

TAYLOR STEIN: So my name is Taylor Stein. I am one of the Fusion 360 evangelists here at Autodesk. And in this class we're doing the Top 10 Productivity Tips in Fusion 360.

I think this is going to be really worthwhile, regardless if you're brand new to Fusion, or if you're someone that's been using the software for over two years. You should walk out of here with something new, something that you had no idea that you could do before. And that's going to save you time and be more productive at the end of the day.

So we have to go through the key learning objectives first. So mainly I broke it down into three sections. You're going to learn important best practices.

Things that maybe you learned Fusion one way, but it may not be the best way to be using it. And also some hard to find things. Something that is kind of hidden.

You have to hold a special button. You have to click a certain thing. Things are a little bit hard to find.

And then I also want to broke it down into things that speed up your workflow. So things that you can do to really be more productive at the end of the day, speed things up, and save you a lot of time. And then lastly, it's a little bit odd, but how to make selections easier in Fusion. I think it's one of those things that as long as we're manipulating a 3D tool in a two dimensional screen, it always going to be really hard to select things. And we have I think eight tips on making those selections even easier, and saving you time.

So again, this is the top ten productivity tips. There is a surprise. We're actually doing 20 tips because 10 wasn't really enough. So I figured we have 60 minutes. 10 was a little bit weak.

We could do 20 of them. What I also wanted to do is make sure that everyone here leaves with something new. Make sure everyone learns something. So what I was doing-- I'm more curious to do, and I hope it's fun for you guys is every one at the beginning is eligible to win.

And I just want to see after every tip essentially, after every couple, I'm going to check in, if you learned something new put your hand down. I want to see who is the last person remaining. The last man or woman standing that pretty much knew everything in the class.

We'll see how long it lasts. Maybe after the first two ever will be gone, or everyone knows everything and you didn't learn much. But hopefully it's the first one. So any questions about

that? No?

So again, we're doing 20. So we're going to have to keep it quick. And one last thing is I think it's best if we just run through these 20. I'll have time for questions at the end. So if you have questions, jot it down, keep it in mind, and you can ask it at the end.

Sound good? Cool.

So the first tip is adding or removing from a selection inside of Fusion. And this is something that I didn't know for a while. So let's say that you have this design right here and you want to make an extrusion. And I want to extrude all eight of these holes around. I can start the extrude command and select the faces that I'd like to extrude.

So let's say I'm not really paying much attention. I select seven of them, and I extrude it down to this surface down here. So I've already started the extrusion, and then I realize I forgot the eighth one that I want to add. It might feel like that you can't add that selection to the extrusion.

So all you have to do is if you're a Mac, you can hold Command. Or if you're on a PC, you can hold Control. And then you can go ahead and add that face to the selection set. And when you let go, it'll be added to the extrusion command.

So I know a lot of people, they try this, they feel like they're stuck in the middle. And you just end up hitting Escape to get out of it and doing it all over again. When really all you have to do is hold Command or Control, and that will be added to your selection set. And this is a button that you'll see over and over again to save time. And it's a little bit of a hidden trick.

And this isn't just for Extrude. It's for essentially any command. So if you're using Fill It [INAUDIBLE] for anything, you can always add to or even remove from. So if I accidentally picked this guy here that I didn't want, I can hold Command and select it and remove it as well. Got it?

And one thing, you don't have to be furiously writing them down. There is a handout that has all 20 of these with pictures and words and everything. So if you want to write it down, go for it.

So that's the first one. I'm going to check in. I guess I'm curious right now.

Put your hand up if you haven't learned anything new yet. OK, just a little bit of work to do. I

think we can hopefully get everyone something new.

So the next thing is really similar. It's adding faces to a shell command. So in Fusion I'll make just a box here to prove kind of how the shell command works.

So let's say I have this box. How the shell command works-- whoops. Let me give that some height. --is if you select Shell, and from the browser you choose a body, it's going to make it uniform wall thickness, hollow out the inside almost like an Easter egg. So that's one way that the shell command works.

The other way that the shell command works is if I go ahead and choose it again, if I select a face, it's going to remove that one face and hollow out the rest of the part. So this is kind of the two functions of the shell command that most people are used to, but there's kind of a question of what if I want to add additional faces to the shell command? So an example for that is this part over here. This is a motor.

So let's say I want to remove all of these sort of external faces as well as the underside here, and create a uniform wall thickness for this part. All I have to do is choose Modify Shell, and select the face that I want to remove. So now it's removing that face.

And again, similar to the last one, it might feel like you can't add another face to the selection. And you might feel like OK, I have to do this multiple times. And now it's going to get mad at me trying to shell a part that already has a uniform wall thickness, or you're extruding and it gets messy really quickly.

So again, all you do is hold Command on Mac or Control on PC. And you can go ahead and add all of those additional faces to the ones that you want to remove. So I can select all of those on the side, as well as this one here on the bottom. And now I have a nice shelled out part.

So a quick tip there. Again, something that a lot of people don't know. So let me switch back over here. I'm not going to check in after every command. I'll do it probably after every five to keep it going.

So this one is Select Other. So this is making selections easier in Fusion. So one thing that I see a lot is it's difficult again, to select things when you're working on a two dimensional screen in a 3D space. Sometimes you have things layered on top of each other. Sometimes things are in the way.

And to counteract this in Fusion all you have to do is Left click and hold, and you'll get a list of everything that is under your mouse. So it's going to give me here Depth. And it's going to give me all of these faces that were under the mouse that I could choose. Or parents as well, which will kind of work its way up the tree. And I could have faces, features that were used to create those pieces of the geometry, the bodies, or even the component that they reside in.

