

SACHLENE SINGH: Also, just because I've been working solely with Fusion, very closely, kind of very closely tied to what the future vision for Autodesk is with Fusion you'll see a heavy bias towards Fusion from me, which is why Jamie is my co-presenter, who absolutely is an Inventor fanboy.

And so we're hoping to balance this out. There might be a little bit of throwing of punches. We won't be candid about that. We'll be very-- But we'll be very, very open about it. But it's really a discussion.

So just, with a show of hands, I kind of want to try to understand the audience a little bit. How many Inventor users in the room? That should be every single one of you. Awesome. How many of you have tried Fusion 360?

JAMIE SCHERER: Good.

SACHLENE SINGH: OK. Very cool. So most of you are kind of here to understand how to get started in Fusion 360. Or are you here to kind of understand what the difference is between the two tools are? Differences.

JAMIE SCHERER: Yes. The answer's yes.

SACHLENE SINGH: Awesome. Very cool. So if you have Inventor most likely you're a Product Design Suite customer, user. And so Product Design Suite, on maintenance, gives you access to Fusion 360. I'm jumping ahead of myself.

Pause for a minute. Everybody knows what the class description is, kind of what you're here to learn. We'll Well I'm going to skip over that very quickly. So Product Design Suite, on maintenance, includes Fusion 360. How many of you knew that? OK. So you already have Fusion 360 if you're an Inventor user on maintenance, right?

There's a hand out for this class, which basically goes over, how do you download and access that Fusion 360 that you already have access to? All right? So I wanted to make sure I clear that up. All of you in the room-- and everybody raise their hand for Inventor user-- you all have Fusion 360 today, if you're on maintenance, right?

So what is Fusion 360? So how many of you went to the keynote yesterday with Amar Hanspal? OK. Awesome. So everybody understands what the vision is for a company like

Autodesk?

We clearly see the promise of the cloud. We clearly see where the industry is going. And as somebody that is a competitor in this industry we absolutely want to be on the forefront and be the company that is setting trends. So I kind of want to set the stage with that.

Autodesk has been in the CAD space for a very, very long time. We started with AutoCAD over 30 years ago. And we've been in the 3D CAD space for over 15 years now. When you talk to any 3D CAD user there are certain themes that you hear. Certain things that they love about their tools. But also certain things that are not easy to work with or work around.

And so we went out and we, obviously, have a lot of users, so we spoke to a lot of users. And we tried to understand, what are sort of the barriers to being innovative? What are you seeing out there that prevents you from designing better products, or it takes you longer to design better products? What's holding you back? And what can we do better?

And this is a little bit of a glimpse into what we heard. Reduce design tool overhead. We really don't want a tool that has a huge learning curve associated with it. So a few days to learn the tool, if not a few weeks, and then definitely months to master it. So we want a tool that is launch and fairly easy to get into.

We want to simplify data handoff between departments. How many people here agree with that? You work with an industrial designer. You do mechanical design. All of the sudden you can't talk to each other. You send your designs to a machinist. All of the sudden you have to start talking, what file format? How are you doing this? Where should I put it? And they send you an email, should I put it on Dropbox? Is it the assembly? Is it the drawing? Where did I put the part file. So all of the sudden data is not very easy to share.

And the third huge barrier that we've always had with design tools is, I have Inventor, or SolidWorks, or whatever other tool and I have it installed on my work computer. And I'm here for 9, 10 hours a day. But now I've gone home and I just realized I have an idea that I want to implement or there's a change somewhere and I need to make it, because there's a facility in China that's going to be cutting parts very shortly, et cetera, et cetera.

All of the sudden you really don't have access to be able to do that, right? So we wanted to be able to give access to our users anywhere. We wanted to be bring down the hardware requirements that-- all of your hardware for you to be able to run a legit CAD product.

So listening to all of that user feedback and kind of understanding what our users were talking about is the reason that we came up with Fusion 360. So a lot of that is incorporated. I also spoke a little bit about Autodesk, as a company, needs to stay competitive. And we need to understand what the future looks like in our market to be able to stay competitive.

So we're seeing three basic trends in the future of manufacturing, or the future of making things, as you've probably heard a zillion times in the last three days already. We're seeing means of production changing. We've seen companies being enabled to do much smaller runs.

So startups are all of the sudden a lot more empowered to be able to do much smaller runs, and still be able to make money on selling those products. We've seen the means of production changing. There's additive manufacturing. There are different ways that you can run the same things. There's flexible factories. There are smaller factories.

We see that the products are changing, themselves. So sensors that you typically found in industrial equipment have made their way into our daily lives and the products that we work with on a daily basis, making them smart products or connected products. And that's definitely what the future looks like. And we see that. And we recognize it.

And the way the consumer's consuming the product is changing, as well. So as a consumer, if something was custom made to you would you pay more money for it? If you could say I want this in red in this size, you know, absolutely mapped to-- we've already heard of the headphones that scan the inside of your ear and then are perfectly fitted to the inside of your ear, just giving you a much better quality sound, right? So custom made for you.

Do you want to understand how sustainable the product is? How sustainably it was manufactured? We all want to understand where was it manufactured. And we will pay more money to both understand it, as well as have it comply with where we want it to be manufactured. So we're looking at trends like on shoring. And we're looking at customers being very aware of how things are changing and consuming based on that.

So all of those three factors, Autodesk, kind of, is responding to and making sure that we are in tune with where the industry, and where manufacturing is going, and that we can keep our users very much on the forefront, and help them stay competitive with where the industry's going. So that's why we built Fusion 360.

We did three basic things. And I spoke about this a little bit earlier. So one of the things is real time collaboration. We spoke about not being able to access data, being able to send each other data, when we work with each other. So real time collaboration.

The cloud enables that. All of your data's on the cloud. You access it on your Mac, on your PC, on your mobile device. And you're really being able to get instant access to your data, no matter where you are.

So if you have to approve something, change something, basically, you'll be able to view on your iPad or show somebody what you're working on. It's very, very easy to do. Again, without those heavier hardware requirements that typically a CAD product has demanded. And then the integration is seamless.

We are moving data from an industrial design environment, into a mechanical design environment, into a CAM environment, in an animation environment, into a simulation environment, into a rendering environment, and so many more.

And you don't have to think about what environment you are in and what file format you're exchanging with each other, because all of these things I just mentioned are under one Fusion 360 umbrella. The UI is the same. The experience you have is absolutely the same. And you don't have to worry about, what file format am I working in? Because it's all right there while you're collaborating with the rest of your team.

So let's go over, very quickly, the functionality that's included within the tool. The slide builds up but I'm going to build it out completely. So we started about five years ago when we put the tool out in beta. And there was industrial design and solid modeling. But there's a vision for fusion. And it keeps getting broader.

And you've heard it in the keynote yesterday. You've probably heard it today, as well as in the opening keynote. Fusion is truly an end to end solution. Where you can Ideate, explore your design, build out your design, incorporate electronics, incorporate designs from different CAD products, be able to do something like sheet metal.

Some of these things are coming very, very soon. Be able to cut your parts, whatever needs to be machined, be able to collaborate with other people. Fusion is parametric. So if you're using the parametric capabilities within in Inventor, which I suspect most of you are, you could absolutely do the same thing in Fusion.

