 0.625 radius ball nose, tool, OK?

In addition to that, we want to make sure that we're only machining down to the top of our vise right here. So I'm just going to set a z-end on this tool path to be at the top of my vise. Now, as far as the end point with my 1/2 inch tool, if I take a look at my tool, clearly I've got less amount sticking out of my holder than I did on my 3/4 inch tool.

So I can't get all the way down to the bottom. And, really, I'm not interested in getting all the way down to the bottom. I just want to come in and further machine this section out with that tool. So I'm going to set a different z-end here on that particular tool. We'll take a look at a quick 3D simulation. It's what we look like right now.

So let's come in and machine on this side here. I've created a setup that's at an angle right here. Now, of course, in the actual five-axis machine, all we have to do is define the one setup and we can machine on any side that we can get to, right? This is just telling FeatureCAM what the tool axis is going to be. So we're going to select our setup.

Same process as before, we'll select our part and add all of those surfaces in using a surface milling tool path. Now, in this case, we're going to do a parallel style tool path or sometimes referred to as raster. So we'll just step back and forth across this part. And we're actually going to use our same ball nose tool that we used, as our second roughing tool, inside of here, OK?

If we take a look at this tool path right now, we're going to notice a few things on here. I'll speed up the simulation just a little bit. So we're over-cutting this model by a fair bit, OK? Right now, this tool path doesn't really understand what the current model state or stock state is of this.

So we're going to define that. We're going to let FeatureCAN know that, hey, we've already roughed some material off of here. You don't need to take additional material where there isn't material really to be removed.

And we're going to do that in a couple of ways. We're going to define the stock model. So a stock model is really a way that you can reference previous items that you've already done, so maybe another solid that you have or an STL model of your current stock state. Or, in this case, we're going to use our operation results.

So we're just telling FeatureCAM, hey, we've already roughed this off with this surface mill one feature. Now, when you create this tool path, I want you to reference that. So we'll say stock model one at that level.

And just to make sure that we don't go down too far in the z-axis, because really we only need to go down to here, we're going to set a z-end on this tool path as well minus 1.75. So-- oops-- let me go back into that path and just show you the difference that we've made there. Significantly less tool path. Now, we're not over-cutting that model in areas that we've already removed material.

So we're pretty well roughed out. Let's go ahead and begin finishing by finishing just these three straight-walled sides. Since they're straight walls, we can use a 2.5D feature. So similar to the process that we have used so far, we're going to select the side feature and extract it with feature recognition. And, in this case, we'll use our side surfaces, make sure we're cutting on the right side, define our bottom down here on the bottom jaw, and then finish.

I do want to make sure that I'm using the same 3/4 inch tool in this case that I had before. It's not necessary. But that's what I want to use. So I'm going to use that same tool. And let me hide the stock for a minute here.

The next thing that we're going to do is actually cut this front profile here. And, again, we can see that we can cut that, especially in a five-axis machine, if we're looking at it this way, with a simple 2.5D tool path.

So we'll do that. We'll select our third set up that we've got over here to define our tool axis. This is coming out this way. Then we'll just select these surfaces that we want to machine.

So using a 2.5D tool path, that's an open cut side, extract it with feature recognition. We'll select our sides and pick the bottom of where we actually want this to go to. And in this case, I'm going to extend that a little bit. We'll just go 50,000 beyond that to make sure that we're fully machining below that surface. We use that same tool just to reduce the number of tool changes.

Now we can concentrate on finishing this inside here. So I'm actually going to select those surfaces that I want to finish. And using a surface milling tool path, we'll do some finishing. And, in this case, we'll do a steep and shallow finishing, OK?

And one last thing. Actually, you know what? Let me double check the tool that I'm using there just to make sure that we're using-- yep, a 1/2 inch ball mill. That should be perfect. One last thing that we want to do is finish this surface right here.

We'll do that again with the surface milling feature, single operation. And we'll do a parallel finishing strategy. The roughing strategy worked well. So we might as well finish it that way as well.

Same 1/2 inch ball nose tool that we were using before. Although, in this case, I am actually going to decrease my step-over a little bit. I want a little bit better surface finish than the 50,000s.

So what you'll notice here is that because we've set this vise as that, as a clamp, we actually avoid hitting that area with our finishing tool path. I didn't do anything special in there. FeatureCAM just knew that it couldn't machine there.

So we've machined everything we can on this part with the current setup. Of course, we would have to go to a different setup to machine the rest of this. Or maybe we know now that we need to add some material on the bottom so that it's sticking out farther or whatever it may be.

