

**MIKE THOMAS:** All, right well I guess we'll get going. Thank you for everybody who's showed up here on Thursday afternoon on the end of AU. My name is Mike Thomas, I'm a Technical Services Manager with a large mining equipment manufacturer. With me, we've got Kevin.

**KEVIN ELLINGSON:** Hi, I'm Kevin Ellingson. Worked with Autodesk projects, a lot of years in the industry, worked for Autodesk reseller, now I'm kind of an independent consultant on CAD/CAM products. Okay, want to go to the next slide?

**MIKE THOMAS:** Okay, so, what our goal is today is we want to show you kind of the ins and outs of what data can go into and out of Inventor. So, we will maybe show you some different areas never maybe considered, some different avenues, also take a look at some of the new features, like kind of come into Inventor with 2016 and 2017. Next slide.

So, I guess maybe just a quick poll, is anybody not using Inventor? Everybody's using Inventor? Great. Anybody-- who's on 2017? Who's on 2016? Who's on 2015? Who's on 2014? And the people who didn't put up their hand, what version are you on? What version are you on? I didn't see your hand go up.

**AUDIENCE:** Oh, it's 2017, I'm just a little slow today.

**MIKE THOMAS:** You want to go to the next slide? All right, so, you're probably tired of seeing this all week long, so, instead of looking at the future of making things, let's look at the reality of making things. So like, what can we do today, not waiting for the future?

So, obviously technology is disruptive. I mean, you look at the two slides here and you can see how things have changed, maybe in the past 100 years. And obviously, everything is accelerating quicker and quicker and quicker. I was listening to a podcast on the way down and they were talking about the history of war, and how you could have taken Alexander the Great's army versus an army from the 1700s, and they would have been equal, but if you would have taken like France's army from World War I they would have completely destroyed Alexander the Great's, and how we hit the Industrial Revolution, and then even in the military it just completely spiked. So, things are going faster and faster and faster. Want to go to the next?

So, in the grand schemes of things, we've kind of gone through three eras. So obviously, we

started with computers, and we started documenting everything. So, we started dimensioning and annotating and capturing all this information we were able to store electronically, and then we moved into optimization where we got into 3D modeling, and we started doing stress analysis, and we started doing all these type of things where we could optimize the models. So, not that we're out of the optimization, but now we're moving into the connection realm, where we want to grow past just the 3D model.

So trends, obviously, is people want it in shorter time, which means we have less time. We don't know when they're going to want it, so it's going to go up and down. And they're going to want it the way they want it, so they're going to want it customized, and we don't know where that demand is going to come from. So, instead of worrying about selling into your neighborhood, now you're worried about selling worldwide, the demand coming from worldwide.

This happened to us, probably the past two, three years, because I'm from Canada, and for the longest time we focused on our province. We were selling into the mines, into our province, and it was good, but then all of a sudden, Potash took a dive, and all of a sudden, now what? We started looking around, and also we started getting demand from Russia, and Brazil, and markets where you know the Potash, or other products, wasn't as impactful as what it was in North America.

So, the transit production, obviously, the focus we are, is Collaborative Distributed Design and Production. That's kind of what we're going to focus in on this class. But also, what can Inventor do with manufacturing? Obviously there's lots of Advanced Materials and Fabrication, we're not going to look at Generative Design. I mean that's going to be coming, but really look at the Collaborative and Distributed Design and Production, and what we can do within manufacturing.

So, what we're going to look at is, we're going to look at the technology. So, what technology within Inventor and Design Suite, or the collection, that can enable us to do that. You want to go two slides? Right, so we're to look at the Inventor. Next slide. So Interoperability, I thought this was perfect. Googled it, and this is the very first definition that came up. As you can see, it's the ability for different information into different systems. And everybody's got that problem, right? We've got to go from CAD into manufacturing, we got to go from CAD into data management. We've got to go from CAD into our URP or MRP or whatever systems. So data's got to move around quite a bit.

So, when I started in the industry 20-odd years ago, we all worked in a silo. We had our team, our company, and maybe you'd send out a DXF, or maybe you'd send out a drawing. Can you imagine 20 years ago saying, here is my entire model? Just go with it? It wouldn't have happened, for lots of reasons. I mean, internet, and there was no real email, but it just was a foreign concept. We didn't exchange data. It was like, we were within our silo, and that's where we stayed. So, Inventor has changed quite a bit from the very first version of it, which could open the data.bg file, but it couldn't actually generate a data.bg. File it could create a DXF and maybe a STEP file, that was about it. So, even Inventor's lifetime has changed quite a bit.

So, we look at the couch collaboration tools one of our next slide Inventor can work with all these formats that you see up here. So Inventor has the ability to read and write the majority of the formats up here. So, it can work with SolidWorks and Siemens, and as you can see Artist products like Alias, it can work with neutral file formats.

So, has anybody heard of the AnyCAD functionality? AnyCAD? It's one of the greatest enhancements, best enhancements that's come in Inventor easily in the past five years. If you haven't heard of AnyCAD, what AnyCAD allows you to do is it allows you to take one of these formats, and you don't have to translate it. You basically just attach it into your model as a reference. You can constrain to it. You can actually kind of in a roundabout way, but you can add features to it. And if that original format changes, guess what? It updates in Inventor.

So, that means that somebody could be using SolidWorks, maybe it's one of your vendors, or your customer, and they send you the SolidWorks file, but you don't have to worry about translating it, and converting it, and missing faces, and reverse faces, and holes, and gaps, and all the stuff that you've got to worry about translating and fixing, whereas you just simply attach the model, and the model stays associative to that file. So, they send you an updated SolidWorks file, it updates within Inventor. And that works for anything that you see up there.

Inventor also has support to bring in point cloud data. May not have thought that Inventor as this ability, but just like AutoCAD, you push it through ReCap-- ReCap's Autodesk's program to deal with point clouds-- and basically it's just got to kind of scan that file, itemize the points in there, and then you can attach it with in Inventor. You can bring in PCB boards. It will work with AutoCAD electrical data.

So in electrical, mechanical realm, you can take an AutoCAD electrical project, which has components in it, basically link that project with Inventor, and say, okay, I want to insert these

components, and you're actually inserting the same component that's in the AutoCAD electrical schematic. So you have basically one item that it's pulling data from, and if you remove one from one side, then there's a little flag that says, oh, we removed this, you need to remove it on this side as well.

