PHIL EICHMILLER: Welcome everybody. Fusion 360 Workflows for design success; Top Down versus Bottom Up. I'm Phil Eichmiller, this is Sachlene Singh. So a little bit about us. There's me working on one of our deployments. I'm a QA on the Fusion Team. I work out of Lake Oswego. I do mostly customer style testing. So I'm just working on it like you guys and reporting bugs and doing workflows and that sort of thing. I'm also an instructor at Portland Community College. I teach [INAUDIBLE] at nights.

SACHLENE SINGH: And I'm Sachlene Singh. I'm one of the technical evangelist for Fusion 360. So my job is essentially to drive awareness around the product. I do a lot of public speaking engagements, a lot of one to many. But I also work very closely one on one with a lot of companies, customers, startups that are actually using Fusion to bring a product to market. And that's where you get the hands on, that's where you get to be a mechanical engineer or a consultant on the team and really get your hands dirty with the product as well.

So that's where we get a lot of the learning from. And every time I have a Fusion question, Phil is my go to person. So you're in for a fun ride for the next hour.

PHIL EICHMILLER: So what is this class about? It sounds like in an hour we could cover the whole universe of design and by the title of the class, right, obviously, that's not possible. But Fusion has its own unique point of view when it comes to what is top down, what's bottom up design. And basically rules of the road. And so some of you may know me know on the form as Phil.e. I answer a lot of these questions from users, and my goal today was to settle out a few key things about top down versus bottom up workflows and Fusion. All the details that you guys put in your designs, that's your business.

But what I want you to do is to get started on the right foot and get to design success because you are taking the right steps along the way to give Fusion what it needs for your designs. So the takeaways from this class are going to be I want you to know the difference between what Fusion thinks it looks like to do top down versus bottom up design infusion. I want you to know the difference between bodies and components, actually that's quite key to this whole discussion. And there seems to be some confusion around that. So we're going to get that cemented in. And along with some other good tidbits for your workflows.

SACHLENE SINGH: Awesome. So with that, let's jump in to a little bit of theory, set the stage a little bit, and then we'll dive right into the product. Phil and I have a lot of great content to show you. We want to

keep it really casual, really conversational, but we are also looking to get through as much of the content as we possibly can.

Quick show of hands, how many Fusion users in the room? Fantastic. That's brilliant. How many of us know the concept of top down versus bottom up design? Excellent. So one of things that we hear from users consistently is Autodesk is putting these powerful tools in the hands of our users, right? So with a tool like Fusion, you could essentially design whatever you wanted to. On the other side, if you're not quite educated on design methodologies or on manufacturability, you're likely going to run into some trouble when you start prototyping or thinking about manufacturing that part.

So it's a fine line between making the product too complex to make sure it adheres to design verses making it wide open in letting you do whatever you want with it. So it's a good idea to take a step back, and let's talk about what top down design or bottom up design might be. And that's review bodies and components as well.

So this is a neat graphic that Phil pulled. And really essentially, it represents top down versus bottom up really well. So top design is let's say you have an idea, and you're trying to ideate and build it out. And most of what you would do is explore the design within one design environment Fusion, we don't have parts or assemblies separated. So same design environment. And you're trying to ideate and figure out what your design looks like, right?

Bottoms up would be where you have all the different components that go into design. You likely have an idea of what the final design is going to look like, and you're putting those components together to get to the end result. So quick explanations and what that might look like. This is a bike light. And so if you had to do top down design on the bike light, you would have a couple of different sketches in three planes ideally. And then extrude those out to be able to get the shape that you're looking for, right?

So those base sketches in the three planes, in the three origin planes are going to drive what that final design is going to look like. If you did this bottom up, what you would find is that you're designing the screws separately or pulling the screws from a design library that you have the lights, you have the different plates that go into the design, and you're bringing them into one environment constraining them together, applying joints, and putting it together that way.

So that's essentially the difference between top down versus bottom up design. So very

quickly those would be the three sketches that you would likely draw when you design that bike light. Looks like a napkin sketch. Again, this is more of the process of ideating. Top down design is also extensively used when components need to be designed in context with each other. So let's say you had a certain component and you needed to build around it. So you need the references for that component. Very much top down design. We'll go into a little bit more detail about that as well.

Let's talk about the advantages of top down versus bottom up design. Let's say you had a bike light that you drew a specific way. Again, you have one sketch driving that bike light. So if you wanted to change it, if you wanted to make it bigger, if instead of two lights, you wanted to add 3, if you wanted to change the dimensions on the light for some reason, very easy to do that. Because you're driving most of your design from one or two sketches. Right? So some advantages of top down design there.

And then there is bottom up design. Which is the advantages there are the reusable part. You're either pulling from a library of components or you already have components that you know were used in some version of this design, and you're just changing some parts of it, and you are reusing a lot of those components. So bottom up design would be very applicable there.