So with FeatureCAM, there's no need for an additional interface to program turning or turn/mill machines. As we program turning features, we'll use features that associate or are used commonly in a machine shop on turning, like we said before, turns, bores, grooves, those sorts of things. As we move into milling on turn/mill machines, you're going to see that we actually use the same milling features that we've been using so far. FeatureCAM is capable of controlling your simple turning machines that maybe have a single spindle or a single tool to very much more complex machines that have multiple turrets with multiple tool locations, maybe main and sub spindles and even b-axis heads.

So in the next example, we'll look at programming a part that could go on a very simple turn/mill machine, one that has just a single turret with live tooling and no y-axis. We're actually going to program using a machine that has a b-axis and a y-axis as well as a lower turret and a sub spindle. But it's not necessary in this case. We're going to utilize that a little bit later on to optimize that same part.

So let me close down this part and we'll open up our turn/mill part. So a fairly simple turn/mill part, like I said, it can be done with no y-axis at all. As long as we've got a machine with a live tool, we can completely machine this part. So I'm going to start by actually just running automatic feature recognition on this part just to see what FeatureCAM finds.

So, in this case, we can see that it actually found everything including the turning feature on the outside and the bore feature on the inside and all of the milling features that are along the z-axis. The only thing that it didn't find were these holes around the outside. That's OK. I'm happy with that. I'll say Finish. And then we'll just create those holes around the outside using interactive feature recognition.

So here's the big difference between turning and turn milling. Any time we're going to be using those live tools in FeatureCAM, we're going to select a turn/mill feature type as opposed to just a simple turning feature type. So we'll go Turn/Mill, Hole, and Extract with Feature Recognition. Hey, we've seen this wizard before, haven't we? This is the same one that we use in four-axis when we were trying to find our holes that were around that cutter body.

So we're going to say Around the Index Axis in Automatic. FeatureCAM finds all the holes we need, select them, and finish. That part is completely programmed less a cut-off on this side.

So let's do that. Let's add a cut-off. That's going to be a turn feature. We'll say Cut-Off. And it's important to note that you have the ability to transfer to the sub spindle right there within the cut-off feature, OK?

And you've got some parameters that you can set including pulling extra material out so that you're ready to start cutting your next part on that main spindle. In this case, I'm not going to transfer to the sub spindle. Although, when we optimize, we will be transferring it to the sub spindle.

So we're just going to do a simple cut-off but not three inches thick. Let's do something a little smaller. And we'll specify our location here. And we can finish, OK?

Swiss machines, configurations can vary greatly from manufacturer to manufacturer as well as, really, from model to model. But, in general, they differ from traditional turning, if anybody is not aware, in a few key ways. They're typically used in high-production environments and usually run really small parts. They have a sliding headstock and, really, a really tight, small machining envelope which kind of necessitates the need to reduce that excess movement as much as we can.

So in this next Swiss example, if you're not familiar with Swiss machines at all, hopefully you'll learn how they differ from normal turn/mill machines. And you're also going to see that we program in FeatureCAM for Swiss machines the exact same way that we programmed for turn/mill machines.

So here's our Swiss example. And this one is actually already pre-programmed. And I did that just so that you can get a feel for what these machines look like in case you're not familiar with how they operate and what they look like, OK?

So as we step through this simulation, what you're going to see here is that this is a bit different from your traditional turn/mill machine. We've got gang tooling along here. We've got our sub spindle doing a lot of movement to machine. And then we've got our sliding headstock up here actually moving to do the z-axis motion rather than the gang itself, or the tool itself, OK?

So let's look at programming this. We're going to focus on just this main side right here. Since there's not a whole lot going on on the second setup that we won't have already done, we'll just delete everything on the main side and reprogram it.

First thing that we want to do is, of course, face it. So we'll just go in and select a face and finish and then probably turn the profile. So we'll go Turn from Curve. And we already have a curve in here again because we had pre-programmed this.

It's easy to extract from the model. So we'll just say Main Setup Turn and Finish. We probably want to go ahead and poke the hole in the inside here in the center. So, again, we'll do a hole. And, in this case, it's a two millimeter diameter.

And I'm going to pick my depth actually off of the model. I'll just go down to here. It's 14 millimeters. I'm going to go an extra 15, or an extra one millimeter, to make sure that I break through that other side. And we've also got a small chamfer on the end there. Sorry, not 0.050, half millimeter. No? Is it 50? There we go. OK, and Finish.

Little red exclamation. Of course, FeatureCAM doesn't know what tool we want to use to cut that chamfer. No big deal. We select it. And we'll just use the 82 degree counter sync tool.

Moving on, we'll program these grooves here. So those are still going to be turning features. Groove, we'll select our first one, main set up groove. And since this is Swiss, we'll turn off our roughing operation and do the same thing for that second groove. And we're ready to start milling on this main side.