Okay, well, on the communications side of things, what we have is we have things like DWG And DXF. You can go to the next slide. So, good old traditional DWG, DXF. I mean if you want to get something laser cut or plasma cut, that person is going to want a DWG or DXF. We can do PDF, and as of Inventor 2016, 2017? 2016. We do 3D PDF, so you can take your 3D models and you can publish them to PDF.

There's also this new option called Connected Design? Paul, what's it called? Design Connect? Anybody? Connected Design. I'm assuming, by the looks, nobody's tried it? Nobody's tried it? How come? Why hasn't anybody tried it? Yeah, a little blue button in the bottom right corner by default. No one's seen the blue button either? Okay, we'll show you what it does. So, we also have the Design Suite, the different flavors. It's also the new product collection of manufacturing software, assuming everybody is on Ultimate or Standard, or some form.

So, you get as part or you download, you get this install window that's about this long that you need to scroll up and down to find all the software. But you got of all these different components within it. So, within this collection of products, you could take an AutoCAD drawing, and you could bring that into Inventor as the starting point for your model, or maybe to help position your model where it needs to go. You then could do things like cable wire harness, or CAM, or FEA within Inventor.

Then we can do marketing stuff, like create videos and pictures within Max. We can, like we talked about before, use ReCap to bring the point cloud data. We can use Navisworks to pull all that data together. So, with all that software in there, they all kind of talk to each other, and you can put data back and forth between them.

The last little area we want to look at is within manufacturing. And that's why Kevin's here, this is his specialty. Move on to the next slide. So, within the manufacturing realm, what we can do is, we can do CAM, we can do 3-D printing, or at least prepping your model for 3D printing. And there's a whole realm of stress analysis. And that too, that area within Autodesk is all connected.

So, if I start a stress analysis, do an FEA analysis within Inventor Professional, I can actually push that into sim-mechanical, I can push that into CFD. And they all kind of talk to each other. So, even if you're using, let's say, Robot Structural, you can push that up. Or you could bring information from the Inventor FEA, into Robot Structural. So even at a stress analysis area, they all talk to each other.

So, then what that brings us at the end, is all these components together. So this is the last slide and then Kevin's going to take over. But as you can see, we've got all these areas that are connected. And Inventor is kind of the hub to it, where we can bring in data from different sources. We can communicate with people outside. They can communicate with us. Everything stays associative, so if it changes in one place, we can push that change all the way around. So connected collaboration, communication, and manufacturing.

So now, Kevin's going to take us through a few things here. I should also say, if you guys had questions, we're a small group here, we can just throw them out. I'll either answer it, or I'll say, wait till later, or I'll say, I'll use the AutoDesk term and we'll take it offline. But if you've got a question, just [INAUDIBLE] and we'll do our best to answer it. I think that's to exit the slide show. There we go.

**KEVIN**

Okay, so first thing I'll talk about is AnyCAD. Has anybody tried AnyCAD at all, any form of it?

**ELLINGSON:**

Okay. So AnyCAD's pretty cool, as Mike was saying. Let's make sure this is as big as it can be. I'm going to start a new file. And I'm just going to open a blank file. So now, I'm just going to open something, and I'm going to choose to open a SolidWorks file. And I don't have SolidWorks loaded on this machine, so I'm kind of going to duplicate the process of what would happen if you're working with a vendor that uses SolidWorks files.

But I'll just open up this first SolidWorks file. You get this dialog box that pops up, and it's asking you, do you want to reference this, keep in its native format or do you want to convert it to in Inventor format? Once you convert it to an Inventor format, you've broken that link back to the original native file. If you leave it referenced, you'll see what'll happen in a minute.

So we'll just go ahead and hit Open. Nothing really crazy about this part but, the area I'd like you guys to focus on when I open this next model is right about where the slot is. So, I'm just going to go into Windows Explorer and duplicate this process. So it's from my desktop.

Okay. So, here's those two SolidWorks files, basically one was created, then the next version

was created and saved as the new name, so I could duplicate what the process was. So, I'm just going to take the one that I opened up, and I'm going to rename it so Inventor stops looking at it. And then I'm going to take this 2D chamfer and rename it as the first one.

As soon as I hit the Enter button, you're going to notice this little lightning bolt light up. So, there, that's lit up. Inventor now goes, oh look at that, there's a new revision of this SolidWorks file that's in this. Did you want to update it? So again, if you keep your eye on this little slot right here, when I hit the Update button, that slot gets bigger.

So, I can constrain to this, I can do things with it. If the vendor sends me a new file, I don't have to start over, I don't have a re-import, break all the things that have breaks, reconstrain it, reposition it, and do my drawings over. I can just keep it in the same, format that it was in.

So, to go a little farther with that, I'm going to open up a STEP file. For me, this is one of the great things, again I'm going to reference in the STEP file, keep it native linked. I work with a lot of people who do CNC machining, and their job shops, and they get a lot of file formats from different CAD people. They don't care about what the CAD format is. They want to put tool paths on the parts, and before they hit the Cycle Start button, if the engineer comes back and says, stop, wait, we made a bunch of changes, in the past that would have been not good because you pretty much start from scratch, or do a lot of work.

This is going to be kind of a simplistic example, but, a side note is anybody here doing any CAM work at all. Have you tried HSM yet?

**AUDIENCE:** Yeah, I've used HSM.

**KEVIN ELLINGSON:** Okay, cool. So it's-- HSM is just-- it's a really good pickup by Autodesk, it's really easy to use fast CAM software. So, I'm just going to switch the CAM tab, and we're going to do something called the set up. This is how we prepare a model to tell the CNC machine how to run. So, I'll just get my work coordinate system set up, decide where I want to put that on my part. And I'm going to add a couple of tool paths. So, I'll just do a quick face, I'll grab a facing tool, click Okay.

There, you can see the tool path generates the face, the part off, and then I'll come back and view a 2D adaptive to clean out that pocket. Tutorial inch, we want a half-inch cutter. For the geometry, I'm going to select this bottom face. Go ahead and click Okay. And you'll see a tool path gets generated.

So now, again let's go back into my documents. And if I were to go and rename this STEP file, again, when I hit Enter, we should see the lightning bolt come on. It did. If I come and I regenerate this now, you'll see the pocket got much larger. My two tool paths now show a red x next to them. So, it's telling me that these tool paths are no longer in sync with the model.