There's simulation capabilities. Again, and Jamie and I will talk about how they match up with what's available in Inventor, as well. Drawing's, because what CAD package is complete without drawings. Real time rendering. And when you think about having the power of the cloud behind functionality like this, it really up-levels some of this functionality to the next level.

So think about simulation. We've all run local simulations using different tools. And we've all seen our computers get locked up for a while, a few hours, maybe go get lunch, maybe for the rest of the night. And now think about infinite computing. So sending that off to the cloud to be computed so that you can keep working and wait, maybe, 10 minutes before that comes back.

Think about rendering. Same thing. Each time we run a render we're basically locking up our hardware devices. It's really not the day and age to be able to do that anymore when you have the power of the cloud. So a lot of that, when you put the power of the cloud behind it, it really up-levels it to the next level.

JAMIE SCHERER: So Sachlene and I, we referred to this slide as the elephant in the room, right? And that's probably why you're all here, right? The elephant in the room, you know, is it inventor? Is it Fusion? What's the deal with this Fusion thing? What are you doing with Inventor? When is inventor going to die? Are you killing Inventor, right? If you're not thinking that, you're a liar, and I'm going to ask you to leave.

So let's talk about a little bit, right? Because truly it's not an either/or, per se. And that's really what we wanted you to leave here with. Is that you're an Inventor customer, you're on maintenance. You're a Product Design Suites, or collections customer, you have Fusion, right?

So that's the fundamental thing that I want you to remember, is that at some point in time, and it might be tomorrow when you go back to the office. It might be two years from now, when you're ready, Fusion is going to make sense. And it might make sense, right now, for a couple of things.

And that's what's Sachlene and I are hoping to impart on you, right? The areas where you might be able to supplement what you do in Inventor or maybe, what you don't have the ability to do today that you can do in Fusion. Sachlene, if you jump to the next one.

Let's talk about it a little bit. I don't want you to get too hung up on colors and things, because Sachlene and I battled about this a little bit. Should that be-- We made that yellow. No, Inventor could do that. It's green. You know. But let's talk about some of the advantages,

right? Some of the differences between the two tools.

So you have a good idea of what Fusion might add value to and maybe where you can't use Fusion yet so. Without going through this, maybe, painstakingly, any device. Browser. You'll see today I'm running on my Mac. Sachlene's running on her surface, right? So Fusion works on a Mac, works on a mobile, right? So cross-platform capabilities, which is huge.

Industrial design. Now, this is one we kind of talked about a little bit. Fusion adds a ton of value in the sculpted environment. If you're familiar with the T-Spline technology that Autodesk acquired several years ago, some of that made it into Inventor, right? A few releases ago. It's continuing to be developed. But there's even more functionality over on the Fusion side. So if you like that freeform tool set in inventor, maybe it's worth looking at Fusion, right? There's considerably more tools with T-Splines on the Fusion side.

And you'll see a lot that are the same, right? Solid modeling? Both can do it. Surface modeling? Absolutely. Animation? The answer's yes, right? But then there's some other areas. For instance, sheet metal. Sachlene mentioned it before. You know, it was on-- Steve Hooper talked about it.

It's coming soon, right? So today Fusion does not have sheet metal capability. It's right around the corner. It's going to be here really soon. We'll kind of look into the future a little bit at the very end. But you know, there's some give and take, right?

The last thing that I'll say, kind of, on this side is, you know, in the areas where Fusion might add some capability, so I'll focus on a few of those areas. Sachlene and I are going to show some of them. CAM, right? You're a Product Design Suite customer, you use Inventor? You've got two and a half, and three axis CAM, with Fusion, right? You have it now. You already have it today.

Simulation. Now, if you're an Inventor Pro customer you might be familiar with linear static, modal analysis. You have thermal stress today, in Fusion. So you can be taking advantage of some of these tools. So I'm going to ask a question, because like I said, I want this to be a little bit of a conversation. Anything that sticks out there? Anything you just want to probe on before we shoulder on? Yes?

AUDIENCE: Any plans to support Linux?

JAMIE SCHERER: Linux. I don't know. Do you know.

SACHLENE Unlikely.

SINGH:

JAMIE SCHERER: Yeah. I would probably say the same thing.

SACHLENE It's not enough large enough of a user base for that to be-- once it's in the browser, not

SINGH: entirely sure how that would work. But I have not heard of Linux on the road map.

JAMIE SCHERER: Another one right here.

AUDIENCE: How does it work with Vault?

JAMIE SCHERER: Good question. So how does Fusion work with Vault? Doesn't today. Fusion has its own set of data management collaboration, you know, kind of file management capabilities separate from Vault. So really good question.

SACHLENE I'm going to elaborate on that, just a little bit. When you work with Fusion your CAD files are

SINGH: not local on your computer. They can be. You can download them, if you want. But they are managed on the cloud. And you access them from within the tool.

But you can also access them in a browser, where you can you view the file, you can view the renderings, et cetera. And versions are automatically managed. So every time you make a save, or if you set it to be a 15 minute save, those versions are automatically managed.

The need for a Vault came from a need to collaborate, one, with a CAD tool like Inventor. And also the need to be able to manage versions and revisions et cetera. So the need to collaborate is taken care of by the cloud. Taken care of by the fact that you have a project. You can invite other collaborators. They can look at the same designs that you're working in and be able to access them that way.

The need for data management, the versioning, and the revisioning some of it is already existent in Fusion today. But a lot of it is coming very, very soon. And again, it's going to be cloud based. It's going to be accessible anywhere. So when you talk about Vault, that's going to strictly remain only for your desktop CAD tools. The Fusion files will not be managed in the Vault. Unless you want to do them manually.

JAMIE SCHERER: We'll take these two more and then I promise we'll have time for questions in the end too, in

the back.

AUDIENCE: So you said that this doesn't support Vault. Isn't Vault moving to the cloud? And then in addition to that question, does this work with PLM 360?

SACHLENE That's a fantastic question. Autodesk is building a cloud product data management solution.

SINGH: The answer to that is, yes. So the capabilities I spoke about where you have versioning, revisioning, data management capabilities in the cloud, that's what we're building out, coming very soon. So the answer to that is, yes. We're building data management in the cloud, which would be the equivalent of Vault.

The second question is, does it work with PLM? I'll actually show that in a little bit of a demo in a little bit. Also, coming very soon, Fusion will seamlessly work with PLM. So that enables incredible workflows. when you think about it. So you're modeling your CAD file. Right within the team hub that you have you have your CAD file be connected to your PLM.

Have a build of materials that is associated back to your CAD file, that understands relationships. We're half the way there. And I'll show you that. But that connection is coming absolutely very, very soon. And those are easy connections to make. Again, both cloud tools. And that's the path that we see ourselves on.

JAMIE SCHERER: So the last thing I'll add of your question, if you're familiar with PLM 360, if you've been kind of following that product as it's evolved, it's been re-branded, right? It's called Fusion Lifecycle and not by accident, right? So it's going to be a critical part of that Fusion platform that Steve was talking about on Wednesday morning. So absolutely, you're going to see a lot of integration there. That's the last one, and then we'll come back at the end.