So, again, those fall under turn/mill features. And, in this case, we'll do this pocket here first. So we'll select that feature type. We're not going to extract it with feature recognition because we actually already have a curve for it. It's going to be curve 12. So we'll select that curve, specify our depth.

Now, if I didn't know my depth, I could actually pick that or interrogate the model to find that, maybe do a location along-- sorry, a distance along my x-axis. And I can just simply pick here and the bottom surface. And then I can just copy that and paste it right in there, OK?

Make sure that we're using an appropriate tool. We are. We've got a nice, small 0.8 millimeter end mill. One last milling feature that we've got on here that's going to utilize curve 10 and curve 11 to machine that.

Same process. This is still going to be an open cut, so it's going to be aside from those curves. We want to make sure that they're facing each other so that we're milling between the two curves. And we'll specify our depth of a half millimeter and finish.

So everything's programmed on this first side. What we need to do now is actually cut this part off so that we can transfer it to the sub spindle. So we'll do that in our cut-off. And we'll also activate transfer to sub spindle. We'll give us some parameters. So we don't really know at this grab distance is offhand. So let's find out.

Using the same tool that we did before, we'll interrogate the model and just say, give me a location in z right about there. Looks like 12 and 1/2 millimeters should be good. So we'll just set that at 12 and 1/2 millimeters. We'll start feeding at 15 millimeters so that way we don't wrap it onto the part there. Let's hit OK, set this out here on the end of the part, and finish.

One thing that I do want to do before I actually cut this off is I want to make sure, before I come in with my sub spindle to grab it, I want to make sure that my main gang is out of the way. So I'm just going to tell it to go home. I'll say Apply and OK. And we'll just step into our machine simulation. And there we've fully programmed the same part including transferring to the sub spindle and machining on that sub side.

Wire EDM in FeatureCAM is going to use those common shop terms that are found normally used on wire machines. So the feature types that we're actually going to use are going to include dye, punch, and side. You'll see that feature-based CAM approach can save time in programming wire EDM machines since all you really have to do is define the shape of the feature. So we're going to program a two-axis part, meaning that it's going to have those vertical walls. And we're also going to program a four-axis part with the tapered walls in FeatureCAM.

So here's our two-axis part. And you can see it's a pretty simple, basic wire part, vertical walls. And that's what makes it two-axis. So to program this, we'll just jump in and we'll use two-axis features. So we'll start with the inside pieces, do a dye, extract with feature recognition. And we'll just say we want this one, we want this hole and we want this hole, OK?

We'll define our thickness, just picking it right off of the model. And then we'll do the same thing. But this time, of course, we'll do a punch for the outside. We want this shape here. FeatureCAM remembered thickness from the last selection. And then we just say Finish.

From there, it's a matter of simulating. If we want to see what this actually looks like, we can just select the pieces that would normally be removed when we pull this off of the wire. And then we've got our finished part.

Four-axis really is just as simple. So here's our four-axis. You can see that these walls are, in fact, tapered a little bit, making it four-axis. So we'll start the same way.

We're going to cut the hole first using dye. Here's our profile. We do want to make sure though that we're cutting to the stock. So the blue shape is our stock. So we'll offset this a bit from the curve location. And then we'll pick our thickness from stock to stock, OK?

Four-axis, well, you probably saw as I was working through this wizard, we've got a whole section for four-axis. So, in this case, we're cutting on the outside. We'll select a punch and extract with feature recognition. FeatureCAM gets rid of everything that it knows you're not going to need to create this. So I'm going to select everything but that one hole that we have there and then just finish this.

Quick 3D simulation. And, again, we can clean those up. And there's our finished part.

So this brings us to a bit of optimization. For the example, I'm actually going to focus on optimizing a multitasking turn/mill machine, in this case, simply because that's where we can oftentimes get the biggest change from optimization. So it's never really necessary to optimize your multitasking turn/mill programs. However, it can offer you several benefits.

One of the biggest is using the machine to its fullest potential. And, ultimately, that means using maybe upper and lower turret's at the same time, doing machining work on the main and the sub at the same time or maybe at different times depending on how things are synchronized. But, really, that's what we want to do. We want to be able to utilize as much of the machine as we can to reduce the overall cycle time.

So, in this example, we're actually going to use that same turn/mill part that we programmed a little bit ago and on the same machine that we programmed it on. So the machine has a b-axis head. It's got a lower turret that's capable of holding live tools as well as a main and sub spindles. So we're going to do a little bit of synchronization with that. OK, we'll load up this part file.

So here's our same part, same program that we had before including our cut-off. I will note that, in this particular instance, I do have it set to transfer to the sub spindle. And I set in some transfer distances, OK?