This is a great example that Phil built. Do you want to talk about it really quickly?

PHIL EICHMILLER: Well, I just wanted to point out that everything except for the circuit boards here is a library component. Right? So getting started on a design like this for me means starting with bottom up design. Because I have a circuit board definition populating it with things I care about. And then moving on to the top down work that I need to do.

SACHLENE SINGH: Fantastic. Another really neat application of bottom up design is keeping a level of complexity within your assembly to a minimal. If you've been using Fusion for awhile, and you're designing every single component within the assembly, you quickly realize that your timeline is expanding. And that there's a lot of complexity that you've built into just that one design, right? Versus if you were doing it if you had some of your more complex sub assemblies as separate designs and then you linked them. So advantages and disadvantages to both methodologies there.

Ideally it's never one situation, or realistically it's never one situation. Realistically, you're

probably ideating that bike light, you're probably building it the way you need it to be. But the bolts are obviously going to be standard. They are obviously going to be something that you pulled from a library, right? So it's something that's called middle out, or it's a combination of top down as well as bottom up. So there's a speaker box here that Phil's been working on.

There's a lot of components here that he's built within the context. So the exterior of the speaker box, all of those bodies and components there are built within this specific file. But then there's also a lot of interior components, electrical components that he's brought in. So real work is always a combination of both of those ideologies.

All right, so top down versus bottom up. I'm going to take a minute. Any questions? Awesome. How many of us have done assemblies in Fusion? Fantastic. What is the first question you had or might have had or confusion you might have had as you ran into assemblies? OK. How about you need to convert your body to a component before you could apply joint to it? Right?

Having spoken to a lot of users for Fusion, they're like, wait a second. What's a body, and what's a component? And what is the difference? And why do I need to convert a body to a component in order to apply a joint? I did some research, and I basically said, well, let me go and see what the material out there is on bodies versus components. This is actually imagery that was on our learning site. And I thought it was really, really compelling. So a body is a modeling tool. It's essentially where you explore the design. It's essentially what you build, and you would potentially put together multiple bodies to create what your final design would look like or create what your final component would look like.

A body is a modeling tool. When you first open up Fusion and start creating something, it is a body. Right? A component is a container for bodies, sketches, and any other construction geometry. Right? So a component is what that final part would look like. So what you would essentially call a part in an equivalent software, CAD software, is called a component in Fusion.

What's really important to understand as you explore assemblies is that any time you have to apply a joint mate constraint, call it whatever you want, it needs to be a component. Other places where you have to have components is when you want to see in CNC parts. Anytime you're going over to CAM environment, it's important to have a component. Right?

Anytime you want to create drawings, also important to have a component. So that's the difference between bodies and components. In the context of assemblies, it's important to

understand that they need to be components. Some of the workflows that we have that Phil is going to show you, is actually going to show you, what you can do with bodies differently, how you would use it as a modeling tool, and then when you need to convert it to a component.

All right, with that, I'm going to hand it over to Phil. Let's jump right into the product.

PHIL EICHMILLER: All right. Thank you. So starting with the idea, OK, let's creating bodies, right? Anytime you've sketched something and, or maybe used a primitive command and extrude it, you've made a body. Right? So in this case, this is mostly just an example here, but what you're looking at and since Fusion doesn't have assembly and part document separate, you're looking at something where the component is the root of the browser, right?

So in this case, and this is what you need to pay attention to as you're working with this stuff, if you just want one part per file this is what it's going to look like. And everything that's in here is well this one isn't parametric. But it would be parametricly complete. This is actually a direct model. I prefer direct modeling, history free modeling for parts like this. That's just me.

But bodies are not also necessarily anything that's solid. It could be if you wanted to use them as a tool for instance to split away some little divots on the side of this thing, you might have some surface bodies or something like that. So those bodies are also going to live in the file. But the main point of the body in this file is just to be the part. So what are some related commands?

So a couple things I wanted you to know about working with bodies for instance is, of course, there's the split body command which is a fun one. Splitting tool can be anything planar or even other bodies or sketched shapes and stuff like that. So this just happens to be sort of a parting line kind of split that I'm doing so. Pretty simple stuff. Now I've got two bodies. So I've got crank, and I've got crank sup 1 down here.

So OK you probably know how to split bodies. But something to pay attention to when you're combining bodies. So if you've got multiple bodies in your part that need to eventually be combined because you're using them as tools, pay attention to the concept of target body versus tool bodies. And I've seen this mess people up before where they wonder where-- they go back and edit something, and they wonder where their body went because they did a combined that actually removed the body they want.