AUDIENCE: For cloud data, is there a cap for how much you can upload? Or how much [INAUDIBLE]?

JAMIE SCHERER: Still not right.

SACHLENE We don't have a model on that yet, on the cloud data management, just because the product isn't out yet. We don't have a limit on Fusion CAD data. So this is really where your data lives, right? The data management is only managing what version, revision, et cetera, you're on. So I want to say the answer to that is no, because we already built the tool that creates the data on the cloud and there's no limit to that. That's a great question.

JAMIE SCHERER: All right. So keep those in mind. Again, we'll have some more time in the end. But Sachlene

and I also want to get in product, and show you a few things, as well. So jump to the next one. All right, so number one. Collaboration and live review. I'm going to switch over to my machine. So what I want to show you guys is a little bit about the collaboration tools inside a Fusion 360.

So if you've never seen Fusion yet, which you're looking at, is Fusion 360. And you'll see, you know, if I have open, this product design, right? A reciprocating saw, as an example. So a couple of things that I want to show, and I'll impart on you guys is, you know, as Sachlene started to say before, what Fusion enables is instant collaboration, right?

The data's already stored on the cloud. So collaborating with people inside of the four walls of your organization, or out with your vendors, or your customers happens with, you know, a click of a button. So you know, you're looking at this design that I have here in Fusion. And you'll notice that I have a project, my reciprocating saw design. It has a bunch of data inside of it.

And it also has a bunch of people that I've invited to collaborate with in my project. Now, those people have the ability to view my files, mark them up, add their own, et cetera, right? So that is an integral part of the way that Fusion works, is that we can unlock, and we can invite different stakeholders, collaborators just by inviting them into this project.

But what I want to show you is if I jump out onto the web you'll see that the kind of backbone of Fusion is Fusion Team. And Fusion Team is built on the A360 Platform. And what Fusion team gives you is the ability to, you know, not only manage your data.

So this is going to give you the ability to communicate, collaborate, and mark up your files. You'll see on the right hand side over here I have a list of comments, right? Markups that were created, URLs, or, you know, comments that came from my peers, or my boss, or something like that.

Fusion will also manage references, as an example. So it will show me any data that is associated to a file, right? This design uses these 20 parts. And it's used in these other 10 assemblies.

But another part that's kind of integral in the backbone of Fusion, and a, you know, benefit of being born in the cloud is that viewing, sharing this data is instant, right? It's a part of it. So you'll see right from that same UI, right? Right from the web browser, I was able to jump off into a viewable, right?

So now think about how we collaborate with our customers, our vendors today, right? And I work with all kinds of customers. SolidWorks customers, Inventor customers, Pro/Engineer, doesn't matter. You might be exporting, right? Saving a step file.

I worked with one company that told me that they export five different formats to send it to their tooling vendors, because they don't know, do they have this same version of SolidWorks? Can they import this file? Did they have the right viewer?

All of that goes away when you use Fusion. So you'll see with a click of a button I'm instantly inside of the viewer, right? This can be viewed by anybody. Even people without Fusion, right? All you have to do is, here's the Share button, and send them a hyperlink, right? It's as easy as sending a hyperlink.

So now if I wanted to get feedback, Sachlene is my tooling vendor. I want to send her this file and find out what it costs to make a mold for it. I could just copy this URL. I could email it directly from within A360, or Fusion Team, send her that link.

What Sachlene would get is a simple little URL that I guess I didn't copy. So we'll try that again. Sachlene will get a URL. That will take her automatically to that viewable. So no need for, you know, a viewer. No design review. No nothing, right? All you need is a URL and you're viewing that data.

Right. So within this view you could mark it up. You could actually access the browser, the model tree, right? Turn on, turn off parts. But here's one other thing that I want to show you that I think is really awesome.

And this, again, could be a benefit of using Fusion today purely for collaboration. If you're using Inventor you could have maybe some of your components, some of the things that you need to send out to another customer.

You load them into Fusion Team. And then you can also allow a customer, vendor, one of your colleagues to serve themselves and download the data in the format that they want.

So what you missed on the other screen was I created that URL. I sent it to Sachlene. And I also turned on a functionality that allows her to download the file, right? So in one case, I want to send it to you guys. I want you all to view it. I don't want you to have the data, right? And in the other case, yes. I want you to not only view it. But I want you to download it.

And I want you to download it in any one of those formats, whichever one you care about so that I don't have to send it to you, send you a new file, tell me you need a STEP. STEP didn't work. Can you send me an IGES? No. I don't want to keep doing this all day, right? So here's a really, really nice benefit of the collaboration side of Fusion. Yes?

AUDIENCE: Does it update when the model changes? Do you have to send another URL?

JAMIE SCHERER: A wonderful question. The answer is, it does update. So that URL will stay connected to the latest and greatest. So I used to work for a manufacturer here in the US, and you know, I can remember sending someone a file. I need your feedback.

They didn't look at it for two weeks, right? Two weeks from now that data's no good, right? It's old data. I had to send you another STEP file. This will stay connected to the latest and greatest.

The other thing that you can do is at any point in time I could jump back into that kind of Fusion Team experience. And I could turn this off. And now, it's not accessible, right?

So if there's a point in time where I no longer want you to have that data, I turn it off. If I turn it back on it's now a unique URL. So the old link doesn't work. So I sent that to somebody else. I don't want them to see it. I send them in a new file, right? So that's kind of a way that you can manage that.

So the next thing on the collaboration and live review topic is a tool inside of Fusion called, Live Review. So what Live Review allows us to do is jump into a collaboration screen share session. So from right inside of Fusion I can start a new Live Review session.

Now, when I start a Live Review session what I can do is, just like I did before, send a URL. I can send this URL to, you know, any number of people that I want to collaborate with. So you'll see I'll just open up my email. I'm going to send this to myself.

And we're going to go split screen mode here. And I want to show you Fusion on one side and a browser on the other. So here's the benefit of using the Live Review or the collaboration tools inside Fusion. I can take that URL, send it to, you know, again, my customer, my vendor, whoever I want to collaborate with.

Now, just by sending that URL that person would get invited to this Live Review session. You'll see I'm going to join this Live Review Session as Jamie on the web. On the Fusion Side you'll

see there's two people in this session so far, right? Me and me, right? Me in Fusion. Me on a browser.

So what's awesome about this is, if you're watching left and right, while I'm in Fusion , and maybe we're on the phone, we're talking, I'm telling you, hey, you know, I've created this new housing for the saw. What do you think? As I use, as I move Fusion, you see it update, move, pan, rotate, inside of the browser.

The other thing that's awesome, if I were to make changes, right? If I were to go through design changes. Or you said, hey, that fillet's too small. I can't machine that. I change it. You see it in the browser. It happens in real time.

The other part that's really neat is, David, this is like a magic trick. This is my favorite thing to do. Have we ever met before? OK. Good. So now you guys know that I'm not a liar. And I didn't plant David in the crowd.

So accessible on a tablet, as well. So what I could do is David's my manager, he's out in the field, right? We're going to invite David to our Live Review session.

You're now my magician's assistant. So again, let's see what happens. So now, as I showed you, as I use Fusion, someone looking on a browser sees it. Someone out in the field on a mobile device sees it, right? David, you can prove it as I zoom in, zoom out, you're seeing it.