So if we look at this in a machine simulation right now, as we watch this, what we'll see is that we're very inefficiently using this machine, OK? All of our work currently is being done with that upper b-axis head on this machine. So certainly we've got some room for improvement.

We can verify that. If we go to our tool posts here, we can see that, in fact, looking at an operation view, everything is done with that upper turret. Nothing is being done with that lower turret or upper b-axis head, really, OK?

So there's a few things that we could do. We could maybe pinch turn the outside profile. Now, in this case, it's not really necessary because we're not removing a lot of material. But if we wanted to, we could easily go in here and select Pinch Turn. And FeatureCAM will automatically drop in a lower turret tool and set those weight codes for us, those sync codes for us. I'm going to set that back to just doing a single. Because, in this case, it's not particularly advantageous.

One other thing that we'll notice on here that we can do pretty easily is, if we take a look at these holes that are around the OD, hole 6 and hole 12 are opposite of each other. Hole seven and hole 13 are also opposite. And, actually, it works that way all the way down.

So what happened is, during the process of recognizing those holes, FeatureCAM just went around in a clockwise direction and recognized the holes as it went around. Since these are opposite of each other, why not machine half of the holes on the lower turret, half the holes on the upper turret, and do them at the same time?

So let's do that. We'll select hole 12 through hole 17 and we'll just drag it over here to our lower turret, OK? So already we're using it on the lower turret.

FeatureCAM is smart enough to tell you that, hey, it looks like you're trying to turn and mill at the same time. This isn't going to work, OK? So we need to do a little bit of synchronization. In fact, what we want to do is set hole six and hole 12 to start at the same time.

So using my control key, I just multi-select those and say, set my sync point at the operation start. Do the same thing all the way down here. Just in case, maybe the upper b-axis head is a little bit faster than the lower turret and finishes drilling one of the holes before the lower turret does. That way they all start at the same time. We index, start again, index, start again, OK?

So we jump into a simulation here. Let me just hide this for a minute. What we're actually seeing here is that we've got a really long tool on our lower turret. We can fix that easy enough. FeatureCAM is giving us that little warning. But as we come in to machine here, now we're using both the upper and the lower to machine that.

So what happens after you've already completed a program, you take it out to the shop floor and you find the machine that you intend to run it on and that machine is not available? Maybe it's down for maintenance or maybe somebody else is still running a part that ran long on there. Ultimately, that means you need to re-program or you need to change what machine you're programming it on.

Now, if you've ever been in a machine shop similar to that situation that has many different machines of the same type, say three-axis mills, you'll know that it's often necessary to run parts that were designed and programmed to be run on one machine or on a different machine for who knows what reason. As a programmer, for us, that means that making that switch to a different machine in the most extreme circumstances means that you need to re-program the entire part for the other machine.

FeatureCAM makes changing machines easier because, really, you're just defining the end result. This gives FeatureCAM the flexibility to change the necessary things when you need to change machines. So, of course, you need to change your post-processor to get the correct NC code. But in FeatureCAM, you can change your tool crib as well. And that will change the tools that you're running so that you're actually using that proper machine or the tools that are in that machine very easily.

Just let me close this part file down and we're going to load up the part that we programmed at the beginning of this class. OK, so same part that we had before. Let's say we intended for this to run on whatever, let's say a Haas machine to begin with, OK?

So we've simulated. We like everything that's programmed here. In fact, we've already got our NC code. We go out to the shop floor and that machine is not available. So now we've got to make some changes. No big deal.

Just to highlight some of these changes, if we look at, say, this pocket right here, right now we're using this generic 32-millimeter end mill that's found in FeatureCAM's tool database. Well, on the other machine that we have out in the shop, we don't have this generic tool. We have something a lot more specific. And, in fact, those guys like to see the program using specific tools with specific names.

That's fine. We've already set up FeatureCAM with the tool crib necessary on that other machine. So all we really have to do is define what that machine is. So, in this case, we'll say DMG. It changed our post-processor for us.

But further to that, it actually went through and changed all of the tools to be tools that we have available in that tool crib that's on that machine. So, at that point, we can just run our simulation. Now we've got our own NC code ready to go out to the other machine.

So through the duration of this class, hopefully you've learned how FeatureCAM is able to program a wide range of CNC machines and learned a bit about the range of machines that are available today. I hope that you've learned how FeatureCAM is able to adapt to programs from one machine to another very easily and a little bit about optimizing multitasking programs to be more efficient.

Your feedback can definitely help me improve in future years. So I would love to hear from you on how I did today. So take the time to fill out the survey. If you have the AU mobile app, you can do that now. Otherwise, you can do it later online.

So it looks like we've got a little bit of time for questions. Any questions from anybody? All right, fantastic. Thanks for coming.