So this is really useful if you have something that's hard to select. One thing that I see a lot is let's say that we have a box like-- I'm not going to do it. Let me undo that one. Let's say we sketch on this plane, and we draw a center rectangle. And I can extrude it up.

And if I now want to split this in half, if I choose something like the Split Body command, I can choose the body to split. And the splitting tool, if I want to use the origin, it's going to be one of those things that feels like I can't click it, no matter where I'm looking. And it's not a construction plane, so I can't make it any larger. So what you really have to do, again, is just Left click and hold. And then you can go ahead and choose that work plane that you want to use to split the body.

So that's one thing that I see a lot is construction planes, you can drag them, make them bigger or smaller, but the origin is as big as it is. And sometimes people have a hard time clicking that. The other way to fix it is you could zoom out all the way till it's small enough and you can click it. But that's not going to work in every situation.

So that is Select Other. We can move on to the next one. This one's a really fun one. So this is more advanced, but if you're creating an Assembly in Fusion 360, and you want to create a joint between two pieces of geometry, and the point where you want to add the relationship is in the middle of space. So you don't have any solid geometry where you want to do that.

So let me go ahead and pull up an example here to show what I'm talking about. So in this situation, this is a caster wheel here. And let's say I want to add a joint between this component here and this one right here. So what I really want to do is-- there's two ways I could do it. I could align this face with this face, and give some sort of offset.

But everyone should get the feeling that that's not the best way to do it. That feels a little bit like it's not going to work. If we change sizes, it's not going to adapt. And it's a lot of additional math that we don't want to do.

So what we really want to do is align the middle of this shaft with the middle of the space between these two faces that we have here. And the way I used to do it is you can create what's called a joint origin. So we have joint origins here, which are just pieces of geometry that you can select and build these relationships between. So I used to create a mid plane, and then create a joint origin, and it was just a lot of work.

And I actually found this one out two weeks before creating the class. So hopefully this is new to most of you as well. So to do it, I'm going to start by creating a joint, just like I would in Fusion normally. And I'll select the middle of this component.

And you'll see that again, I don't really have a joint origin where I want it. It's going to line it with one of these faces. So all I have to do is Right click and choose between two faces. And now it's going to ask me-- this is kind of a small tip inside of this one-- if you stop moving the mouse, Fusion will always tell you what it wants you to do. So it tells me Select First Reference Plane. So that's going to be this one.

And Select A Second Reference Plane. That's going to be this one right here. And now select the location, which is really just where between these two faces, kind of what piece of geometry do you want to build this off of? And I want it to be that circle there, to align it properly just like that.

So that's one thing that I didn't know about for, I think, two years using Fusion. And I just found it out recently. And again, to do that you start the Joint command. You can Right click, and then you have this between two faces button, kind of hiding down there at the bottom. OK?

So where are we at? Is there anyone left that still hasn't learned anything new? OK, couple. I guess employees do count, I guess.

So we'll keep it going. At the end, if that's OK.

So the next thing I want to show-- so it's creating a joint between two things in the middle of the air. I actually need to do it for the next one, so let me do it one last time. So I'll do it between the middle here between two faces. And I'll select those, and go ahead and align it that way.

So now the next one is again, with creating joints, is making the selection of joint origins even easier. So that one, everything was easy to click. Nothing was kind of hiding in the middle between parts. And this is something that you find a lot. And it's just a really fast way to select

a joint origin without having to hide parts or make them visible or not visible, or move things out of the way.

So what I mean by that is if I now want to bring this sub assembly on top of this one, I can start the joint command. And again, I want to align the middle of this hole that I have running through the center with the middle of the shaft. And when I have my mouse right here, I can point and I can see the one that I want, but when I move my mouse over, I now have a really hard time clicking it. I can try to kind of look down the barrel and see if I can grab it, but it's very jumpy, it's hard to select. So again, this is that one key that will save you all sorts of time.

What you do is you move your mouse until you see the joint origin that you want to select. You hold Command on Mac or Control on a PC, and it will lock in just those three options. So it's kind of like keeping everything else put. You're not going to pick up anything else. And now I can select that middle one and align it with this one right here.

And I can say Revolute. And now we have a caster that's completely functioning. So that's how you can lock it down. Again, that's holding Command or Control. To eliminate additional ones from showing up, you just have to make sure that you can see the one that you want ahead of time.

So let's switch over to the next one. We're doing pretty good on time. Maybe we could have done more than 20. But I didn't want to be too ambitious.

So the next one is opening local neutral files. So what I mean by neutral files are Fusion 360 file types, IGES, STEP, and I think SAT. There might be another one that we'll see. But this is relatively new to Fusion. And traditionally, to bring any sort of geometry into Fusion 360, what we have to do is open up our data panel, choose Upload, choose the file that we want to upload, upload it, let it translate, then open it, which works.

But if you have a neutral file type, and you just want to quickly open it in Fusion, there's a way to do that. So all you have to do is from the File menu, choose New Design From File. And now we'll open up the finder here on Mac. And I can find the file that I want to open. And this is the file types that we can open here.

So we have Fusion, IGES, SAT, SMT, and STEP. So this is useful if maybe someone just sent you a Fusion file and you want to open it. And you want to get a quick glimpse at it. You can do that here. So I'll go ahead and choose this STEP file.

I'll choose Open, and it brings it right there without having to bring it up to the cloud, because these translators are local. So we can do it right away. And the reason why I love this is that often I'm grabbing parts off of GrabCAD or somewhere that someone has sent to me. And I want to quickly look at it and see if I like it or not. I don't want to have to upload it, and then realize I don't like it anymore, and then have to throw it away, or wait for that.