Typically what we would do in a traditional CAM software, we would throw it all away, start working again. With AnyCAM-- AnyCAD I should say, AnyCAD inside a CAM, I'll right-click on my setup, tell it I wanted to generate all the tool paths, and now my tool paths now understand what the change was made, even though this is a STEP file, it's able to work with different versions as if it was a native Inventor file, working through the design changes.

Any questions on AnyCAD, AnyCAM type functionality, so far?

**MIKE THOMAS:** So, for those who are aren't on 2017, AnyCAM was actually introduced in 2016. So, they made some minor changes to it in 2017 but you do have access to it if you're on 2016.

**AUDIENCE:** 2016, you can't reference STEP files, right? It has to convert?

**KEVIN** No, 2016, you should be able to reference STEP files.

**ELLINGSON:**

**MIKE THOMAS:** I think that came in '17.

**KEVIN** Oh, it did? Okay, I lied. 2017 STEP files. Okay, so another environment that we have built  
**ELLINGSON:** inside in Inventor, we go to the environments tab, and we're going to see the 3D printing tab. So under 3D printing tab, I'm just going to choose that, it's going to open me up into a new window. In this case, we can set the orientation. We can kind of reposition our part within the box. The box shown right here would be the printer that I'm using, and the print bed size, so I can kind of get my part position as I want to. It's also got some cool tools. If I had a part that was too big for the printer that I was making, I can partition the part, break it up, and build that print in multiple stages.

So, once you're happy with the results, one thing we could do is generate an STL, which most printers want. I'm just going to put this out on the desktop, and we'll call this-- enter. Part-- we'll just call it that. We'll go ahead and hit Save. So we'd send it to our printer. If you had-- you could put Print Studio on your computer as well, which is another free Autodesk product that helps you do some more manipulating of 3D printing objects. I'm just going to go and exit the

3D print.

And this will give me another opportunity to show that we can work with STL files. So, if I go over to the manage tab, there's an Import button, and on the Import button-- go to my desktop-- and I'm going to tell it that I want to look for STL files, and when I do that I should see my Interop file right here. Go ahead and open it. And now what I've opened up, this is a mesh, it's not really a real solid. You can do a limited work with it. We do have some tools now though that will help us with that.

On the 3D model tab, you'll see that we have the fit mesh face, so I can do things, convert faces, edges, cylinders, to real geometry. Kind of a manual process. It works somewhat reliably. I mean, the more complex your model, the more difficulty you're going to have with it.

There's also another tool that Autodesk makes. We'll put a link to this in our documentation for the class so you can go and find this tool. But what it will let me do is take this STL file, right-click on it, and say, I want to convert it to a base feature. I want to choose a solid that I want to convert it to. And if you'll notice, the faces are triangles, STL files are basically just triangles fitted to whatever shape they need to make up. When I click OK, you'll see the triangles disappear off the flat faces. So, if I go ahead and click OK here, does this little conversion

So, now it's found all the planar faces and things like that, and it's capped those, and made those smooth. On the edges, we still-- it just kind of took the triangular faces and converted them to solid geometry that are triangles. But at least now with this, I have a solid body. I could machine it or constrain it better, things like that. Any questions on any of the STL files? Import/export?

**AUDIENCE:** So, with the mesh file, will it import STL and OBJ, I think, the two file formats?

**KEVIN ELLINGSON:** Yeah. I don't know if anybody had tried a mesh model prior to 2017, but more than just the tools, also the performance. So used to be, you'd bringing in a mesh model in 2016, and it would bring your system to its knees, even something like this. It was just so many faces for it to deal with. So I don't know what kind of black magic they are doing now, but the performance is significantly better with mesh models than it was before.

Okay, so the next one, I know we're in an Inventor, class but we're talking about using Inventor data, and we're going to talk about using Inventor data in Fusion a little bit. So, I'm going to-- if you haven't seen Fusion, this is Fusion. There's really no file open in Fusion. The way you get

data in is to import it into your cloud account. It converts and gives you a file you can work with.

So, I'm going to open up my data panel. I'm going to click on the plus to add my file, go to my desktop again, and I want to import this Inventor file. I'm going to upload it. It's going to take a second to upload.

Anybody, ever created a design that got such a complex design history, or was designed in such a way that there was no real way to back yourself out of making the changes you want to do, because there were so many feature dependencies on there? If you ever have, this might be a good workflow to get you to make that geometry change that you need, a little bit easier using direct modeling.

So hopefully the internet's not too slow in here. Give it a couple minutes. How many people have tried Fusion so far? Definitely something to keep your eye on in the future, along with Inventor. We were at a lunch with-- Carl Bass was speaking. And he gave Inventor 5 to 10 plus years, so, if you're sitting on inventor, there's no panic that it's going disappear overnight.

**AUDIENCE:** [INAUDIBLE]

**KEVIN ELLINGSON:** We'll tell you what, Mike, what do you think about doing an audible? I'll let this run, well, maybe we can save this section at the very end.

**MIKE THOMAS:** Sure, yeah, because it didn't take this long before. The internet connection must be a bit--

**AUDIENCE:** [INAUDIBLE]

**KEVIN** Exactly. And there's no way it's going to manage the 100,000 files I've got in Vault, either.

**ELLINGSON:** Whoops. All right, so what I want to show here, if my mouse would come back, is kind of a different form of the AnyCAD functionality, is that we can do what's called a DWG underlay. So I'm going to create a new assembly. In that assembly I'm going to create a new component. Let's say we'll just call this floor for right now. And I'm going to position this so this actually is the floor.

And what I'm going to do is I'm going to go to the manage tab, and I'm going to do an import. And one of the things that we can do is we can actually import AutoCAD drawing files. So kind of the same thing as the AnyCAD-- that's not the drawing-- is that we can bring this in as a reference. I'm just getting a message because it's outside my project file. But we can bring this

in without the need to convert it. So I'm getting another message because I haven't saved my file yet.

So, what it's doing is almost like an AutoCAD xref, is that it's bringing this in an associative fashion, so I can place it within my part, I can locate where I want it, and then I'm able to use that in different means. One might be to project geometry, one might be just to use it to position objects.

So, now I've got this AutoCAD drawing attached here as a reference. What I'd like to do is bring in this assembly, and let's bring this assembly in and let's just drop it in there. So I'm just going to do a copy, and we'll paste that in there. And then what I can do is, I can use regular constraints, and constrain this to the AutoCAD drawing.