So now, from Fusion, what I can do is first I'll make Jamie on the web the presenter, right? So as soon as I switch that, over here in the browser I gave my customer control.

So now they can zoom in, zoom out, access the browser, you know, maybe I don't know, I'll isolate the one side of the saw. The could interrogate, kind of dive into the data. And anything that I do here in the browser happens in my Fusion session.

Just the same if I were to give the controls over to David, hands off the wheel. Now David can be out in the field. He can view, mark up, access the data, right? So anything that he does on the iPad, I'm seeing, although I've kind of ran out of screen real estate, so I'll move this out of the way.

SACHLENE

I know some of you would applaud at that.

SINGH:

JAMIE SCHERER: Yeah, you can high five.

SACHLENE At least some of you.

SINGH:

JAMIE SCHERER: It's late. It's long.

SACHLENE That's slick. Isn't that awesome?

SINGH:

JAMIE SCHERER: Cool, right? So just the fact that you're here is applause enough. You don't have to. Well, you can applaud as much as you want. But, right? So I mean, it is that easy, as I zoom in, zoom out, right? That is happening in real time on the browser and also inside of my Fusion session. Yes?

AUDIENCE: Is there voice involved in that?

JAMIE SCHERER: There's not. Yeah, we'll figure that out soon.

SACHLENE Coming soon.

SINGH:

JAMIE SCHERER: Not yet. Soon.

SACHLENE Voice and video is coming soon.

SINGH:

JAMIE SCHERER: So in terms, you know, of collaboration, you know, this is a set of tools that you could take advantage of today. Inventor has some capabilities, right? I mean you can connect your data. You can host it inside of A360 Drive. Some of this builds on that, right? And maybe there's some little nuggets that you can take advantage of today, purely by just using the collaboration and the Live Review functionality in Fusion.

SACHLENE How many people in the room have used A360 Drive?

SINGH:

JAMIE SCHERER: Good. A Couple. Awesome.

SACHLENE You win.

SINGH:

JAMIE SCHERER: Well, yeah. See?

SACHLENE

I really didn't think there would be any hands. So the idea is taking collaboration to the next level. Really being able to share your designs. Really being able to work with other people.

SINGH:

And make it easier. Encourage you to work with other people, that, traditionally, any CAD tool has been unable to be able to do.

So we set out with the goal of pointing out how Fusion adds Value. How Fusion, that you already have, adds value to your existing workflows. I will say this, just for this piece, that even if you use Fusion just as a translation engine-- so the next time somebody asks you for a CAD file, show me your designs, send me your design, and you say, what file format do I need to send you? And he says, can you do, SolidWorks? You're like, can't. Yeah, tell him--

Don't have that conversation. Send them a URL. And let them figure it out. Save yourself some time. Even if you just use, and I told you, Fusion's capable of 20 things on a slide before this. But even if you just use it as a translation engine you're already adding value to your existing workflows, right?

OK. We stopped for questions in between. And I'm thoroughly disappointed that nobody asked this one. I'm going to let it pass because it's 3 o'clock on Thursday. But nobody asked us, how am I going to bring my Inventor data into Fusion?

AUDIENCE:

[INAUDIBLE]

[LAUGHING]

SACHLENE

That doesn't count.

SINGH:

JAMIE SCHERER: That's the true elephant in the room.

SACHLENE

No, I'm just kidding. All right. So let's talk about that. How do you bring your inventor data into Fusion? So I just said, use it as a collaboration engine. We want to make that really, really easy, right?

SINGH:

We want you to be able to use both of these tools exactly where you see them be the right fit. So we want to make this really, really easy for you to do. And this is going to get better.

So first thing I'm going to say is this is technology that's coming soon in the future. Very, very soon. And that's how you'll be able to do this. So let me go into this and show you. So there is a technology called Autodesk Drive, that is a virtual drive that you download, and will soon come with your Fusion download, which you download onto your computer.

So think of it as a Dropbox that is CAD aware, OK? So all of the sudden it understands all of your assembly part drawing file relationships. How it's different from Dropbox is that it's a virtual drive. Those files are not, in fact, occupying that space on your computer. It's really just an image of that file, right? And access to that information.

So you install Autodesk Drive on your computer. And you basically sync it up. It sinks up to your hub when you use your Autodesk ID to log in. So I'm in a few different hubs. I'm going to go to the one that I use most of the time. And these are all the projects that I have been working on. I started. I am the owner on. All I did was download Autodesk Drive and sign in with my Autodesk ID. All of my projects are all of the sudden available on my computer. So it's synced up.

The next thing I'm going to do is I'm going to go into a specific project. And what I have here, as well, is an inventor designed dataset. It's a planetary gearbox. So I brought it in already before. Let me do, quickly, a new folder. Let's call it Gearbox 2. And what I'm going to do is, once I'm in that folder-- again,

I'm doing this in Windows Explorer. Once I'm in that folder I'm going to grab the assembly file, let's say, and drag it over. All on my computer. Windows Explorer. And it's going to ask me, what's your project file? Where all Inventor users here. We're all aware that Project file lives-- yes?

AUDIENCE: And this will recognize all relationships?

SACHLENE SINGH: Yes. Yes. So all I did was grab the assembly file, drag it over, it said pick your project, and that helps. So go ahead and say, OK. And then it's going to chug through and put those files in my Autodesk Drive. Now, again, we're using the cloud. So give it just a little while to sync up.

But let's go back into Fusion. And this is the project that I went to in Windows Explorer. So it says, data in this folder has been updated. Let's refresh. Otherwise I could have hit this little spinning wheel up here to refresh. Look at that. The folder's in there.

And I'm using conference center internet. And once it's in there, again, there's the assembly

file. It's taking just a minute to generate the thumbnails. But all of the part files got dragged in, as well.

JAMIE SCHERER: Ow!

AUDIENCE: Wow.

JAMIE SCHERER: It is cool.

SACHLENE SINGH: So how do you bring your Inventor data into Fusion? It's Autodesk Drive. It will very soon, in just a couple of months be downloadable with Inventor.

Does anybody in the room have Fusion Lifecycle, or formerly PLM 360? So currently that's the only tool that Autodesk Drive is downloadable from. So you are welcome to try it out. But the download link will be available in Fusion very, very soon.

So the next thing I do is I right click on the assembly file. And I say, create a fusion design. And if you've ever been through the upload process of uploading a file into Fusion, that basically means you're uploading it to the cloud. And it's understanding relationships, geometry, et cetera.

What this does is it goes through the upload process. This is the longest step in this process. Everything else, the syncing, was absolutely seamless. So this is the longest step. It's going to go through and upload it, and convert it into a Fusion file format.

While it's doing that, what I'm going to do is I'm going to go look at this assembly on the internet. So Jamie was showing you accessing your data in the browser. And it's absolutely available there. So I synced up in Windows Explorer. I just did this in front of you. And this assembly exists in the cloud it.

You can access it in your browser. You can access that information. You can send that URL, that Jamie you just showed, to your suppliers or whoever else you need to send it to. And then it understands relationships.