This just lets me quickly look at it. See if I like it. If I do, I can save it. And if I don't like it, I can just close this and say don't save.

And now it's not anywhere where I don't want it to be. Does that make sense? Cool.

So let's take a look at the next one. And this is, I guess not necessarily like a pro tip, but it's something that I recommend for everyone doing, which is customizing the preferences to be exactly how you want them to be. A lot of people don't know that there are some preferences that you can change, that will save you a lot of time in the end, if you find yourself repeatedly doing something over and over again.

And what I mean by that is there's three that come to mind for me. So if you click on your name and you go to the Preferences, the three that I see most common are the Default Modeling Orientation. So by default in Fusion, you have the y-axis pointing up. So this isn't a find for most people.

A lot of people don't really care which orientation they have. But if you're often sending your parts to maybe something like MakerWare for doing slicing for 3D printer. And your parts keep coming in on their side. And you wonder why you keep having to flip the 90 degrees every time, it's because they're being built with the y-axis up, when the other software thinks that z is up. So there's that disconnect there.

So if you want to eliminate that, you can put z up by default. And now you'll have your z-axis pointing up, which is one thing that I like doing. In my brain, I picture z-axis going up, so I changed it to that.

The other one that I have are Material. So under the Material tab, you can set the default material as well as appearance for your designs that you create in Fusion. So the material is the physical property. So if you create a part, you look at its properties, it'll tell you the mass and all sorts of things like that. The appearance is just how it looks.

So personally, I liked working with a kind of simpler matte gray appearance. So I checked the box for Apply A Different Appearance. And I chose a opaque plastic. And now all the parts that I make have this kind of matte finish to them. I just wasn't a big fan of the shiny parts.

So those two parts, and the last one in the Preferences, is if we come over here to, I believe it is Design. So the default workspace, I believe the default is Sculpt, which is if you import a part, or if you make a new file, or for imported parts, it will be Sculpt. I switched it over to Model. That way all of my parts come in, I'm already in the model workspace. I don't have Sculpt tools if I don't want them-- typically, I'm importing a part, and I want to have Model tools. I don't want to have Sculpt tools.

So those are three quick tips for getting the preferences right. The biggest one, I think, is probably-- or two are the z-axis and the Materials. So if you find yourself always working in wood, you might as well set the appearance to always be wood. That way every time you don't have to reapply the same appearance over and over again.

So let me go back here. And the next thing is enabling the parametric timeline. So for those of you that are not familiar with it, in Fusion we have this parametric timeline at the bottom, which captures in chronological orders all of the features that you use to create your design. And you have the option to turn this on or off.

But the one thing that I tell people is that it's not just a quick switch that you can toggle off and on. There are consequences when you turn it off or turn it on. So if you turn it off, it essentially flattens everything that you've done. So it kind of forgets all the history that you put into it. You have the same geometry, but it forgets how you got there.

When you turn it back on, it's just going to have one feature that represents the x number that you did ahead of time. And then you can start capturing history from there. But one thing that we'll see is that if you import a part-- so if I open up that STEP file, it's going to come in here. And it looks like I can start modeling, and I can.

But one thing that you'll see is that the timeline is off by default. And this is one thing that I see a lot of people doing is they import a part, they're all excited, they start working on it, they start sketching and extruding, but the timeline isn't on yet. So one thing that you want to do ahead of time, if you want to have that parametric history, is if you Right click on the top of the browser and choose Capture Design History, from that point moving forward, it'll capture all the features that you do.

So if you don't do that, and you start sketching, extruding, your features are going to show up here on the left side. And it's more limited as far as what you can do with the timeline off. So if you do want to capture that parametric history, I recommend that you turn it on and then start capturing from there on out.

So let's go ahead to the next one. So this is another one that I didn't know really until probably a month before I started working on the class. And this is Changing Component Opacity.

So I was doing all sorts of workarounds to make this work. And it turns out that it was there the whole time and I just didn't know it. So to do that in this example I have an enclosure where I have a board inside and I have this enclosure around it. And if I just want to make the top component somewhat transparent to see through it, what I used to do is apply like a glass material or acrylic and it kind of worked. Or I would choose a different visual style.

So I'd choose shaded with hidden edges. And now you can kind of see inside, but it's not really exactly what you want. So if you change it back to you shaded with visible edges, all you have to do is if you have a component, you can select it and Right click. And then just choose Opacity control down at the bottom here.

It's kind of down here hidden a little bit. And you have this little window right here that you can now adjust the opacity of that component. So if you know that maybe you just always want to have the top somewhat transparent, you can apply that. And now you can keep working with it. When you go to render, it will be perfect just as you need it.

And you're not having to jump back and forth between adding acrylic materials that are kind of hard to see through or anything like that. So again, this is just for components. And to do it, you Right click and choose Opacity Control.

One thing I'm curious. Who is left? Who hasn't learned anything new yet? [? Kaching? ?]

OK, there's like two of you. We have 11 more, so hopefully we'll stump you with one of these.

So moving on is the next one is editing an appearance swatch. I call them swatches infusion. And what that is is so we have these appearances in Fusion. If I open up this model of an iPhone that I got off of GrabCAD, and I started applying appearances to it. I added some white plastic that I liked.

For this color, this kind of looks like a mix between the rose gold and the champagne, but I wanted to do the goldish champagne color. So I added a copper appearance, because I thought it might be close. And if I open up my appearances, you'll see that there is a copper patina. And that's what I added because I thought it would get me in the right direction.

And one thing that a lot of people often, they're like, hey, I love the rendering, but you only have like four or five colors. That's kind of limited. I wish you have more. All you have to do on any of these swatches is Right click and choose Edit. And it will bring up this full color editor where you can change the color that's applied to it.