So, I could take, let's say, this plane, really, I can take this plane, and what maybe I'd like to do is I'd like to constrain it to that line. And then maybe I'd like to take the edge here on this foot, and let's constrain it to this purple box here in this.

So, now I've used that AutoCAD drawing to position. And if we take that purple box, and let's just move it, and we'll save it, when we go back into Inventor, same lightning bolt, and what we'll see is that the purple box is going to move. And since that assembly is constrained to the purple box, it's also going to update as well. Hopefully.

So, you can see that I didn't have to convert any geometry, there was no copy and pasting, re-constraining, anything like that. I was able to attach it as what's called an underlay, but it's really the same as using the AnyCAD functionality. The original AutoCAD drawing changes, then it updates within Inventor.

Now, we can also kind of do it the opposite way. So, I've created a drawing of that assembly, and what I'd like to do is, I'd like to reference and use that geometry inside of AutoCAD. So I'm going to use what's called the Design Center. And if you don't use AutoCAD, the design center is just a way to access content within a drawing without having to open it. So you can access layers and blocks and things like that.

So, I'm going to take a drawing view. So when you save in Inventor a drawing, whether it's a DWG, or an IDW, you can go into AutoCAD, you can use the Design Center, and you can drag and drop it, and insert those views in as blocks. So now I could take that block, I could position it within the layout, make sure everything is going to fit.

Now, the same thing works, is that if it changes in Inventor, obviously, so let's change the level of detail here. So, since it's a block in AutoCAD, it's not going to update automatically, so we'll just save that drawing. But what I can do is, using the Design Center, I can quickly redefine the block. So, by redefine the block, in a sense, it's going to go through and update that block anyways for me.

So, I go back to the Design Center, I can right click, I can do a redefine, and what we're going to see is that the view wasn't quite updated yet in Inventor, so I get the green squares. But once that view is updated in Inventor, then you can redefine the block.

Something that's also pretty cool about this, is with that block inserted into AutoCAD, you can actually explode it, and you get the lines. Now obviously, it's not going to be associative anymore at that point, you won't be able to redefine. But if you just need a view of that model as 2D lines, sometimes the easiest way, insert the view through a block, explode it, and you got the lines. There's no save as, no worrying about layers, or things like that. You just drop it in, and you save as, and it updates.

So, that's just a couple of different ways that we can deal with AutoCAD data. So, we're into AutoCAD. So you can use the DWG underlay, bring it in, and you can use the Design Center to bring blocks into your AutoCAD drawing.

Something else that we can do from an Autodesk perspective, is we can work with Alias Design Data. I'm not sure-- anybody use Alias? Not one? Anybody know what Alias is? No? I won't spend very much time on this, then. So, the idea behind the Alias, is that it creates surfaces. It's basically an industrial tool for sketching kind of free form surfaces.

So, you can see here that I'm using the AnyCAD idea here, and I'm associating this Alias waterfall. So the same thing will happen. I've got all these surfaces within the Alias file, I can pick and choose which ones I want to bring in. But again, it's going to stay associative to the Alias file. Now, let's take this-- let's actually save this. What should we call this. Let's call this Alias . I'm going to use this as an opportunity to talk about something else. So this doesn't necessarily have to be coming from Alias. I don't know why I keep pasting-- trying to paste over there.

Anyway, I'm going to create a new part. And what I want to show is this tool that's called copy object. So, copy object will work anywhere and everywhere. So, I'm going doing it off an Alias

file. You can actually use copy object on another Inventor part file. You could do it off a STEP file, you can do off a SolidWorks file, or you can do it off all these different-- they're different files. And the idea is that you can copy these objects.

So I'm going to copy faces. Because I only really want to deal with a couple of faces from this Alias model. And notice the associative option. So, the idea is, I'm going to copy these objects into my new part, and it'll stay associative. So, now I've brought in an Alias file using AnyCAD, I've now copied objects into a new part file, and I'm going to use these faces that I brought in, and I can use the thickness feature.

Okay, I shouldn't have done it as a composite. But the idea here is I could take this now, and I can do a thicken, and I can thicken that face, to get a solid model. So then if my Alias model changed, it would update with AnyCAD, the data that I copied object would update, and then turn my part would update. So, copy object is one of those little hidden tools because it's kind of buried in the modify drop-down, but again, it works for anything. I could have an Inventor part file, and I could copy a face from Inventor part file into another part file. And I've got this link that I could sever or maintain to make sure it updates.

Another format that we can work with, which has actually been Inventor for quite a while, is IDF files. And they keep working on the options that we can do with IDF files. And if you were an IDF file, where would you be?

So, really, what this is, is this is a PCB board that was generated in a different piece of software. We actually use the Eagle software at work, that's become the Fusion schematic PCB information, but this is where it came from. And IDF has been in Inventor for as long as I can remember. So IDF has been in Inventor since, I don't know 2010, 2009. In the past couple releases, what they did is, they provided the ability to actually recognize your Content Center components, so instead of it generating brand new components each time you bring it in, it'll actually look through your Content Center, and say, oh yeah, you already have one of those components, and whether it's a terminal or something else, and actually replace that. So you're getting your component with your iproperties, and your information.

In this case, I don't have any, so if I do a check, it's just going to come up and say I can't find any of them. But you can see that it is going to do that check. So I'm click okay, and what we're going to see is that it's going to create for me a new assembly, and within that assembly will be all the components. Now, I picked to do it as an assembly, but I could have easily done

it as a part, and it would have come in as a multi Solid part file. So there's the board, now it's placing the components, and I could take this and then paste it into and constrain it.

Okay, so one more quick one here, and then we'll turn back over to Kevin so he can-- what do I want to do? So, let's change the representation. And let's look at the little blue box here that obviously everybody in this room ignored when they put 2017 on their machine. What we're able to do is, we're able to use this and create a design share.

So what this is utilizing is, this is utilizing A360, as you can see up in the corner there, so you're utilizing A360. So the idea is that it will take your model, and basically publish it to this lightweight format using the A360 viewer, and generate a link for you. You send that link to somebody else, and they can hop on, and see your model in real time, and they don't require an A360 account. You require an A360 account to publish, but they don't require an A360 account to view it. They can add comments, they can take a look at your model, and then when you're done, you just kill the design share, and the file goes off into no man's land, never to be seen again.