So in this case, crank is the one I want. And anything downstream from this is going to refer to

that particular body. And so the target body is going to be the body I want to act on which is crank. And then the tool body is the thing that I'm using to act on it, and then at the end of that process, I have the original body is still there. So just paying attention to dialogue boxes and paying attention to the browser to know exactly what you're acting on at any moment, that's actually going to get you downstream with fewer difficulties in this case. So something as simple as just working with bodies in this case.

So pitfalls, things to avoid. I've seen people's problem files that have several hundred commands in the timeline and at least 2/3 of it is body moves. That's a lot of compute times doing this with the body and you know nudging it all around the screen until finally it's in the right spot. So how do you avoid that? Just sketch it in place, create it in place. And so that's just like one typical pitfall of working with bodies.

SACHLENE SINGH: Remove.

PHIL EICHMILLER: Oh, yeah. That's right. I meant to show you the remove command. Let's see. No, not that one. I've got to find my thing here, hand on. Oh, this one. Here we are. So another big difference, and this works for components and bodies is remove versus delete. Right. So Fusion has two concepts of getting information from your design and getting it out. And so just concentrating on this body here. If I use delete which is probably what most people would default to in a parametric model what happens is you get this warning that apparently a lot of users ignore.

And so I see their timelines are chock full of red and yellow warnings and things like that, which are all just a bunch of failed operations that are working on cash geometry. This is what this is about to give me. So I'm going to say, OK, delete it. And now I've got you see my model changed. There's confusion reigns here. Because I thought I was just getting rid of something I didn't want. The thing to use is remove.

What remove does, and that's right down here on the right click menu, and this works for bodies and components, remove will get rid of it as a moment in time in the timeline. So right now I'll use remove. There's no failures, and the body is gone. And down here at the end of my timeline, you guys should be able to see if you can look over there way in the corner, there's a remove. And so I can move the timeline before the remove and after the remove. And you can see that it just becomes part of the parametric history.

So typically what you would want to do for something like those surface bodies I showed you

on the other model is if you want to clean up your model at the end, remove them. Right? You need them back in time when you're using them. And you don't need them at the end of your design. And I think usually what happens with most people is they get in trouble when they're in the middle of the timeline editing something and they wind up doing something like that that has downstream repercussions.

So you wind up with a bunch of failures. And Fusion is nice enough to cash most of your or all of your geometry or whatever. I mean in that case, it didn't seem to work very well. But that gives people confidence that nothing is really wrong. And they ignore the little red and yellow errors because their stuff is still on the screen. But Fusion's really telling you, you just broke the parametric history of this thing.

SACHLENE SINGH: So really quickly, we're going to talk about this towards the end of the class, but Fusion is capable of both direct modeling as well as parametric modeling, right? How do you know that you have parametric modeling turned on in Fusion? Anybody? There's a timeline, right? Fantastic. So if you see the time at the bottom of the screen just like Phil was just showing, that means that every move is being recorded in the history of the part. You could actually go back in time to a different version of where that part was and start editing at that specific point or create features there. And that affects the timeline downstream.

You could also direct model which is the first model that Phil had up. I bring this up because unless you're in parametric mode, you're not going to see the remove option. Just FYI.

PHIL EICHMILLER: Delete is painless in direct modeling, so history free modeling.

SACHLENE SINGH: Awesome.

PHIL EICHMILLER: Except things get removed from your model, and that might not be what you want. All right so sure. What's that?

AUDIENCE: What would be the difference between removing the [INAUDIBLE]

PHIL EICHMILLER: Well, removing it takes it out of the model. So if you're going to like export for 3D printing and stuff so forth, it wouldn't be there. Hiding it would just turn it off. So that it's still there. I mean that's a perfectly valid workflow. Some people leave all of their construction geometry in. So

that's not necessarily, it's not like advice to go remove all your construction geometry. That was just an example. All right. So all right.

Creating components. Right. So the main thing that I want to get across here is the activate component workflow. This is the one that ensures that what you have is a parametrically complete component. And, again, there's a few caveats here. Do you want to save this thing as a copy somewhere? Then you definitely need the full parametric history. Do you want to just keep everything in nice tidy boxes, mental boxes for you as you're working through your design? So that know what you're working on at any particular moment, that's a great reason to use activate component.

So I'm just going to demonstrate it real quick. New component command is found by right clicking on the top node of the browser. We hope to make that into a real command.

SACHLENE SINGH: Can you switch the screen?

PHIL EICHMILLER: Oh. I' not used to such a big switcher over here. All right. So let me back up. New component is found by right clicking on the top node of the browser. Like I was saying, we hope to make this into a real command one day that will live in the toolbar which would be really nice.

Once the thing is created though, it's not active. And so there's a little radio button off to the side that says activate component. And activate component does a couple of important things. One of the things it does is it puts the focus on the part you're making. The other thing it does is that every sketch and every body and most of the construction objects that you're going to make, the other activities you can do inside Fusion, will be stored inside of this component because they relate just to this component.