So if I don't really like this and I want it to be more orange, I can switch this over to orange. I can go ahead and drag this cursor around here. Or if I have a particular RGB value in mind, I could type that in here. So if maybe I'm matching some existing colors, I can pull some RGB values, type those in, and this isn't really copper anymore. So if I wanted to rename this, I can select this text and I could call it the champagne color that Apple calls their gold iPhone.

And now I have this swatch here that's updated. And it's any color that you want. And there's also even more things you can customize in there. So if you want it to be rougher and less reflective, you can increase the roughness. Or you can drop it down if you want it to be a shinier finish on the part.

And these are just the ones that are right here. It gets even more advanced if you open this one. You have some more controls over here, such as some advanced highlight controls. I typically don't get in there, but if you want to have even more control over the appearances you can do that as well. So that's how you can edit this.

Again, this is for any of these, so it's plastic. It looks like there might just be these, but these are really just starters that you can drag and drop and then edit as you see fit. So let's go to the next one. So we're halfway through, and it is almost five. Perfect.

So the next one is replacing appearances. And this is another one that I don't know about very recently. And this is a little bit different. In this case, I just had the copper appearance. It was close enough, so I could edit the colors and get it right.

But what if I wanted to tell Fusion that wherever you have this champagne appearance that I created, swap it out with something completely different? So maybe changing the colors isn't going to give me the end result that I need. Maybe I don't want to even look at anything

copper. I want it to be aluminum. There's no way that I can edit that and tell it now be aluminum.

So all you have to do is find the material that you want, and instead of dragging and dropping it on top of all the faces again, you just drag and drop it on top of the swatch over there, and Fusion will take whatever that color is applied and then replace it with that one right there. And they'll change the name. And now we have this blue one. And again, I could go back and edit this and now change that one, give it a new name. And it's a really quick way for jumping between these different materials.

Often, if it's plastic and you know that are going to keep it plastic, you just have to edit it. But in this case I'm completely switching the materials and they have different properties. So you can just drag and drop it on top, and Fusion will quickly do all the work for you, finding where that appearance was and switching it. I know I've spent probably half an hour before, dragging and dropping and replacing it, trying to click all the faces. And really, this does it in half a second.

Pretty cool? Yeah? Cool.

[? Kaching, ?] did you know that one? You did not know that one. OK, cool.

Cool. So no one's left. I think everyone learned something new and we're halfway through, so that's good.

We did it in 10. Maybe we should have done 10 the whole time. If you're OK with it, we'll do eight more.

So this is what's called Box Mode for T-spline modeling. So if I switch over to somewhere in here, I have a controller. So this is a way that you can view the geometry when you're doing free form modeling in the sculpt workspace. So this controller, you'll see that this is the default view. You have these T-spline faces and edges.

I have symmetry. I have these creased edges, which are represented by the bolder lines. And I can work with this. I can choose Edit Form. And I can move these points or faces around.

Do anything that I need to to kind of manipulate this in a free form way. So this is normally how I like to work. But sometimes if you have really tricky geometry, something that's really detailed, maybe this can't be at all a little bit wavy, needs to be perfect, you can enter a different visual mode. So in this case Box Mode.

So to do that, you select a face on the part that you want to change. You then hold Control and you press 1, and it will switch between the different modes. You can also do it by if you select it-- there we go. If you hold Control and press 1, it will jump into Box Mode. 2 will put you into this kind of mixed environment where you see both the smooth surface as well as the control cage around it.

And 3 will give you the smooth mode at the end. So Box Mode, it looks almost like a low poly version of it, but it's really useful for creating really nice topology throughout the model. Because if the model looks really smooth in Box Mode in this kind of very creased way, you're almost guaranteed that it's going to look amazing when it's smoothed out. So if you have intricate details, I always go into Box Mode.

And now I can see that maybe some of these points aren't exactly where I need them. They're not aligned where I want them to be. So now it's a little bit easier to manipulate. It's requiring less processing power. And it's just faster in general.

So this is how you can do that for manipulating the surface. And then I can select it and go back to Smooth Mode. And now I have the nice looking controller that I want.

The other way that this is really nice is if I have, let's just say a T-spline box over here. And so the Edit Form tool typically-- it stretches things. It's not adding any geometry to it. I can pull this.

But if I really wanted to almost extrude that face up, what I have to do is hold Alt, and then I can drag. And that will add additional edge loops as I go up. So you'll see that I hold Alt, I drag, and it's adding more faces. So I'm not just stretching it into some point that I can't manipulate anymore. I'm adding geometry.

And you'll see that it's pretty quick, but it's even faster if you're in Box Mode. Because it's a really simple shape now. And now in Box Mode Fusion has a lot easier of a time adding those faces. So you can quickly come up here and select the faces that you want, and work kind of in this more low poly way. But at the end you know that you're going to have this smooth shape at the end of the day.

Like that. So that's Box Mode. It's a little bit-- I at first didn't quite see why I would use it. But I love it for making really nice smooth topology. And you can really follow the lines all the way through.

So the next one that we have is utilizing the S key. So this is new. This is in, I believe, the last release. And the S key really just gives you all of the tools that you need right at your fingertips.

So if I go to a blank design-- let me close out of this one. What the S key is is you just hit the S key, and it brings up this tool box based on the workspace that you're in. And you can pin any tools here, or you can search for any tools that you need. So if I wanted to start the Loft command, I could type in Loft, and it searches for tools called Loft. If I type in Pattern, you'll see that it brings up the three pattern tools that I have.