So it gives you this real quick ability to push it to A360, collaborate with somebody, and then remove it. So, you can see right now, it's going to share what's currently visible on the screen. So, I'm just going to start a design view. We'll give it a name. We'll give it some content. We can include components, we can include the part properties, and then directly from here, you can also send an email and say, hey, take a look at this. But I'll start the design share, and you can see that it's extracting the 3D graphics.

So we might have as much success as Kevin had going to the cloud. Yeah, I think this connection might be overburdened. I switched to my phone, and as soon as I switched the connection on my phone line, everything happened almost instantaneously.

**KEVIN**

**ELLINGSON:**

We'll give 30 seconds. So, as you can see, we took the model as it was, so which components were visible. So I'm able to pan, zoom, you can even do these exploded views, or you can expose the components part. There is a bottle tree so can see the components there within it. You can interrogate the model, get some properties on it. And then you can see that there is the ability to do comments.

So, right now there is no ability to add comments directly in the model, leaders or dimensions. It will come. But right now it gives you this ability to quickly publish your model, shoot the link to somebody else, you guys can collaborate, and then you can make the share go away. I'll float

it back to you.

**MIKE THOMAS:** Okay, so here I am back in Fusion. My model published. Here it is, this Intro to Inventor, or Intro to 2D Machine Part. I'm going to right-click on it, and say that I'd like to open it. So now, I've opened this file. And you'll notice that there's maybe some-- we don't get the Inventor feature history tree. It's gone through and done some automatic feature recognition for us, which works okay. We'll talk a little bit about what you can do instead.

But let's just say-- there's four fillets in the back side, and if I switch my units here from millimeters to inches, maybe what I'd like to do is, I brought this in, this is an imported model, but I would like to change these four dimensions. So, I'm going to right-click on my fillet here, and say Edit, and now I could give it a bigger value, say, like 0.2. The fillet updates, click okay. This is being done history-free, there's no parametric history for this, but we can still do some editing.

Another thing you could do is-- so this is all part feature recognition. I could right-click on these features and say dissolve, and now all those all those features have gone away. However I can still kind of-- sometimes Fusion doesn't group things the way I would like it to, or maybe I want-- there's four fillets here, maybe I'd want those all to be linked as one feature. So under the create, I'll go through and say, find features. I'll unchecked the select-all, and say I just want to find fillets, and the fillets I want to find are that one, and that one, and that one, and that one. And we'll go ahead and click okay.

So, now I get a fillet, I can right-click on it, and edit it again. It's found out what the value is. I'll switch to something big like 0.2. Click okay. Now we've got, again, non-parametric history-based direct modeling. Fusion is also really great, like I said, I've done this before. I just need to make one small change, but there are so many features that are dependent down the road on that change that I'm going to do, it's going to break a lot of stuff. Now, the Fusion has really great direct modeling, so maybe I don't want this hole, I'll just come delete it. Maybe I don't want this chamfer. I'll delete it.

It self-heals-- okay, so then I'll go ahead, delete that off-- so you can just delete chamfer or push-pull, maybe I want to make this the top surface of this part a little thicker, so I'll right-click and say press pull, grab the top face, drag it up to the value that I want so that you can see these chamfers are giving me a little bit of an issue. I could go and delete those and redo this instead. Let's just do that.

So let's click on the chamfer and delete it. You can control select them as well so we don't have to wait all day. So I'm going to get rid of my chamfers, now I can right-click press-pull. Do that design change, that might be really hard to do in a parametric based design when I have my history that is kind of locking me in a corner. I get my design change made, and again, this is all about getting data in and out of Inventor.

So now, once I make those changes, I could go to my menu, tell it that I would like to export this. I would like to save this as a STEP file. Choose where to save it, I'll save it on my desktop, it should go out there. And now we can come back into Inventor, open a file, there's that intro to machine STEP, I could open it.

Again, it's asking me if I want to stay linked to the STEP file that I created. So, I could continue to make design changes in Fusion, and re-exporting those back out as STEP with the same name. When it overwrote it inside of the computer, you would get that, update that, the changes have been made, you want to update it, and you can update it, and all those design changes will follow through into Inventor. So click OK, and there's my there's my model that's been changed from Fusion back into Inventor. So, just another tool in the toolbox to do some editing as need be.

**MIKE THOMAS:** Are there any questions?

**AUDIENCE:** I've got a quick question about-- so, we've got another customer that we work with, they send us their STEP files, and they also use Inventor. They send us the STEP files though. A single Step FILE. When we import them, if we reference, does it just keep that single STEP file? The problem we've been having is we've been like, bring it, every single part in assembly.

**MIKE THOMAS:** It basically will treat it like a multi Solid part. You get a single reference. Yeah and then similar to what would I do with the Alias file. You can switch the second tab and click Load, it will list all the components in there, and you could say I don't want these ones. So might be 500 parts in there, you only want 100. But still one STEP file, one reference within Inventor model.

**AUDIENCE:** So then, like you said-- we've never referenced them. So, say he sends us a new step file. How do we get it to update? Replace it?

**KEVIN ELLINGSON:** Just replace it. Don't change the name. Make sure it's the same name. Replace it, Inventor will-- keep it the same name, Inventor will say, oh, look at this, there has been a change made, did you want to update this? And you hit the Update button. Because, if you look, let's see if I

still have open. This SolidWorks file is a feature in my history, in my design tree now. So, it's--

**MIKE THOMAS:** No, they can't see you. It's like that doesn't look like anything.

**KEVIN ELLINGSON:** I'm sorry. So, I switch to SolidWorks file I used earlier. In my browser, this is the inserted SolidWorks file, but if you look, my Inventor part file is named nothing, it's still just a generic part, so if I were to say my file, it now knows that this is part of the history. So you drop that in your directory, wherever it's supposed to be stored. You open that file again, Inventor goes, whoa, look at that, there's a change, wants to know if you want to hit the lightning bolt and make the update.

**MIKE THOMAS:** And that's a good point, because Kevin inserted into a part file, that you could just place it into an assembly file, so you have either method you want to work with. Yeah, like I said, it's one of the best features in Inventor, easily in the past five years. Were you on 2017? Yeah. So it should make a big difference for with that those STEP files.

**KEVIN ELLINGSON:** And again, every year they seem to make it a little bit better, the way AnyCAD works, and the reliability, and what it's able to do.

**MIKE THOMAS:** So just to expand on the DWG underlay, is, I'm going to import, again I'm going to import an AutoCAD drawing file. And I'm going to pick the view I want to drop this in, and I'm going to pick an origin. I'm getting messages because, again, I didn't see my part file.