Well, there's an advantage to it. Can you see the timeline now? Right now there's nothing in this component. So when the component is active, what's going on is that the timeline is focused on just what makes that component. So that right there is hands down the main reason I would use it all the time. Even if I'm not going to be saving these copies out. Because I have a timeline that's super long, and I want to focus on just what I want to focus on.

So starting a sketch and creating just like let's say a support ring for this thing. See if I can get this thing in the middle here. There we are. Stop the sketch, and extrude it. And the default is new body. And the new body is going to be where? In my new component. So I'm perfectly

happy with that, and now I've got bodies and sketches that matter to this new component for my support ring. Now I want to go up and go back to activate the top component. You see the full timeline, and when you're down here, and you're interested in the part that you're really interested in, there's the features that make that part.

So that's a great way to stay organized, keep the timeline organized, and also this is a parametric with complete components. So now I can right click and say, save copy as, and this thing is going to go outside of here with all the information it needs to be a good part. I think that's all I wanted to say about that. I think we switched back.

SACHLENE SINGH: Super. So ideally you would create a new component and then create the sketch on the body within that component, especially if you were meant to copy that component to another location. Right? So that's the key to remember. Because then the sketch and the body are both in that component container. So you can go back and edit the sketch if you had to, and that would drive the body.

PHIL EICHMILLER: And the second point there is that I made the thing in place. Right? I didn't draw it over here to the side and use 15 body moves and a couple of snapshots to get the thing in the right spot.

SACHLENE SINGH: So creating assemblies. The first thing to know about creating assemblies is you cannot joint bodies together. Right? Joints can only be applied to components. We've already spoken about that. We've spoken about how to create new components so that the geometry of that component is held within that component. If that's what you choose to do especially considering the component needs to be copied to other places. The other way to create components is to simply when you start, create a design, it would typically be a body, right click on the body, and say create component. Right?

That's the other way to create a component, again, you can start applying joints to it and so forth. You can obviously reuse components. Right? So components that might exist in a different design, components that you might have the same design. So, again, any time you're editing or reusing components, make sure that you activate that component. Probably good best practice to have, especially when you're copying a component, you already know what that workflow looks like. And then we've spoken about what happens when you don't activate a component, the sketch gets left behind, right?

We're not going to go into a lot of depth about joints in this specific class. There's a specific class on joints tomorrow that Joel and I are teaching at 10:30. It's called Joints--

PHIL EICHMILLER: So Much More Than Just Rigid.

SACHLENE SINGH: So Much More Than Just Rigid. Thank you, Phil. So we're going to go into a lot of depth about the different kind of assembly options and joints and Fusion in that one. And then do you want to talk about Snapshots?

PHIL EICHMILLER: Yeah. Let's. So I want to show you something about Snapshots too. Because Snapshots are one of those areas where I think people don't quite understand what's going on, and they're saying yes to things that Fusion is leading them down the path of saying yes to a bunch of stuff just so they can keep working. And you wind up with-- I've seen other timelines where it wasn't body moves, it was literally hundreds of snapshots that didn't matter to the user.

So what is a snapshot? A snapshot is a parametric moment in time where that component has a specific position, right? So think of some sort of sliding mechanism, right? When it's in it's closed state, you have to do modeling around that. And when it's in some sort of activated state, that mechanism, you need to do modeling around that. So you want to capture those positions as snapshots.

A lot of what I see is people doing stuff like this because fiddling around with joints and stuff is lots of fun. And especially for my students at the college, they just wonder why things don't screw in and out and things like that. But then this is just hanging up here, the snapshot capture is just hanging out there. And when you do some other modeling command that requires a compute of the timeline, which is pretty much most of everything you're going to do from this point on, if you don't realize that you're in a different position, Fusion's going to ask you, hey, do you want to capture this position?

And if you say yes, what you wind up with is a snapshot in time of some unwanted position. So use the dynamic simulation capability of Fusion all you want, but remember to revert your snapshot. Because if you don't need that parametric information, why would you put it in there? So that's kind of the purpose of that.

Derive parts.

SACHLENE SINGH: Right. So we've talked about creating new bodies, creating new components, and then creating assemblies from that. So let's talk about editing bodies and components.

PHIL EICHMILLER: Yep. So you know editing a body is something that's going to happen naturally whenever you go to the sketch or whatever that created that body, right? Just like you can't make something without making a body. Well, you can't edit that thing without editing the body. So that's pretty much a given. Editing the editing components is a matter, of course, using the activate component workflow. You're going to want to activate that component and then go do the edits on it. So anything further you do to it is captured down inside that component. But with bodies, there's an interesting way to-- there's probably more than one way of doing this inside Fusion. But this is just one way and actually let me back up a little bit there.