So if I went to one of the components within the planetary gearbox, all of the sudden it understands where it was used, references the assembly. Again it's just taking a minute to generate the thumbnail. And if there were any drawings associated with these components, et cetera, it would understand those as well.

JAMIE SCHERER: Pretty cool, right?

AUDIENCE: So you said it's converting that over to a Fusion format file? So if you upload it to this, and then want to bring it back to Inventor, do you have to re-translate it in Inventor?

SACHLENE SINGH: You have to re-translate it into an Inventor file, which is a very important point. And I want to mention this here, just because they don't want any of you to have any false expectations. That means that it's not associative, because all of the sudden you're going through two translation steps, right?

So while your data syncs-- so a really neat workaround to this would be to have your project file just live in the Autodesk Drive folder, right? It's an interesting workaround. So if you just had your project file live here. And this is where you access the Inventor data from, when you used Inventor, and just saved it to here. When you wanted to go to Fusion that would update all of this Inventor information. So it would save you the step of dragging it into Autodesk Drive because it already exists. It would update all of the assembly and part information. But you would still have to convert it to Fusion. But it would just already be there. All you have to do is convert it.

AUDIENCE: Is there any plan-- I mean, I [INAUDIBLE] about the AnyCAD.

JAMIE SCHERER: So I was just going to add that.

AUDIENCE: Is this going to be incorporated with that?

JAMIE SCHERER: So as you can imagine, the short answer is yes. It's definitely on the road map. And you know, so the common question, just like you asked, right? Is great, I want my features in Fusion. And I want my Fusion features to go to Inventor. That's not in the near term. But what you mentioned, like so Inventor has AnyCAD functionality. I was working with the Inventor team the other day, and Fusion is on the roadmap to be able to do that.

AUDIENCE: Well, I mean before it was 360, you know, it was the Inventor Fusion.

JAMIE SCHERER: That's right.

AUDIENCE: And so when you would go in and explore that in Fusion you would basically just create a separate feature in that feature tree that you can't do anything with Inventor, obviously. I mean, do you see it being something like that?

JAMIE SCHERER: In the nearest term what I think you can expect is to see, if I reference Inventor geometry into Fusion I'd be able to reference it.

And if it changed in Inventor it would change in Fusion, and vice versa, like AnyCAD. That's what I think you'll see in the near-term. Longer than that, I'd be like you, hoping and guessing. But I don't know. I don't think that's in the anytime soon category. Yes?

AUDIENCE: [INAUDIBLE]

JAMIE SCHERER: For Fusion, to say it's on the roadmap is yes. But not in the near term, right?

SACHLENE Yes, yes, yes.

SINGH:

AUDIENCE: Is there a Fusion API?

SACHLENE Yes.

SINGH:

JAMIE SCHERER: Yep.

SACHLENE Just to complete the PLM question that was asked earlier, you would be able to then, just from
SINGH: uploading your CAD model, be able to convert it into building materials that would live in PLM, manage your change, and all of that good stuff there.

So this, in the browser, what you're looking at now, would be mapped to your building materials, to any new project, new product, introduction, et cetera, you're doing in there. And that would complete that workflow. Is there a question?

AUDIENCE: What happens to my add-ins that I have accrued over the years?

JAMIE SCHERER: Add-ins ins in Inventor?

AUDIENCE: Yeah.

JAMIE SCHERER: Be more specific. So-- yeah, be more specific. How do you mean?

AUDIENCE: Well, I've created tools for my CAD people. [INAUDIBLE].

JAMIE SCHERER: Be faster, better designers. Yeah.

AUDIENCE: And if they start creating in Fusion, is there a way to translate my add-ins into Fusion?

SACHLENE I'm going to take that one really quickly.

SINGH:

AUDIENCE: Do I have to start from scratch?

SACHLENE So it depends what the add-in does. So I've worked with a few customers that are doing add-

SINGH: ins just so that your drawing template always looks a certain way. And you're only allowed certain standards on your drawing. Or only certain library components are accessible for certain designs. So things like that. If that's the custom add-ins that you've done, those will be very much available to do in Fusion.

The same gentleman that wrote the Inventor API has done the Fusion API. The code will be migratable, not in a very direct sense. But you can literally look at your code, bring it into Fusion, maybe tweak it just a little bit, and go from there. So if it's an add-in you wrote, you should be able to replicate it in Fusion with a little bit of work. If it is an add-in that exists in Inventor today, that is offered by a third party company, because we obviously have a lot of partners doing that. We are encouraging all of them. And they sort of see the future of where Autodesk is going to replicate those items in Fusion, as well. Fantastic.

JAMIE SCHERER: All right. So we are like 16 minutes away from beer, or wine, or drink, or whatever your persuasion is. But we'd be remiss if we didn't talk a little bit about CAM. So again, I mentioned it before, if you're an Inventor customer, Product Design Suite, collections, right? You have CAM, right? This is an entitlement you have through Fusion.

And I think there was a slide that I should have shown first, but we're going to get right to it. Fusion 360 has two and a half, three axis machining capability. So right within Fusion, in the top corner is what we call the workspaces.

It's as easy as switching from which, would be your parametric solid part of the world, to Cam. And then once you're in the Cam environment, or workspace, you have the ability to start creating tool paths, and export that, obviously, as G-code to your machine.

So I'm going to go through a really, really quick example. And Layton's here to make me nervous, as a CAM guy, because he's going to pick apart my CAM capabilities. Well, let's go ahead and let's make some tool pass, quickly.

So what I'm to do is start a new setup. And inside of Fusion, you know, what I want to do is reference or create tool paths for machining one of these components. So the component that I want to machine-- Oh, come on, projector. We're going to close her. Hold on. Sorry guys. One second.

SACHLENE How many of you anticipate using or need to create tool paths, and anticipate using machining
SINGH: capability?

JAMIE SCHERER: Awesome.

SACHLENE That's incredible.
SINGH:

JAMIE SCHERER: Boom. OK. There we go. So back to some really bizarre panning things that are happening right now.

AUDIENCE: It's because you're on Mac.

[LAUGHING]

JAMIE SCHERER: Touche. Touche. Maybe. No. Come on. Man, Sachlene, you saw me do this--

SACHLENE Yeah.
SINGH:

JAMIE SCHERER: Just a minute ago, right?

SACHLENE It's selecting the wrong part for stock.
SINGH:

JAMIE SCHERER: Yeah, and it's stuck in like pan mode, which is kind of bizarro.

SACHLENE Do you want to try to relaunch Fusion maybe?
SINGH:

JAMIE SCHERER: Yeah, that's what we'll do. Sorry, guys. One second.

SACHLENE Once you're connected to the projector it helps to relaunch.
SINGH:

JAMIE SCHERER: This has been a long week. I'll make an excuse. Let me try that again.

AUDIENCE: One of the major hurdles that I always see, cause I work at a manufacturing facility, and the re-use of data, the re-use of parts, I mean, for new designs. Parts are made that have nothing to do with that, except for the fact that, hey? This plate that I used 5 years ago? I want to use the exact same plate here, same part number, everything. How can you do that in Fusion?

JAMIE SCHERER: So is the question, how do I do that with my Inventor plates that I have today? Or how do you do that if you were to create a new plate in Fusion you want to use in the future.