This is also different for each workspace. So let's say I'm in a sketch and I want to draw a spline. And I don't want to go into the Sketch menu. Or I don't even want to go up here, I'm too lazy to do that.

I can just hit S, type in spline, hit Enter. And now I have the spline tool ready to click right away. So that can save you a lot of time. I love using it for the commands that kind of take a little while to get to. So if I wanted to do something like include 3D geometry, I don't really want to put my mouse all the way up here, go to Sketch, find the project/include, come across, come down, and find Include 3-D Geometry.

Include is pretty unique. So I can just type in Include, and it's right there. So if you find yourself wanting-- if you know the command that you want and you don't know where it is, or if you're too lazy to move your mouse that many pixels up to the top, you can just use the S key and type it in.

And the part that I wanted to show also was if you wanted to pin a command to this window here, type in the one that you want, and click this Up Arrow, and it will be added to the tool box right there. So pretty cool stuff. It's really recent, and I use it all the time now.

So the next one that we have-- hoping this rapid fire mode is all right. And hopefully we have time for questions at the end-- is adding constraints. So in the sketch work space, constraints are how you eliminate degrees of freedom of your sketch geometry. And to do that-- let me go ahead and delete all of this.

Let's say I have a couple lines. And that's a small tip there. If you want to draw a line and end it, and still have the tool active, if you double click at the end, it'll stop the line and still give you

the line tool. So you don't have to draw it and then hit Escape to do that.

So let's say we have these two lines. I can use the S key that we just learned about, and I can choose a spline. Then I could connect this point to this point with a spline. And now if I want to add constraints here, maybe I want to fix this point. I can select it and I can come over here to the Sketch Palette and choose Fix/Unfix to do that.

I'm going to undo that. But I can also do it with any sketch geometry selected, if I Right click down here it will give me all of the possible constraints that I can use for whatever is selected. So for a point, I have nothing else selected, so the only constraint I could use here is Fix or Unfix. I can't tell it to be horizontal with itself. That doesn't make any sense.

So the only one I can do is Fix or Unfix. And this works. It's really quick. I love it. It saves time moving the mouse around.

So I can select these two lines, I can Right click, and you'll see that I have these three options. And they're the ones that make sense for a spline and a line. I can say Tangent. Now that one is tangent. And I can come over here and again, choose these and make that tangent.

And I can still move these around and do anything that I need. So a lot of things you can get to in the Right Click menu, the Marking Menu. It is contextual, based on what you have selected in that you'll see that I have these standard commands here. If I have a line selected and I Right click, I have different things that show up down here in the bottom. And this is true, even in the model workspace.

If you have a face selected, and you Right click, you'll get different things than if you have an edge selected. So check it out. Right click. It will have a lot of things in there for you.

The next one is adding spline control points. So this is pretty unique, but often a lot of people are in a situation like this. So let's say we have these lines that are geometry that can't change. This is just a simple example, but you can imagine one that's even more complex than this. Is I have a spline here that does exactly what I'm telling it to do.

But what if I want to change the curvature somewhere between point A and point B, and I want to add an additional control point somewhere in the middle to change the curvature along the way? I used to, and probably a lot of people do it, is you select it, you delete it, and then you do it all over again, but you remember to click a third time in the middle. What you have to do is if you select the spline and Right click, you'll see that you have this insert spline fit point

option. So you can choose that.

And now you can go ahead, and with this crosshair, you can add where you want that control point to be. So I can add as many as I want. I add two. And now I have these control points that I can move around, and the spline still listens to the constraints that I have set up, and you're just adding these control points in the middle. If you want to remove them, all you have to do is click it and hit Delete, and you can remove the additional control points that you add. So again, you don't have to delete it.

That's kind of the theme of this whole class is you don't have to undo the work. There's a smart and easy way to undo it and fix it the way that you should do it. OK? Let's see here. So we have a little over 20 minutes left with five more, and time for probably 10, 15 minutes of Q&A at the end.

So the next one is going through all of the options that we have for distributed design. So Distributed Design allows you to reference designs inside of another design. So an example here is I have these two parts. So I have this one, which is a mount. And you'll see that I have this red component as well as this prop mount component, and it's linked.

So this prop mount component is actually a Fusion design right here. And it has all the features that I used to create it down here at the bottom. And what I did is to insert it I opened up my Data Panel, I found the part that I want to insert, I Right clicked, and I chose Insert Into Current Design.

All the things that I'm about to show work for if they are in the same project. If they're not in the same project, you won't have all of these options for linking them back and forth. If they're in different projects and you insert it, it's going to essentially copy and paste it in. But it's no longer a link to the original Fusion 360 design. So this one is brought in here, I assembled it.

You can see in the timeline that the steps that I used to create it, where I inserted it, I just created a sketch off of it, extruded, and made these cutouts, and created the joint. So I have these holes and everything. And I have this link icon here telling me that that is a linked Fusion 360 design. So the options that I have is I can Right click, I can choose Open to open up that design. So I actually already have it open here.

And what I can do is I can make a change to this design to show how it works. So maybe the length of this changes to 130. So this whole thing is growing that way. And then maybe I can

change the size of one of these holes in the middle. So let's do that.

Maybe we'll make this larger at 25. So my design has changed to really trigger all of this. To come into play is I can hit Save. I can change-- pretty much just say larger. Give it a name.

It will create a new version up here. It's going to say New Version Created. And when I switch back to the design where it's been inserted in, I have a notification down here telling me one component is out of date. And I also have it up here at the top. So I have options as far as how I can fix it.