So the best way to show this is just to do it. And this is what would be considered a derived part workflow, right? Where the body defines maybe a family of parts that come after it parametricly. And so you still want to be able to control the source body and also do edits to the downstream body. And, again, if you'll notice too something that's about to happen is I'm about to turn this into an assembly.

So the icon is a box. To create an assembly, you create a new component. Now that you have a stack of boxes, right? So maybe another new component. And then I'll just copy this, the construction body into these components. All right. So now when I pasted the bodies in, you saw that it was offering a move command there. I'll back up and do that one again.

Notice that the move command is focusing on the body. So if I move the body right now, I'm actually moving it away from its origin. It has a component origin, and the move command is asking me, because I'm getting the move command because I pasted, and so Fusion wants to offer you the chance to see what it is you pasted by moving it right away. But this is actually probably not the best time to move the thing. Moving components is better done by right clicking on them and using the move command again. You could just start the move command and filter for components.

But I like working with-- I'm an old inventor guy, so I like working with the browser. I like to right click on objects and tell them what to do. It's a very explicit workflow. And it gets me what I want every time with no questions. So that's why I like it. Notice, I have one snapshot for both of those moves too.

So I will capture this snapshot there so we can look at it. And then make some changes to these things. So you guys should be able to see some of the texture there. I can't see exactly on the screen how well it's lit up. I'm going to go to appearances and switch to faces, and I'm

going to just drag some different appearances on there. Where is the lines? There we are. Yeah, clouds. That's the one I like. Looks a lot like mosaic. But OK, I'll buy that.

So I have three different cups now that I can look at and decide which one I like. And I realize, you know what? This edge is all wrong here. So I want that edge to be the same in all three cups. So I'm just going to use the timeline and roll back in time to before when I created the components. Go in and do some direct editing on this thing. Just delete those fillets.

So now I have a nice crisp edge on the cup. I might decide to do something else with that later on. Who knows what the idea is? This is a very simple example. Because I'm like, I said, you guys know how to design your own complicated stuff. I'm not here to dazzle you with that kind of stuff. I'm trying to show you the workflow. And so now the downstream cups all have their different textures, and they have the source cup the body edit that I did to get the crisp edge out of it.

So just simply using the timeline, right? Before a certain point in time, the body has a definition. And after that it's been copied into components. And now we can do all sorts of things where we couldn't before. Like if I wanted to make this cup unique using press pull, just another little tidbit here, people really like press pull, look what the first option is, automatic. Do you know what automatic means? It's going to go back in time. It's going to change all of them. It wants to go to the source body.

So instead I'm going to pick new offset. And what new offset does is creates a new parametric event at the end. And so now the bottom of my cup, this one's got extra thick bottom of the cup, oh, it doesn't look like it. But I swear that's what I just did. It's hard to tell. I should've known better with the glass here.

But in any event, the other ones did not update to that, right? So press pull, if you want an event at the end of the timeline, make it a new offset. If you just use automatic, it's going to reach back in time, and I've seen that also really, really mess up someone's design because they'll use press pull because it's so fun to just drag on stuff. And they don't realize that they're going back in time and breaking things. And they have a big timeline that already has a bunch of red and yellow stuff in it. What's a few more red and yellow things?

You're doing exactly what Fusion is advertising you're doing there. So just pay attention to little things like that.

SACHLENE SINGH: Super. And then we talk about editing components. There's really not much to show here except the best practice we talked about before, which is any time you want to edit a component, it's probably a good idea to activate it. Right? Activate it, and you see a much simpler timeline and you're within that component, and you're making all of your changes there rather than at the top level assembly.

PHIL EICHMILLER: All right. So multi-body modeling. So anybody here do multi-body modeling in other CAD packages? Right. So my own experience with Inventor was that while it works, and it's awesome, it's not the best thing that Inventor's good at. That's just my opinion. I think Fusion's way better at it as far as a top down modeler goes. And so a multi-body workflow lends itself to-- or Fusion lends itself to a multi-body workflow, I think, more uniquely.

So let's see let's find my box. Oops. Here it is again. So let's back up to before our new component. It may not want me to. This--let's see. Hang on a sec. So there's our new component. Oh yeah. This has already been converted. Let's back this up then. That's a kind of a cool dance.

So here is our multi-body part. This is all driven by parameters or sketches. And it all exists as bodies. Now what's the advantage of multi-body modeling? That's mostly it's to keep all the information based off of one sketch or one set of parameters. And to relate the bodies to each other as you go. This doesn't show it off, but if you have something organic like this mouse, for instance, you would model this is as a single body and then carve it up into the various plastic pieces that it needs to be.

So in this case, there's a bunch of construction plans that have been placed to hack this thing up. And that's how we have a bunch of split bodies in the timeline down here that got us there. So really what's the tail end of this is if you're doing this workflow is to right click and say create components from bodies.