So, it gives you a message right away that you've inserted this file, and there's a project command if you want to use the geometry. Now, before I do that though, I'm going to right-click on this option here. And notice the ability to suppress the link and break the link. So just like derived parts, if you don't want this to update real time, you can suppress that link so that they can change the drawing all they want, and until you unsuppress the link, it won't update.

But they also do things like redefine, reposition, I can also do layer visibility. So if there's certain layers in there that you didn't want to see, so maybe I didn't want to see the dimensions, maybe I didn't want to see the hidden lines, I can actually pick and choose which objects I want to bring in. Reference. And I can also reposition it, so perhaps what I'd like to do is, I'd like to translate it, because I'd like to change the position of it, so maybe what we want to do is, I want to change the origin point, because I want the origin point to be that corner instead.

So, now I want to do is I'm going to create a new sketch, and I'm going to use this new project DWG geometry feature, and I'm going to project these objects. So it's just like the regular project command, but what you're really doing is saying, these are the options that I want to reference. And I'm going to place in a couple of dimensions. I shouldn't have finished my sketch.

Now, the reason that I'm doing this is because, as it stands right now, it won't let you-- it won't let me edit the sketch, but the way it sounds right now is, you can actually reference the AutoCAD dimensions, so I'm just going to place in these reference dimensions so that I have some dimensions to reference when I'm doing my models. So because I placed these in as reference dimensions, again, the AutoCAD geometry changes, updates in here, dimensions update as well.

So, now that I have that, I can do an extrude now, and I'll be able to relate this-- of course I didn't add the dimension I wanted-- but I can relate this to the dimension, and it's all associative. So if I go back to AutoCAD and we do a stretch and we save it. Go back to Inventor. Go Update. And I updated the side that I didn't actually reference. But hopefully, you get an idea of what we can do. So it updates in AutoCAD, and it updates in Inventor.

Let me just switch to a different file. So a quick thing. I've been a big proponent of don't use AutoCAD files in Inventor to create 3D objects. I'm pretty OK with it now, now that we can keep those two things linked together. We can always go back to the AutoCAD file and edit it. And now, I'm going to get that change propagated through into Inventor, keeping my design intent.

Well, now, if you do have AutoCAD geometry and you do want to convert it to actual Inventor objects, Inventor has the ability to open drawing files and convert it. I personally prefer the method of copy and pasting.

So I can come into my AutoCAD drawing. I can copy the objects I want to work with. I can create a new Part file in Inventor, generate my new sketch, and then I can paste those objects. Paste those objects. Oh, I've got to pick where I want them to go. But I can paste those objects.

And at this point what Inventor is doing is Inventor's converting those objects from AutoCAD geometry into actual Inventor lines, arcs, and circles.

A little tip about this is that the Auto Dimension command works really well, not to dimension it,

because Auto Dimension never dimensions it the way you want it to, but if you turn off the Dimension feature, it will look for the obvious constraints, so verticals, horizontals, collinears, perpendiculars, tangents. There's got to be some more.

But it looks for those constraint types that are missing. And it will apply those constraints. So now I went through and applied the parallels, and such like that.

So at this point, there's no linking. So the AutoCAD drawing and the Inventor drawing are not associated whatsoever. But it's a quick way to take that geometry, make it into Inventor geometry, and don't forget about the Auto Dimension feature to apply a lot of the required constraints.

So here I've got a simplified version of the model I was looking at before. And what you're going to find on your File menu is a whole bunch of suite workflows. Not sweet as in like S-W-E-E-T, but suite as in S-U-I-T-E. And you can see that it gives you this ability to push the model into 3ds Max. It allows you push the model into Showcase.

Now, I used to use Showcase quite a bit. I love Showcase. But it's basically gone into maintenance mode. So Autodesk isn't doing anything with it anymore. So I've been trying to do more and more with 3ds Max. Definitely way more power. You can do fuller realistic rendering. You can do animations in it.

And it's actually quite easy just to pick the option, click Run. I'm just going to put it into the existing-- actually, let's put it into the active, because I've got Max running already. And it's going to take this and send it out.

Now, you don't have to use this workflow, because from 3ds Max, you can go to the File menu. You can say Import. Pick the Inventor model. Get a dialog box. Configure it the way you want. And bring it in that way as well. But it does provide this workflow to go back and forth.

Now I'm obviously on the Product Design Suite. I don't know if the Industry Collections have renamed that. Is anybody on the Industry Collection? Is that what they're called? Product Design? No.

**AUDIENCE:** [INAUDIBLE].

**MIKE THOMAS:** Suite, yeah?

**AUDIENCE:** [INAUDIBLE] the collection.

**MIKE THOMAS:** The collection.

**AUDIENCE:** Product Design Suite.

**MIKE THOMAS:** Right. So I don't know if they call the tool any different or if it's still called Suite workflows. I don't know. But that feature will still be there.

I didn't expect it to take this long.

Now, once in Max, you can assign your-- it'll take the cross, the Inventor materials, which is cool. Because one thing that Autodesk did quite a number of years ago is they merged all the material libraries.

So you might have noticed if you ever go to Add or Remove Programs, you'll see the Autodesk Basic Material Library, and Advance Library, and something else. It's because all the products reference the same library. So Inventor and AutoCAD, Max, and Showcase can all reference the same library. So if you apply Stainless Steel in Inventor, well, then Max is just going to say, oh, yeah, that's Stainless Steel, and use it.

Now, you do get within Max access to Max's library of materials, which, in some cases, can be much more advanced, if you want to create that perfect picture. But you can see that it brought the model in. It's grouped right now.

So I have the ability to open that group, and I could work with those components. So I could use the Material Manager. And I could say, well, let's take this metallic red. And I was going to drag and drop it, add it to that component. So I have full access to all of Max's materials. But like I said, though, it has maintained all the materials, because by default, it will look to the Autodesk material for materials regardless of the program it's coming from.

We also have the ability in here now to do animations. So if we want to take some components and we want to drag them and have them do some things, drag this, we'll do a move. It doesn't really matter. Let's take this component. And you can see now that I was able to create that animation fairly quickly.