I can click up here and say Get All Latest. So this, if I had, imagine 20 components that were out of date, instead of doing them individually, if I just want to batch, get all of them in there, I can choose Get Latest. And now I have all of the new versions in there. What I can also do is Right click and we have Get Latest. I also have Choose Version.

So if maybe the latest version isn't the one that I want, I can choose Choose Version. It will bring up a list of all of the versions of that design, and I can pick the one that I want and stick with that one. What I can also do is maybe I don't want this to be referring to the original design at all. I can Right click and say Break Link. And essentially all the connection between those two parts is gone.

But in this case, I'm just going to say Get Latest. And it's going to take a look at the other design, drop it back in, and recalculate all the features in the bottom of the timeline. So it put in a new one, redid the sketch, the two extrusions, and the joint. And now it is larger, and the holes adapt and everything like that. So again, all you have to do to kind of trigger this is just make a save of your new design.

That is one thing that I think is really easy to overlook. But having the flexibility to say I want to stick with this version, or I don't want the latest version, or stop looking at this one, a lot of other tools you'll find yourself kind of having to play by the rules that the tool chooses for you. You're stuck with the new version, you're stuck with the old version, they're not linked. And this really gives you the flexibility to work with it however you need to. So we have a couple more.

This is user parameters. This is something that I use all the time, almost in every single Fusion design that I have. Probably obsess over it a little bit too much. But if I find-- I have been here-- This is probably the most Fusion designs I've ever had open at once. So this is something new for me too.

What user parameters are are essentially variables that you can use throughout your design process. So if I come down here to Change Parameters under Modify, you'll see that I have these user parameters that I created. And so I created one called Material Thickness and one called Leg Width. And they have a value. And these variables are being used throughout the design.

So what that allows me to do is design-- this is a laser cut laptop stand-- I can design it with roughly a quarter inch in mind. So assuming it's going to be quarter inch, but when I actually make this and I go down to the shop to cut it and I grab my calipers, and I measure it, we all know it's not going to say 0.25 exactly. It's going to be a little bit different. So what I can do is come in here and type in the different value. So in this case, I can type in 0.2. Maybe it's completely off.

And I can change that value in my entire design updates. So wherever I am referencing that material thickness variable, it will show up. And that same thing goes for the leg width. So maybe I was making this, and I was too lazy to measure my laptop. I could create this variable, and I could change this to any size.

And if being that I have built this parametrically in the proper way, it all rebuilds. And where these variables show up is really important, obviously, to get it to work. So you'll see that for all of these extrusions and all the sketch geometry, we have these variables. So want I extruded this sketch down, I extruded a distance of negative material thickness. So if that variable ever changes, that extrusion changes, the Boolean operations update, and everything flows from there.

And to create these, again, under Modify, you can choose Change Parameters. And you can click a plus. Create your new user parameter. You can give it a name. Units, I love how many units we have.

There's nautical miles, which was new to me. Everything you could ever choose. And then you can apply that parameter wherever you need it.

So that's one thing that I use all the time. And I love making designs that are fully parametric, where if something needs to change, you'll save yourself-- I like to tell people by spending an extra half hour making it perfectly parametric, you'll save yourself hours at the end when it doesn't work the first time, because it's probably not going to. And you can just change

numbers really quickly and save hours at the end.

So the next one is how to automate some cloud renderings. So Fusion 360, being a cloud tool, allows you to create renderings on the cloud. So you don't have to use your local resources. So let me find-- if I can find the one that I wanted. This one.

So for this one, let's say I wanted to have for every version that I make a save automatically creates some cloud renderings for me. What I can do is switch to the render workspace. And down here at the bottom we'll have our rendering gallery. And that's going to load.

We're at mercy of the Venetian Wi-Fi right now. And what this is going to do is pull up all of my name views, which by default, are essentially top front, I think, and [INAUDIBLE]. There's a couple, so let me open them up here. So here are the default, use top, front, right, and home. While that's loading, let me go to Model.

And these views are just the default ones that are in any single design. What you can do is you can create your own name view. So maybe there's a certain angle that I want to always come back to. Like there's this hero shot of the model that I always want to come back to, no matter where I am. If I'm at some funny angle, I can click this Name View, and it smoothly brings me back here.

It's really awesome for any screen recordings. If you want to get nice panning shots and everything, you can do that too. And how to create that is it's really simple. You just move to the view that you want. Maybe I want something that's essentially from the top, but at an angle I can then Right click on Name Views and choose New Name View. And I can give it a name.

And I could say Angled Top. And now that is a name view that I can switch to whenever I want. And the reason why you're creating these is maybe it's something useful. An angle that you want to work from. But what's really important is the render workspace here, which looks like it's not the biggest fan of the Wi-Fi.

But how it works is you'll have-- let's see if it will load-- these thumbnails down here for each of the name views. And all you have to do is drag the one. There's a box on the left-- I feel like I shouldn't have to demonstrate it-- but there's a box that says Render On Every Save, where you can drag it across, and every time you make a save, that view will be rendered on the cloud for you automatically.

In the handout it shows it perfectly. So we'll let that run in the background. Let me go to the

next one.

So the next one is how to promote versions from the Data Panel. So in Fusion we have-- let me go to this one. We have the Data Panel where you have all of the different designs that you have in your A 360 account. And for any of these, I can click on one. It tells me the version number.

I can click on this I to see more information about it. To manage versions, you can click here to open it up in A 360. And that's going to launch your default web browser, sign you in to A 360, and you can do all the data management things that you need to do there in a web browser. But you can also manage versions here in the Data Panel without having to go to a web browser.

So do that, you can click the See Versions button here. And what it's going to do is it's going to load all the different versions that you have. We'll wait for that one to load. Venetian Wi-Fi is really bad, I guess. I don't think it's gone out completely.