So moving from a multi-body to a multi-component workflow is basically achieved by create components from bodies. Just to do that, I will do it right now. And then you'll see that the components are created and the body still retains the name of the body left side. So parametrically it all tracks as far as what's associated with those bodies.

Another thing about create components from bodies which is kind of cool is you can do one at time too. So you could just right click on one particular body and turn it into a component if you want to. So in a multi-M body workflow obviously this sort of violates the activate component

workflow, right? There's competing workflows here. So I tend to do things heavily parametrically when I need to and not parametrically at all when I don't need to.

And in a case of something like this, this type of assembly would be one that just lives on its own. It doesn't matter what I do to it as long as it's all contained within this assembly. So while doing create component from body gets me a component that just has a body and no sketch to it, if I were to export this, what you'd basically wind up with is something like this where you have a body and it's a base feature. Right?

So if you do export bodies, their components with just their bodies and not their sketches, you wind up with a base feature. So I could still work with this file, it just doesn't have any parametric history to it.

SACHLENE SINGH: The other advantage of creating it as a multi-body part and then mass converting those to components is that they're already in place to where the assembly needs to be, right? And Fusion has this really neat tool called as built joints. So you could based on how they're already placed just specify what their relationship is. Or if there is any degrees of freedom between one or two components, and then off you go, right? You don't have to worry about placing them in the right direction, another advantage of top down design.

PHIL EICHMILLER: And another thing you could do too is just create a rigid group out of them as well. So you could do as build joints or create a rigid group. In this case, this would be a custom subwoofer box that's manufactured by somebody for specifically for the purposes of fitting a specific subwoofer. And so it doesn't have any real motion to it. It's just going to be a box at the end of the day. But if you wanted to use it in other assemblies, for instance, I would use rigid group there. To avoid being able to pull it apart just use rigid group. Now it's not moving anywhere.

Oh. Skeletal workflow. All right. Let's take a look at scalable workflow, which is a close cousin to multi-body workflow. Although I've used skeleton sketches to drive components around as well. So it's not necessarily strictly a multi-body workflow. In this case, it sort of is. But I just wanted to show it. This is kind of like, I'm obviously not a furniture designer, I'm way more of an audio guy.

But I just wanted to show that this is basically a workflow where you create a number of parameters that drive your design up front. For the speaker box, it was all driven by the volume inside the speaker box. That's what woofers need. Is they need a very specific air volume. So you could set that up for your car and set height and width, and you'd have the

right air volume at the same time.

In this case, I just wanted to show with something a little bit more skeletal looking some of that workflow. So just let your parameters drive the design. Then that's essentially the essence of skeletal modeling. You make it a little shorter, make it a little smaller, right?

So I developed the entire set of sketches that's running off of these parameters before I made any of the parts in this file. And so everything just sort of follows it around. The seat itself I believe is just completely derived from an offset edge that follows that around. So there's some scalable workflow for you. Any questions? Awesome.

SACHLENE SINGH: Super. So that was top down workflows. So let's talk a little bit about bottom up workflows. I have to say that when you spend weeks prepping for a class that's called top down versus bottoms up, there's no way that you don't get to that. Because what about the bottoms up workflows? What about the bottom up workflows? Anyways.

So there's a few different areas where you use bottom up workflows, right? We spoke about that. Where your reusable components that are going into several different designs, where you are trying to work with the complexity of your model. And Phil is going to show you a really neat example of configurations as well.

PHIL EICHMILLER: All right. So I discovered a little un-advertised yet awesome workflow involving configurations. So what I have here is I have a part standard electronic component. So I did a lot of audio design. So I've got libraries full of these things. And we had one and two channel versions of this thing. This is the one channel version. And it got me thinking that if I used versions inside of Fusion, I could actually just model on this thing and capture every variation of this object as a configuration. And it would be saved as a version, and I can label the version. So when you save, you get the little dialogue that says what are your notes about this? And you can enter in a label, that shows up on the versions list when you're doing bottom up workflows and you're right clicking on files that you've inserted and so forth.

So this is the one channel version. So let's get this thing inserted in here and see what happens. I'll just leave it where it's out for now. So that's how you insert parts into assemblies in Fusion. The assembly needs to be saved, and, of course, the part you're inserting needs to be saved as well. You have the little link here that shows that it's a linked file or a distributed design. And the cool part is you can right click on it and say choose version. And when the

version list shows up, it says to channel right there, and I click OK. And it gives me the two channel version.

Now there's one caveat. And this is something that I'm going to work on is can we ever get to the point where we can do this and break link? That's interesting. All right. Now I know what's going on there. I've got a failure in my timeline. I can just delete that. There we go. There. I got rid of the red thing in a timeline everybody.