So Max can be a little bit daunting, because its interface is unique on itself. But if you kind of boil it down and just look for the key things, materials, really just knowing that you have to

ungroup it. And then there's some real basic map modifiers to make your materials look right, and then you render it. So if you kind of break it down and don't get overwhelmed by everything that's in Max-- I mean, you don't have to make the next dancing baby-- if you boil it down, you can get some really kick-ass imagery out of it without really having to put that much effort to it.

So now what I would do is I would just do the render. Now it looks like I picked the material that I'm missing a map for. But you can see now it's going to go through and render that image. So obviously, I'd spend a little bit more time on maybe some lighting and stuff like that. But just to show how easy it is to push that data into Max and create the imagery.

Now, unfortunately Showcase was a lot easier than this, but you can get probably better imagery out of Max than you could or Showcase.

Now, this is a dangerous tool, I should say, because once any salesperson within your organization gets a sniff that you have this ability, you might have a new job description-- will have.

So kind of a-- oh, my Max just crashed.

One of the last areas I wanted to look at here was the point clouds. And I'm assuming that probably you haven't run into point clouds yet. Has anybody had to run in and use any point clouds? Yeah, you've used some point clouds?

So Inventor has the ability to read in these point clouds. So you get ReCap as part of the Suite or the Industry Collection. So you just load in the point cloud. It does what ReCap does. I think it's indexing. I think it's called indexing.

And then what you can do is you can attach point clouds, if you know where you put the files. If like I said you know where you put the files. Do they have an extension type that's [INAUDIBLE]? I think I know what I did.

So I'm not sure how many points are in here, but I'm just going to insert it at the origin. I can rotate it to get it the way I want it. I can play with the density.

So you can see I can specify the maximum amount of points that I want to allow. I can set to scale. What it's going to do now is it's going to attach that point cloud file.

Now, perhaps I'm not actually that interested with everything that's in there. So I do have the ability to crop it. So I can take that point cloud file. I can say, well, I'm really only interested in that area. And it will create a bounding box and remove the rest of the points for me.

I'm able to take a look at-- maybe there were some points that weren't assigned within ReCap, so I could explore those. And I can actually attach multiple point clouds.

Now, with that data attached, what you do is you extract out points or planes. So the whole purpose of it within Inventor is to help you position your objects where they would be in the real world. So perhaps what I want to do is I want to position my assembly at that location, so I'm going to extract out a plane at a point to give me that area that I want to constrain to, that I can bring in my assembly, position it geometrically, or I guess geometrically where it would be in that point cloud. And then either detach the point cloud or just leave it there as reference.

So you can see how we have that ability to work with that. So somewhat limited tools. But it's a somewhat limited workflow with a very specific goal of allowing you to reference that geometry.

Now, if you wanted to do more with the point cloud and the Inventor data, you might want to look at a part called Navisworks. So Navisworks is also included in the Product Design Suite, and the Industrial Collection.

And Navisworks is used more heavily in the AC and construction industry. But when you've got data from a lot of different sources, it can be a really great tool to bring all that data together. So Navisworks will allow you to bring in point cloud data. It allows you to bring in Revit data. It allows you to bring in Inventor data, and really collect it all in one area.

So if I go and take a look at the Append option here and look at the file formats, you can see that Navisworks becomes this viewer. And its goal is viewing data. So it can deal with this data a lot better than, let's say, something like AutoCAD, or something like that, because there's no intention to modify the geometry. The intention is to bring it together, locate it, look for collisions and conflicts, and really mark it up and [INAUDIBLE] in that case.

So I'm going to select that ReCap, that same ReCap file. And then try to remember where I had put that. And then maybe what I'd also do is maybe I want to bring in a Navisworks file. Again, if I can-- I should have put this in not so many different locations. I apologize. I don't know why I can't find it. How come nobody said to change my extension?

Maybe I want to bring in a Navisworks file. And maybe I also want to bring in an Inventor file.

So with a Navisworks, again, you can bring all this data in, collect it, position it, where it's going to be, look for collisions and possible areas of concern. You can mark it up, and then, just like anything else that's associative, so if I go back into Inventor or back into Revit or change the point cloud data, that data will update within Navisworks.

So while we wait for this, any other questions? Anybody else got anything they would like to talk about?

**AUDIENCE:** Well, the collaboration the between Revit and [INAUDIBLE].

**MIKE THOMAS:** Oh, good point. Good question.

**KEVIN** Do you want to talk about that or do you want me?

**ELLINGSON:**

**MIKE THOMAS:** Yes. We have a couple-- there's going to be a couple of products we're not talking about Works going into, or we're not focusing on. But one of those is Revit. Inventor has some really nice features. You don't want to take an Inventor file that has all the level of detail to it. It's too big for Revit. So we have some nice features in Inventor to defeature things to kind of get a more simplified manner.

If you flip me over for a second, I'll just show a couple of these things. I can kind of give you an example what it shows up in Revit. So here I-- I clicked Finish BIM Exchange. So I came--

I've opened this file. Let's say this is a piece of equipment we wanted to bring into Revit. Obviously, this has way too much detail to bring in. Maybe this is like an air conditioner case or something like that that I wanted to show. Once I use some of Inventor's features to defeature this, I can then go to BIM Exchange.

And now important things that I want to do is add like a work coordinate system, so that when I import this into Revit, it knows how to snap to its mater. It comes in the right orientation. So that's something you can do. The Revit integration is pretty good there.

Another thing that-- another product that we're not talking a lot about is Plant Design. So obviously, Inventor's probably good at making things like processing tanks and things like that, and inside of Plant 3D, you can do-- you can route piping runs and things. And we can-- in Inventor, we can author import and export nozzles, connection places, like what the size is going to be where the piece of material connects to.

And then when we bring that into Plant 3D and run a route, just by routing from one piece of equipment to the next, the Plant 3D will recognize the inputs that we gave in Inventor and connect those two at the right tubing sizes, and connections, and fittings.

Any other ones? Questions about data types, product types? We planned for more questions than this.

So Inventor's become really good at-- if you think about all the other products we've talked about, those are all kind of on the outside, and Inventor is kind of the hub. It can create data to many of those programs. It's probably best at creating data out. But it can also take a lot of those inputs and bring them in as a starting point. And it's getting better and better with things like AnyCAD, and drawing underlays, and different functions like that.

**AUDIENCE:** You showed bringing in point clouds. Do you bring in things like structures from Revit to [INAUDIBLE]?

**MIKE THOMAS:** The intention is really to go the other way.

**AUDIENCE:** And that's what you have.

**MIKE THOMAS:** Yeah, the intention is really you would take your machine, what you designed and push it into Revit, and then see how it's going to work inside Revit.