And it'll give you a list of all the versions. And there's a simple promote icon. There's a little up arrow that you can click.

And it'll promote that to the top version for you. I wish I didn't have to demonstrate with my hands, but we might have to do that for right now. We'll see if that's back at the end.

And the last one is just dynamic sketch changes. So sketches are essentially at the base of everything that you do in Fusion. And being able to manipulate them freely, and watch as everything updates is really powerful and something that I didn't use too much. But I've started using it more and more.

And what I mean by this is-- yeah, we'll close the Data Panel. Let it do its thing. So if I have this design, if I can find it. This one. This is one that one of my colleagues has in a video that I love that really shows everything that you can do a sketch geometry.

So how we did it was created first this one sketch here, which has a construction line and these other lines here. And one thing what I'm talking about about dynamic changes is you'll see that I'm not inside of sketchbook 1. I'm not editing the sketch, and I can still drag on the geometry to move it around. So a lot of people think that you'd have to go in here and say Edit Sketch, and drag this around and say Stop Sketch, which works, but you can manipulate the

geometry just by clicking and dragging on it. And this is lines, sizes of circles, anything like that.

And what we did next is created construction planes on each of these lines. So you can see them here. And now, kind of going one step further, when I move the lines, the construction planes update as well. And you can roll with this as far as you want. So I created sketches on each plane.

And now we have all of these sketches and all of these work planes. So I'll hide the construction plane. And what I can do now is create something like a loft between these. So I'll select all the profiles that I want. And you'll see that it lofts them all together.

That's perfect. And what I can do now is any of the sketch geometry that's not fully defined, I can move it around just by dragging my mouse. So I don't have to go into the sketch. So if I want to drop this part down, I can drag the line down. And the line moved, the construction plane moves, the sketch moves, and the loft moves all dynamically.

So this is something where maybe-- yeah, I think it's awesome. So if you don't really know the dimensions, you can drag it around, find the size that you want, and then maybe you can apply dimension afterwards, once you use this to find the design that you're looking for. It's faster than going into the sketch, changing a dimension, hitting Stop Sketch. You can even take it a step further by moving these lines around, which is pretty awesome. You'll see that it is working dynamically and really fast in a really quick way to explore different design ideas.

And the last example that I have is this, but another one. So we have here-- we have a sketch where we drew two rectangles. And you'll see that they're blue, telling me that they're not fully defined. So I can move them to change their size. And I created two extrusions at different heights.

So you'll see that I have different heights there. And then I created this sketch of 3D splines that are tangent to each edge. And now what I can do is create a loft between the two faces, and tell Fusion that these four splines are the rails that I want to use for the loft. So for rail loft. And then I even put a Fill It on all of them.

Why not? And now what we can do is move that initial sketch geometry around and everything will update. So it doesn't matter where I put these. It's going to recalculate the extrusion, the loft, and all of those splines dynamically. And it's even redoing the Fill It.

And you could go a step further and shell this out to a uniform wall thickness, and move it around. And it's a really quick way to explore ideas. And way faster than-- I can't imagine any other way of doing this. So that is dynamic sketch changes. And we've been flying through this.

It's 5:17, which is, I think, just about how much time I wanted to leave for questions. So with that, are there any questions? Yeah?

AUDIENCE: [INAUDIBLE].

TAYLOR STEIN: What do the rendering things look like? I can see if I can find them, or I could find--

AUDIENCE: [INAUDIBLE].

TAYLOR STEIN: It doesn't really like this presentation at all, I guess.

AUDIENCE: [INAUDIBLE].

TAYLOR STEIN: Yeah, so if you go to the YouTube channel, there's a video on it. You'll see that's where I got about five or six of these. I stole them from YouTube. Sorry.

And it's also in the handout. So on the AD website, there's a handout that shows a picture of how it works. Question?

AUDIENCE: [INAUDIBLE].

TAYLOR STEIN: So how that works is if I can find whichever one that is--

AUDIENCE: [INAUDIBLE].

TAYLOR STEIN: So the question is, how do picking planes for 3D sketches? It doesn't really matter which plane you select. It's just going to ask you to select a plane because every sketch inside a fusion needs a plane to live on. But once you have the 3D geometry, it will work like a 3D curve, and you can add the tangency as you need to. Yeah?

AUDIENCE: [INAUDIBLE].

TAYLOR STEIN: So it comes down to the amount of the degrees of freedom that the sketch geometry has. And one thing to see the black and the blue lines, is you have to go in your Preferences under

Preview. Say check the box for sketch color, sketch geometry, based on constraint status. So that's the first thing that you need.

And then how it's set up is if let's say I turn the origin on, and I sketched straight here, if I draw a circle, you'll see that I didn't give it a diameter. So I can drag it and change its size around. This is a quick way that you can test to see is it constrained or not. I could then maybe select these two points and say coincident.

So now it's fixed in the middle. So I can still drag it around. I can add a dimension, and maybe we'll make this 75. And now it's black because I've eliminated all the degrees of freedom. So it really is based on the dependencies what it's constrained to.

And really using the constraints to eliminate those degrees of freedom and lock it down until there's only one solution for it. And that's the only way that it can be sketched. A question in the back?

AUDIENCE: [INAUDIBLE].

TAYLOR STEIN: Yeah, so the question is, how do you remove constraints inside of Fusion? Correct? So if I have this circle here, and let's say that I tell Fusion that I want these two points to be horizontal from each other, all I have to do is select the constraint by clicking on it. And then hit delete, and it will get rid of the constraint.