So using versions as configurations, right? I don't care if this thing doesn't have a link. It's no longer linked back out to the original thing. This is a standard component. It's never going to change. I don't need to edit it. And I don't need it to update. The only caveat in this case is that I'd have to reinsert it again if I really did want just the one channel version. So just a neat way to kind of extend individual files so that your library parts have a little bit more variation to them. And while I'm here, let's just turn this guy off. Another workflow involving inserted components that I found as well is here's a nice AC jack that would go somewhere. Notice that it has lugs on the back for solder though, right? So inserting this into a circuit board design, I would always be using a point that's off of the part that's marked down on the circuit board somewhere for this thing. So my typical workflow would be to get a circuit board from the electrical guy. I would populate all of the parts. And it was just a painstaking automatic process. And iMates and Inventor never quite got it for me.

But in Fusion, we have a thing called joint origins which works a lot like the thing that I wish I had. Joint origins are really cool because you can place them places and then position them. And you know so when you go to Insert this or joint this, even if you're not inserting the part, this gives you a joint origin wherever you want it, and this is all parametrically captured.

You can edit the joint origin, nudge things around. So the biggest painstaking problem I always had was that the circuit board would change, and everything would move over a tenth of an inch. And so then I'd have to go through, and then eventually I got good at it in Inventor being able to get that done. But I still had to manually take a bunch of steps to get that done. And I found it's pretty easy when I'm using a joint origins in the circuit board file that are all based off of a single parameter like some sort of offset or something like that or just a set of parameters.

Another thing you can do with joint origins is actually just go ahead and just dump it right onto a sketch point. So if you'd rather sketched out where the joint origin lives, I've still got control over it according to the sketch that's running this thing. There's a sketch in here. Let me turn

on the-- oh, a sec. Oh yeah, there it is. There's our dimension for the joint origin.

So what does that look like when you get over here? Oh, actually let's go ahead and save that thing. Where did it go? No. I didn't name my parts very well. Let's go ahead and save this. Give this a sec. So when you insert into designs, you get one free move. Just FYI. There's no snapshot right there. Because there's no parametric history that's actually included that move. That move doesn't have to be included.

So then the next step would be, of course, to apply a joint. And the part goes into place. So now I can edit the location on the circuit board according to where the joint origins are all placed.

One caveat about working with files. And these are some of the things-- inserted files. Some of the things that I want you to keep in mind is that certain things have visibility, and certain things have visibility control that is. Once they've been inserted and some things don't. So joints you can turn on and off. It's hard to see here. Let's turn off this joint. Here we go. So you can turn on or off joints in inserted components. But you can't turn on and off sketches yet.

So this will probably change one day. But for now, if you don't want to see the sketches, prepare your components that are going to be inserted ahead of time. So that I probably should have turned this sketch off. What happens if I have to go fix this now is that I have to save this design, and I have to go save the other design. So we get version bloat over simple things like visibility control. So that's another thing to keep in mind when you're preparing your components for bottom up workflows.

SACHLENE SINGH: Fantastic. So just a quick recap of distributed design. That's what Fusion calls bottom up design, right? How Fusion does distributed design is you can insert components into your top level assembly or any assembly for that matter, and for every component that you've inserted, you see a little link sign, right?

That's where you go in and you say this is the version of this component that I want in this design or choose a previous version just like Phil showed, and that would update in your design. You can go ahead and break the link so that it's not associated back to the component you originally had. And now it just lives in the design that you insert it in. Right? So distributed design is bottom up design in Fusion language. Awesome.

So this is where we get to the part of direct modeling versus parametric modeling in a little bit

more detail.

PHIL EICHMILLER: All right, so something that's kind of cool about direct modeling in Fusion or history free modeling, you've just got to remember that it's a one way street, right? So if you're using it inside the context of a parametric model, what happens before the base feature can't affect the base feature. But what comes after the base feature is actually can be driven by the base feature.

So in this case, let's go ahead and capture design history on this part. You'll see that I have a base feature down here at the bottom that contains everything for this part. And I'm going to go ahead and just quickly make a sketch up here so that we can see the results of this. Now thank goodness none of my inventor students are here because I'm not going to completely constrain this thing which I just relentlessly pester them to do.

But I will put a dimension on it so that we know exactly what size this little leg is here. So this is one inch, and well, let's go ahead give it a real width too. 0.75. There we go. So we're just adding a part to this, a little feature that might be for a different version of this particular widget. Who knows? You guys make the complicated stuff. I'm going to join this. So now we've changed the future.

So we have history before and after this feature, right? So if we go to edit this base feature and use press pull, make this a little bit bigger. That's pretty cool how the fillet follows that, huh? That's just awesome. All right, finish the base feature. I guess I didn't really need to roll the timeline there.