AUDIENCE: Say that I created it in Fusion.

SACHLENE I'm going to get the question, while you--

SINGH:

JAMIE SCHERER: Go ahead, yeah. I appreciate it.

SACHLENE You have, let's say, project A in Fusion, right? And you've invited five colleagues to it. Those
SINGH: are the people that contribute. Typically projects are organized in a few different ways. Maybe you organize it by customer and there's five things that you made, products that you made for that customer, right?

Now, all of the sudden you say, well, I have customer B that needs one component from the second design I did for customer A. You could simply copy how many ever components you need from one project to another. That data moves within projects as long as you are the person moving it, and you understand what the permissions of both of those projects are.

AUDIENCE: When you're copying it, you're not using the same [INAUDIBLE].

SACHLENE So very good question. If it's a different project, so customer A, Customer B, and you want to
SINGH: reuse some of this. And the way you're set up is a different project, then you're not re-using. If the way you're set up is it's the same project, but it's different folders, then you absolutely can re-use, and have it be associated back. Now, if at some point you want to break the link of that association, you're welcome to do that too, even within the same project. You have that flexibility. So you have to think a little bit about, how am I using these designs?

And what are they used for? So a lot of times you actually do want to break that link. Or you do want to take it to a separate project. A lot of times you'll have it be in the same. So different

ways to solve that problem.

JAMIE SCHERER: And that'll get better soon in Fusion, right? There's more functionality coming. All right. So now I'm ready. So I've you know, I've got my part. I want to start creating Toolpaths. I want to machine this.

And you see I've selected my part and located, you know, my zero, right? I've used the corner of my stock. I'll get that corner of the part. I can even go as far as, you know, modeling a custom stock, right? If you know, we were going to water jet a shape and that's what I was going to machine it from I could absolutely do that.

So with this new set up I'm going to create two Toolpaths really quickly, in the interest of time. And you'll see, in Fusion we have a list of 2D and 3D strategies for creating Toolpaths. So what I'll do first is, you know, let's just go ahead and create a facing Toolpath, right? I want to take a few millimeters off the top of my stock.

You'll see Fusion comes out of the box with a huge library of tools and the ability to create new tools, right? So if you're a machinist and you're in the room, or if you're working with the machinists back at your office, you know, what we'd want to do is set up your tool library, right? Number them, name them. And then use those, obviously, to create our Toolpaths.

So I'm going to grab my face mill, and say OK. You'll see that in my library, by default, is all of the speeds and feeds that, you know, we've dialed in over a number of years that we like to use with that tool. And I can absolutely adjust them, right? If I wanted to run this at a different spindle speed today. You know, it's all, obviously, here to be adjusted.

I won't go through all of the little tabs. But if you're going to go home and try this out, I'd say the best advice that I was ever given using CAM and Fusion is work left to right, right? And it works. It's pretty logical, right?

I'm going to create my Toolpaths. And even through every path that I'm creating. I'm going to start off with the tool, right? Speeds and feeds like I showed you. What's the geometry that I'm selecting?

Now, I might want to look at clearance height, re-track those type of things. I can start to look at different es and ultimately linking my paths together. But I'm just going to say OK in the interest of time. And you'll see I've created one, right? I've got my facing Toolpath.

And the next thing that I'll do is I'm going to use 3D adaptive clearing inside of Fusion. If you're not familiar with what adaptive clearing is in the HSM Works, or Inventor HSM World, or in Fusion, FYI it's the same tech, right? In all of them. What this allows me to do is, you know, it's a roughing strategy to optimize my tool load, right?

So I'm going to use a adaptive clearing strategy. I'm going to come in here and find, you know, maybe a quarter inch flat end mill. We'll just search through my library. There she is. And again, in the interest of time, I'm going accept the defaults. Those are, you know, the spindle speed, the speeds and feeds that I would typically use.

And you'll see in a matter of seconds even a dummy like me, who's not a machinist, can create Toolpaths, right? So what I've created in just a matter of seconds is, you know, maybe your roughing strategy for this part. And what I'll do now is just see-- if we go into the simulate tools I can now look through.

And we'll put away all the paths and we'll just look at the latest greatest. But I can simulate and go through the Toolpaths that I created, right? So now I might be looking for, you know, is this the right strategy? Am I going to have any interference issues.

One thing that I didn't do before was include any of my fixturing in the selection. There was a dialog that I skipped through but would allow me to add in all of my fixturing so I know that I won't have any interferences there.

So you'll see as this goes through its strategy-- and we'll speed this up just a hair, cause it certainly looks cool when it starts to fly. But now I'm going through and, you know, I'm looking at the roughing strategy for this component. Now, what does this look like-- we'll let it finish first.

AUDIENCE: [INAUDIBLE]

JAMIE SCHERER: Absolutely. Yeah. 3D printing is also a part of the tool set. So you know, what does this look like if I were to maybe take a little bit more time? You'll see in the kind of Toolpaths that are already created, you know, I might have created, you know, one set up to do, you know, the A side of this part. And another set up to do the underside.

And you'll see, as I go through here, now I've got, maybe a facing strategy. I've got two different roughing strategies for the pockets, right? I'm going to run a contour around the outer edge. I'm going to bore out the holes, drill, and tap, right? All in separate operations.

And then, you know, if you want to look at maybe a more robust, complete example, with the stock on-- we'll just let that fly, because it's fun. This is the capability that you have today, soon as you go home. If you haven't tried it yet, right?

So from here the more important, or the next important thing to do would be post-process, right? So I can obviously send this directly, you know, to create g-code. If you're not familiar, you're probably all familiar by now. Autodesk is leading the industry in the CAM world.

So when it comes to post-processors there's a huge library. If you just Google, you know, Autodesk CAM post-processors you can kind of look through those that are out there. And they're all accessible to you. Yes?

AUDIENCE: Can you add your own?

JAMIE SCHERER: Say that again.

AUDIENCE: Can you add [INAUDIBLE] post-processor?

JAMIE SCHERER: Can I add a--

SACHLENE You own post-processor?

SINGH:

JAMIE SCHERER: Oh, yeah. Absolutely. Yes. And you know, when I talk with customers, will these posts work out of the box? Absolutely. Will they work on your machine out of the box, I don't know.

So in that case, yeah you'd have to maybe go in there and make some tweaks. I'm going to go ahead and post with the generic costs processor. And we'll just save this locally for a second. And In a matter of seconds, in brackets, there's my g-code, right?

Yeah you absolutely can edit those posts. And you might have to, depending on it. We have a great relationship with Haas. The Haas generic post is pretty impressive. And I've worked with a customer, and he's like, are you sure? Try it. And it worked, you know, the first time, for him on a super mini mill haas machine. So back to and now we're--

SACHLENE What's that?

SINGH:

AUDIENCE: Can you [INAUDIBLE]?

SACHLENE There's a lot of different partnerships. So there's Haas, there's Tormach, there's Othermill. So

SINGH: I mean, you could scale up.

JAMIE SCHERER: So now in super fast forward mode, because we've got about five minutes, so what do you have today? The four that you see on the top, additive manufacturing, which this gentleman asked about. Turning, water jet, laser, plasma, two and a half, and three axis are in Fusion 360.