**AUDIENCE:** It does go the other way, though.

**MIKE THOMAS:** Yeah.

**AUDIENCE:** I just come from division, and we actually wrote a tool that takes a section of code [INAUDIBLE].

**MIKE THOMAS:** Yep.

**AUDIENCE:** [INAUDIBLE] program [INAUDIBLE]. Is anything pointing that direction?

Is there an option on the import?

**AUDIENCE:** [INAUDIBLE].

It keeps happening to like [INAUDIBLE].

**MIKE THOMAS:** So there is a Revit file type. Also, I don't think-- like, it depends on how much geometry you have too. I don't know how good the performance would be if you brought like entire complex in there. It's not something I've had a lot of experience with, but it does read the data. And perhaps that will be a future AnyCAD enhancement. A It would make sense to do that as a future AnyCAD enhancement.

Typically, though, it seems Inventor's better going out than it is coming back in.

OK.

**AUDIENCE:** [INAUDIBLE] by you guys. Is Dynamo being integrated into Inventor at all?

**MIKE THOMAS:** I have not heard anything about Dynamo and Inventor, whatsoever.

**AUDIENCE:** Is Dynamo a Revit [INAUDIBLE]? That's kind of what I figured. [INAUDIBLE] objects do anything [INAUDIBLE]?

**MIKE THOMAS:** So what kind of stuff would you be hoping to do with that Dynamo in Inventor?

**AUDIENCE:** Oh, so I don't really have any idea.

**MIKE THOMAS:** Oh.

**AUDIENCE:** We're just kind of-- I guess the idea around Dynamo is being able to extract and use it in a [INAUDIBLE] fashion [INAUDIBLE] spit it back out.

**MIKE THOMAS:** So yes, you're right. Dialogic would probably-- could do some of that for you. Paul, you were going to say something else?

**PAUL:** I was going to say I think they also [INAUDIBLE].

**MIKE THOMAS:** Right.

**AUDIENCE:** My assumption looking at AnyCAD, when you reference a [INAUDIBLE] art file in Assembly, and then check in Vault, it's going to capture that?

**MIKE THOMAS:** Yes. That was one of the enhancements-- one of the few enhancements to Vault 2017 is that it recognizes the reference, and it will bring that Solidworks file into Vault as you check it in. And it will come in-- I think it actually will attach it as an attachment within Vault. So yeah, it will deal with that data.

**AUDIENCE:** Sometimes it's an [INAUDIBLE] attachment and not a--

**MIKE THOMAS:** No, it comes in as a Solidworks file, and it will attach it to the Part file. Yeah. So you'll end up getting two files checked in.

**AUDIENCE:** Once it's in lines, like for a Step file, do you put-- so it's in a Step file, will you put it in the same Vault or other part of Assembly? Will it be checking the Vault and then it would bring that in with it?

**MIKE THOMAS:** Yes. It wouldn't actually matter which folder it was in, as long as it's within your project file. But yeah, it'll bring that STEP file in at the same time.

Yeah.

**AUDIENCE:** [INAUDIBLE].

**MIKE THOMAS:** It might have been size, or not wanting to manage it, or-- yeah, because Vault just treats it as like that model can't really exist without it. So it's going to bring it with it.

**AUDIENCE:** [INAUDIBLE].

**MIKE THOMAS:** OK. Well, we'll be available after, if you guys want to come up and talk after. I'm just going to-- one little spiel here, as a member of the Autodesk University Customer Advisory Council, is everybody in this room next year should consider putting a class in and being a speaker next year. So if you haven't spoke before at Autodesk, it's a great opportunity to showcase your knowledge.

And you might question, Well, I'm not an expert. You're an expert in some field, and people love to hear about how you're using your software. So don't forget next year to apply to speak at AU. And they pay for your conference fee if you get selected. So it's a bonus for being selected.

**KEVIN ELLINGSON:** So it makes it a lot easier to get it approved through work if they don't have to pay for your conference fee.

**MIKE THOMAS:** Can you go back to the PowerPoint and go to the last slide?

**KEVIN** Yeah.

**ELLINGSON:**

**MIKE THOMAS:** So like Kevin said earlier at the class, we built a handout, and-- no, I should go back to your screen-- it's got a lot more content about how you attach point clouds and what you can do with all these certain file formats. We'll update it to make sure we put a link to that plug-in that Kevin used. But the class handout, as it is right now, is up on AU.

If you need additional assistance, the Inventor Help is actually really helpful when you know what you're looking for. So if you know that I want help on point cloud, you just search for point cloud. You'll actually get a lot of useful information about point clouds within Inventor. So I keep saying public clouds, but it's everything.

**KEVIN ELLINGSON:** So one other thing on the Inventor Help. If you don't know how to use a command, but you think it's the command you want to use, start the command, and then push F1 on your keyboard. If you don't see a help symbol, that will take you exactly to the spot in Help that is related to the command that you're using. And then you can read about it, learn more about it, make sure it's something that you're trying-- that you want to use.

**MIKE THOMAS:** And then if you need further help with that, the Autodesk Discussion Forum is also another great resource, especially for this type of thing, because maybe a surface doesn't come across quite the way you expected it, or the cam's not working quite right, the way you expected. There's probably other people who have experienced it. So you can go into the Discussion Forum. So another great area.

**KEVIN ELLINGSON:** Another thing there is. Autodesk has put a lot of-- they have an entire team of people that just work on forum things. And they have a lot of cool tools in there.

For instance, if you post something in the forum and you're kind of a paying customer and nobody replies to your forum posts in about four hours, you're going to get an email that says, hey, we've noticed you've made a forum post. It doesn't look like anybody's helped you. Have they? Is this still an issue? Would you like to create a case from this? And they can automatically create a support case right from that forum posting.

**MIKE THOMAS:** And really, the last thing is please fill out the surveys. There's no excuses. This is not the last class of AU, the second last class of AU. So when you're sitting in your next class, you can fill out the survey for this class. It would be greatly appreciated, whatever feedback you guys give us. And remember that five is the good score.

**KEVIN** Yeah, five.

**ELLINGSON:**

**MIKE THOMAS:** One is not the good score. Five is the good score. So thanks for coming.

[APPLAUSE]

**KEVIN** Thank you.

**ELLINGSON:**