And if you're having a hard time clicking that, you can use whatever tip number 4, and Left click and hold. And then choose Sketch Geometry Constraint there, if you're having a hard time clicking it. Yeah?

AUDIENCE: [INAUDIBLE].

TAYLOR STEIN: I don't think so. I think the end is the end. I think that's the only way to do it.

AUDIENCE: [INAUDIBLE].

TAYLOR STEIN: So the question is, you can edit these appearances, create your own, create your own name. Is there a way to save them?

AUDIENCE: [INAUDIBLE].

TAYLOR STEIN: I don't believe there's a way to make it global where that one appearance swatch is all over. I think you'd have to manually do it in each of your designs. Unless they're being brought together with distributed design, where you're inserting them in.

AUDIENCE: [INAUDIBLE].

TAYLOR STEIN: Right. I don't believe so. But I'm not 100% sure.

AUDIENCE: [INAUDIBLE].

TAYLOR STEIN: Was there a question over here? Yeah?

AUDIENCE: [INAUDIBLE].

TAYLOR STEIN: Yes. So the question is, is there a sketch tool for equal sections? So that's going to be over here in the Constraints. There is an equal constraint that you can apply between certain things. So if I have this circle, and I want it to be equal to this circle, I can select it and choose Equal.

AUDIENCE: [INAUDIBLE].

TAYLOR STEIN: Gotcha. Right now you would have to do that with essentially some sort of kind of math going on. So you would have the quickest way I think to do it would be draw a line, let's say this is 100, and now I have a line here. You're going to have to add a dimension here between these two. And I could say that this is-- let's see.

AUDIENCE: [INAUDIBLE].

TAYLOR STEIN: Right. You can add these, do some math to have it equally spaced out. And then use-- I believe you can break. You can break the line. But it's not like one button to do that in Fusion. So I think we have time for maybe two or three more questions, if there are any. Yeah?

AUDIENCE: [INAUDIBLE].

TAYLOR STEIN: So the question is, if you have a distributed design, and you Right click on a component and say Save Copy As, is it still linked? I believe that once you save it as a copy, it's purely a copy,

and it's not linked to the original one. So there's no link there. And it's going to save it out as its own Fusion design file. And whatever features are were the timeline for that component will be brought along with it.

AUDIENCE: [INAUDIBLE].

TAYLOR STEIN: Yeah. When you insert it in. Any other questions? Yeah?

AUDIENCE: [INAUDIBLE].

TAYLOR STEIN: Can you clarify?

AUDIENCE: [INAUDIBLE].

TAYLOR STEIN: I think if I'm understanding it correctly, so if I have like a design that is--

AUDIENCE: [INAUDIBLE].

TAYLOR STEIN: Gotcha. That's an interesting question. Maybe we can chat after about it. Yeah.

Maybe time for one last question? Or if there are any? No?

AUDIENCE: [INAUDIBLE].

TAYLOR STEIN: Yeah? I'm going to be around for a little bit afterwards if--

AUDIENCE: [INAUDIBLE].

TAYLOR STEIN: The only way to do it is going to be going to Modify and pulling up the list, and looking at it here.

AUDIENCE: [INAUDIBLE].

TAYLOR STEIN: You're going to have to close it. So if you don't remember it, you can just put in a value, and then go back and add it afterwards. That's probably the best way to do it.

AUDIENCE: [INAUDIBLE].

TAYLOR STEIN: One more. Over there.

AUDIENCE: [INAUDIBLE].

TAYLOR STEIN: Correct.

AUDIENCE: [INAUDIBLE].

TAYLOR STEIN: Correct. So to do that, try to find a good example. We have maybe this one. So under Inspect, the question was how do you turn the color component cycle toggle on? So to do that, you go to Inspect, and you choose Component Color Cycle Toggle.

And now it will color code it. And you'll see that you have different colored components. And they match here in the timeline. So if I work on just the frame component, if I activate that, you see that we just have essentially these blue features there.

And that's a good way to see how your features-- where they live. If they live inside of a component, or if they all live at the top level here, which is kind of if you're doing more of a bottom up, or top down design approach. I'm going to jump around a little bit, if that's OK. Yeah?

AUDIENCE: [INAUDIBLE].

TAYLOR STEIN: Yes. So the question is, can we isolate a component? So if I want to just work on the frame, I can activate it, and then isolate by Right clicking and say Isolate. And now I just see that one. And I kind of went quickly, but when you choose a component, and you Right click, and you choose Activate, you're simply just working on that component level. And then when you want to return back to the top level, you can Right click at the top and say Activate there and jump back.

So that's how you can kind of work on individual components at a time. Because unlike something like Inventor, all of these parts are living inside of one Fusion design file. They aren't living in their own design file. It's just a little bit of a different approach. If that's not unclear, we can chat after about it.

AUDIENCE: [INAUDIBLE].

TAYLOR STEIN: That is a good question. Right click on it, and say Unisolate. That wasn't a planted question, I swear.

I know there's a ton of questions. We'll all be sticking around for a little bit afterwards. But I have a few more slides to show before I let everyone go.

So we have this slide, which just shows everything that's going on at AU for Fusion. We're pretty far down the list here of things. But we do have office hours, primarily on the third floor.

So if you have questions about Fusion, you want to come to that. Come with your questions. We have people there all the time.

And this is just a good slide of everything that's going on at AU. So the last thing is all that I ask from you guys is if you enjoyed the class or if you didn't enjoy the class, please provide feedback. I'd really appreciate it. Last year I got a ton of good feedback and would have won something, but not enough people filled out these forms. So hopefully everyone does their homework and fills out the form, and rate it however you see fit.

But that's everything that I had. So thank you, everyone for coming. And I'll be here for questions.