So that's a part of the entitlement for all the Inventor customers that we talked to before. And what just was announced was Fusion 360 Ultimate, which will be a subscription offering, which has probing, three plus two, four, and five axis capabilities.

SACHLENE Thank you. So there's two tiers of Fusion. The top tier was just announced two weeks ago.

SINGH: The Fusion 360 Tier, which is the lower tier is what all of you should have access. The general rule of thumb is 80% of our users need the basic tier Fusion.

And then there's Fusion Ultimate. So the 20%, that's the outlier that needs the five axis simultaneous, that need some of that capability, would be using-- and that's true across the board for simulation, for CAM, for a lot of the other capabilities that are in Ultimate.

JAMIE SCHERER: So there's a couple of more things that we just want to introduce you to, see you know when you go home that you have these capabilities. Simulation's another great example. So if you're an Inventor customer. Not an Inventor Pro customer, this is a great benefit, right? If you're an Inventor pro customer, there's even more benefit that you'll get out of Fusion.

So just like before, the four that you see on top, linear stress, modal, thermal, and thermal stress are all in Fusion 360. So if you have Inventor you have thermal, and thermal stress today, right? If you have Inventor Pro that's what you wouldn't have in Inventor. What?

Back to the Fusion Ultimate topic, the bottom four that you see are in Fusion Ultimate. So you know, any one of those bottom four, right? Buckling, event simulation, non-linear, are \$20,000, \$30,000 offerings, right? Out there in the FEA world. They're inside of Fusion Ultimate.

So again, think of that as another benefit to, you know, Fusion maybe in the future. If you need that capability, it's inside of Fusion Ultimate. But even today, linear, modal, thermal, and thermal stress. You guys already have it as an entitlement.

SACHLENE Add to that, cloud solving so you're not--

SINGH:

JAMIE SCHERER: Thank you. Yes?

AUDIENCE: That's like a hell of a simulation mechanical figure.

SACHLENE Yep.

SINGH:

JAMIE SCHERER: Yep. And it's the Nastran Solver that is doing it. Yeah. So huge value. You know, at the value side, or the cost-- I talk with brand new customers all the time that look at Fusion and say,

I mean, any one of the things that we've talked about so far, whether it was five axis machining, or three axis machining, or non-linear, or event simulation, right? I mean they're all huge value adds.

And you know, the ones that we've talked about so far you have access to. And some of the more advanced manufacturing or advanced simulation pieces are just announced in that Fusion 360 Ultimate tier.

SACHLENE Up Yep. These are independently--

SINGH:

AUDIENCE: [INAUDIBLE]

JAMIE SCHERER: Say that again?

AUDIENCE: Included in our subscription today?

JAMIE SCHERER: Included in your subscription today is Fusion 360.

SACHLENE Yep. So that's the top four, here. Stimulation, mechanical. Yes. It's in the logs. And each of
SINGH: these products standing independently as the desktop version is \$10,000. You're paying nothing for it. Zero dollars.

Just to go over the price point for Fusion Ultimate. We announced that two weeks ago at--
Correct me, Jamie, is it \$1500?

JAMIE SCHERER: Yes.

SACHLENE Annually. So that's \$1500 per user, per year. And that's ultimate talk to your functionality. So
SINGH: five axis simultaneous. All of that simulation that you just saw. And so much more.

Do you want to go through this video? Should we? How many people here are interested in simulation? Is it OK if we fly by this video? Do you kind of understand what the capability is within the tool? We have a few more things to cover. Thank you.

JAMIE SCHERER: Yeah we're going to jump ahead.

SACHLENE Rendering. I'm not going to do the demo. I think we have like three minutes left. So rendering
SINGH: is another huge capability available within Fusion. I already touched on the point that add to that, that you can cloud render, not use your local resources, and all of the sudden, huge, huge benefit, right?

I want to point out that I've mentioned cloud solve for Fusion. And cloud render for Fusion. With your subscription offering that you have Fusion with maintenance you have 100 Cloud Credits. As you use cloud capabilities you burn Cloud Credits.

And so if you run the simulation, depending on the size of the simulation, the size of the render, Fusion is going to tell you before you run it how many Cloud Credits you're going to use. So 100 will probably last you about 20 really good renderings. Probably the same number of decent sized simulations. Just to give you an idea.

So that's a pay as you go model, right? You get access to this functionality. And you very much have the option to solve locally. If that's what you choose to do. It does not cost you anything. If you choose to use the cloud that does cost you Cloud Credits.

And that's how that goes. Cloud Credit is Autodesk currency that we created that you could equate one Cloud Credit to \$1. So think of it as, it costs you \$5 to run a kick ass render, right?

Let me show you some kick ass renders. This is a rendering that one of our users sent to the cloud. Our cloud rendering product development team. I literally went to them at the mothership and I said, show me some of your best renderings. Show me what some of your customers have done that you've gotten.

And this is some of the stuff that they sent over to me. So again, all of these are renderings. Very much capable. You're running fusion 360. And you're being able to render in the cloud.

JAMIE SCHERER: So the biggest benefit that I've seen with cloud rendering is if you do any type of visualization today it's typically a linear process, right? I'm going to hit render. It's going to retrace at some point between one and an hour I've got the image I want. And then I start to rotate it and I start my second view. I change the material and I do my third view.

So what happens when you've got a meeting in 15 minutes and you need to bring six renderings, right? You are stuck in a linear battle of making each one of those renderings.

With the cloud renderer in Fusion you set up your shot, lighting, materials send it to the cloud. Set up the next one. Send it to the cloud. Next one. Next one. Send it to the cloud. And then in the parallel process, in 10 minutes you get back all six renderings, as opposed to waiting for each one of them to render. So a huge benefit, if, you know, you do any type of visualization. Do you want to--

SACHLENE Yeah.

SINGH:

JAMIE SCHERER: We're right at the top. So we've got some forward looking kind of topics, if you're interested in what's coming in Fusion. But we're also right up at the top of the hour. Should we take questions?

SACHLENE

SINGH:

I'm going to go over the concept, especially just for these ones, really quickly. Just so everybody understands what they are. Branching and merging. How many people here understand what that is? So I spoke about a project. You have a file. You shared it with five other people. One person's working on, let's say, the electronics. Another person's working on the enclosure.

All of the sudden you are not in a situation where you have to say let me-- you have to check out a file while the other person waits to work on it, or goes to work on something else, right? If you're looking to meet a deadline, or if you're working on a timeline you could simultaneously work on the same file.

It's called branching and merging, right? So you basically branch the file. Whatever version you need to branch it at. And you say, all right. I'm going to take this branch and go work on this specific part of the assembly. And then you say, all right, I'm going to work on this. Once you're done, you merge it. And both of the branches come together, create the latest version. And all of those changes incorporated.

The point being, you could work on the same exact design at the same time. If there are clashes when you are merging it will go through them and ask you to say, make a choice, which one you want to keep, which one you want to lose. But it works really, really well.

Or if you want to explore design concepts. Branch A. Let's try this out. Let's try what this shape looks like. Branch B. Let's try what this looks like. How many branches you want, whatever you decide you like. You like this of branch A, that Branch D, merge it together, and you've got what you want your design to look like. It's branching and merging.

Shape optimization and generative design. It's been explained a few different times in the keynotes. It is a way to have the computer do your work for you. It is a way to have the computer generate design based on the requirements that you have specified.

So it is a simulation technique. Given that you have a bracket, you have certain mounting points, you need to be, you know, flat on one side, for example. And then you say, all right, what are my options of being able to create this bracket? Design this bracket?

And the computer will come back and say, based on the laws that you've given me and the fixtures that you've given me, these are the various different options that you have to be able to optimize this bracket, right? And then you take that design. You say, all right. I can keep this. I can lose that. This is what I can work with.

And so you let the computer kind of do the design for you. Carl Bass, our CEO, likes to say, CAD stands for computer aided design but computers really never aided us in our design, you know, of course, right to the point where it's doing the design for you. So that's what shape optimization generative design is.

You're lightweighting through this capability. So you're lightweighting in a few different ways. You're optimizing the shape. Another way to lightweight is create lattice structures that have the same structural property but are like lightweighting your object. And you might have seen that in a few exhibits in the exhibit hall, as well. So those are the concepts. Gonna fly by video.

JAMIE SCHERER: I think we should jump.

SACHLENE Yeah I'm jumping.

SINGH:

JAMIE SCHERER: Because you guys have been wonderful. And this is the last thing that you're here to do at AU before beer.

SACHLENE So sheet metal is coming soon.

SINGH:

JAMIE SCHERER: Sheet metal's coming. It's going to be inside of Inventor. I'm going to talk really fast. E-CAD integration is coming inside of Fusion 360. You saw a little bit of that in Steve Hooper's video, presentation, I should say. Browser. Fusion will be browser based soon, shortly.

SACHLENE There's a question right there.

SINGH:

AUDIENCE: When is [INAUDIBLE] dependent?

JAMIE SCHERER: Not soon enough. I don't know. Do you know?

SACHLENE It's a little bit later in the--

SINGH:

JAMIE SCHERER: Yeah, it's not--

SACHLENE I'm sure it will come. But it's a little bit later in the schedule.

SINGH:

JAMIE SCHERER: That's a good question. It's not there yet. And it's not tomorrow, unfortunately. Yeah. Good question. Yes?

AUDIENCE: So they're working on model-based definition in Inventor?

JAMIE SCHERER: Yes.

AUDIENCE: That's what they're working on in Fusion as well?

JAMIE SCHERER: It's another one of those that in time, but I don't think in the short term. But the model definition in Inventor that's coming is pretty awesome. If you've seen it, it's great.

SACHLENE I'll put it this way. If Autodesk was putting time and resources into developing anything for

SINGH: Inventors, it will show up in Fusion very soon.

JAMIE SCHERER: Any other questions? No/ All right. This is the--

AUDIENCE: [INAUDIBLE]

SACHLENE Which one?

SINGH:

AUDIENCE: Laser cutting.

SACHLENE Laser cutting?

SINGH:

JAMIE SCHERER: Laser cutting is in Fusion 360.

AUDIENCE: The [INAUDIBLE].

JAMIE SCHERER: Yes. Yep. Yeah, the entitlement that you have today is there.

AUDIENCE: And the five axis is too?

JAMIE SCHERER: Five axis is in Ultimate.

AUDIENCE: Is it going to have nesting capabilities?

JAMIE SCHERER: Yes. There's even-- I think there's some apps, some third party partners that have created some types of things. I could be wrong. But it is coming. As part of that sheet metal project, there's going to be-- Yeah. And as you all know, right? We have these tools. And it's becoming increasingly easier.

That's why we built Fusion. We have this new platform. We can start plugging things in, like the TruNest capabilities, right? Like the Nastran Solver, like you know, the CAM capabilities, which are already there. So that's a benefit. That's one that's going to be with that sheet metal project.

AUDIENCE: [INAUDIBLE] flat pattern or five axis?

SACHLENE Flat pattern.

SINGH:

JAMIE SCHERER: Flat pattern. Yeah.

SACHLENE Laser cut, water jet, so two axis. And anything five axis is Ultimate.

SINGH:

AUDIENCE: [INAUDIBLE]

SACHLENE Laser cutting? I'm not entirely sure. I'm going to have to find that out.

SINGH:

JAMIE SCHERER: I'll give you my card. We can get someone to answer that question for you, if you want to know more.

SACHLENE So basically, to summarize, there's a lot of different functionality that's available within the suite
SINGH: of tools that you have. Inventor offers a lot. We showed you a comparison matrix. There was enough that Inventor offers that Fusion does not have today. There's a lot that Fusion offers, because it's built up differently from the ground to encourage certain things, like collaboration, innovation, be able to reduce any overhead, that Inventor is not built to do as well.

So the idea was to highlight the advantages that you have in the workflows. It enables for you, specifically, to have both the tools in the box, which you do. So you need to get back out there. You absolutely need to start using both those tools. Figure out how they're adding value to your workflow and how you're going to go out and use them.

I am going to say, if it's not been apparently clear in the direction Autodesk is going in the keynotes, and how everybody speaking, Fusion is what the future of Autodesk looks like. That is where a lot of effort is going into. Inventor is absolutely an established, and a mature product. But if you want to understand what the future looks like, how does a company like Autodesk stay competitive, and that's a move that we absolutely have to make.

So this is a good time to make sure that you're comfortable with both the tools, that you're starting to use both of them, and you're making sure that you're using the advantages that both have over each other. Yes?

AUDIENCE: What integration with 3DS Max will there be?

JAMIE SCHERER: So funny you ask. I was talking with a gentleman named Chip Weatherman. He's one of the Max product managers. He told me you know, I don't know, some mock of being able to take joints, so you know, think constraints like in Inventor, from Fusion, directly into Max. And it would be an automatically rigged up object. So that's not there today. But it's coming. I hate that we have to say it's coming so much. But it's definitely something that we're working on.

AUDIENCE: Well, if you've got all the necessarily, like the insurance on that, just the file itself being able to be brought in, and textured, and rendered?

JAMIE SCHERER: Yeah, so you could export from Fusion FBX and bring it right over to Max, would be the best way to do it.

AUDIENCE: Do they use the same rendering engines?

JAMIE SCHERER: No. I don't think so. Do you know what render engine Fusion uses?

SACHLENE SINGH: I'm not sure. The local solver is different, and the cloud solver uses a different engine. They might, on the back end, use the same technology, because I know that our architectural products are using the same cloud solver, as well. So there's probably a bit of overlap going on there.

JAMIE SCHERER: The last thing I'll say, because we've said it's coming, it's coming, it's coming, you know. If you're not familiar, if you haven't seen it, if you go to the Fusion 360 community, go to the roadmap. The roadmap for fusion is completely exposed, right? Our development team has it out there for about 18 months.

So if you're curious, you know, we said, sheet metal. We said, hey, whatever it was. Look at that roadmap. And they've done a really awesome job at updating it when it needs to be updated. Things have moved up. Things go back, right?

But it's out there. It's not a secret where you're going to be waiting, and you're hoping, and it's never coming. They have that stuff out there for everyone to know what's coming and when. So

SACHLENE Thank you so much.

SINGH:

JAMIE SCHERER: Thank you guy.