You still have your one inch leg sticking off the end of it. So the direct history free model has downstream parametric history to it. It's going to push around things that happen after it. How you get there aside from converting something that was created without parametric history is to use one of these commands here either create form, everybody knows what T-splines are, I sure hope. Everybody knows what T-splines are?

So those are the same thing. It's a little history free node that shows up in your timeline somewhere. And, of course, create base feature which is just another way of adding a history free node including all of these commands here that just operate without necessarily the parametric history. After the base feature, the parametric references are all valid. So you just go back in time to edit base features and things downstream, populate from that.

Was there anything else here? I don't remember.

SACHLENE SINGH: No, I think you got it.

PHIL EICHMILLER: From parametric. Oh, yeah. And this is one thing. There's one thing I wanted to show you too. This may not be something. This is more like what it takes to rescue a design. Where the hell did I put it? Here it is. That's just remove history. Every once in awhile it might actually be beneficial to remove the history of your design. And then continue doing parametric history, parametric modeling rather.

So if you said do not capture design history, and it wipes out the timeline, and then say capture design history. Then basically all that parametric stuff is now just one group of base features at the bottom there. And you can continue on doing parametric modeling. That's a rescue situation. But sometimes it would work. The biggest problem, and the reason I thought of this originally was because I remembered all the times I would be been using Inventor on-- this is 10 years ago, so on really antique hardware that could barely do anything. And I would do a plastic part, and I'd get to the end of the plastic part, and it would be considering the entire parametric history every time I added a fillet.

Well, I had to put 200 or 300 fillets on the bottom of the plastic part. So right when I needed to finish everything, it got the most complicated it could possibly be. And it took 30 seconds or 90 seconds per edit. Where if I could have just said, you know what, I know this isn't changing. They've already signed off on this design, and especially in Fusion where you capture versions, and it doesn't even matter, I didn't have that at the time, so I would have to make a backup manually.

But if I could have just flushed all the history out of that thing and then moved on, it probably would have actually saved me lots and lots of time. So that is just something to consider as you're working towards the end of a design. What is that you really want to get done right now? Put the rest of the fillets on, or edit the parametric history?

SACHLENE SINGH: Super. So just as a time check, we've about 10 minutes left.

PHIL EICHMILLER: OK. Whoa. Are you awake still?

SACHLENE SINGH: Yeah. OK, so very quickly talk about how to upload and reuse data as well, right? So assemblies is not just creating bodies and components within Fusion. Realistically, there's a lot of CAD data we may be bringing in from other CAD products. [? DRABCAD ?] is a fantastic place to start, always a great place to bring in components. Or maybe standard components that live within other CAD products as well.

So there's a bunch of different workflows that you can use while uploading components. Phil, do you want to talk about that?

PHIL EICHMILLER: Yes. So if you're uploading SolidWorks or Inventor Assemblies, the number one thing you want to do, at least for my sanity, is like when I do a pack and go out of Inventor, I flatten it. So I can get all the data into one folder. And it's much easier to select all of that. If you've got multiple sub assemblies, Fusion, on the upload process, will prompt you to pick what's the top level assembly. So it will list all the .iam files at the top, and you pick the one that's the one you want to be the top level assembly.

So SAT files a STEP files. So SAT files come in from-- so if you convert an Assembly to a SAT and bring it into Fusion, it's going to show up as bodies. So it's going to be a flattened structure of bodies in the body folder in one component. So you'll automatically have to convert to components and start working with it.

STEP files will retain as much as it can, actually a lot of colors, names, component structure in the browser rather. And that's about all I wanted to say about that. So there's a few other things there. We're working to improve DXF and SVG input. So I really didn't want to demo that right now. But that's one way to get into the information into Fusion. And the canvas workflow is as simple as looking at the letter F over there. I brought in a Fusion F, sketched around it, and was able to-- and also calibrated it. So that it was exactly the right size.

Has anyone ever done that? I always wondered about the real value of sketching over things. But in Fusion, you can actually calibrate it down to about a pixel accuracy, which is pretty good I mean. And get to a design fairly quickly. So that's how I did that. I just right clicked on and calibrated the image, made it the size I wanted and sketched around it and so forth.

I guess that's about it though. If you guys have any questions, there's some stuff I wanted to cover about the rest of AU. So personally I'll be in the exhibit hall tonight from 6:00 to 8:00 PM in the manufacturing booth if you've got questions. I'd love to just talk with you one on one

about anything that's a challenge for you or any ideas you're having. And also I'll be at Answer Bar which is just outside the exhibit hall from like 8:00 to noon tomorrow morning. So that's where I'll be if you want to come find me. I've got business cards up here.

We also want to encourage you to go talk to the voice of the customer folks that are out in the main hallway out here. There's drawings for a I GoPro, and they're may be other things you can get from them. So just go talk to them. I don't know if you have to bring in this coupon, but here's some of these objects up here if you want to take one with you. There are nicely printed cards. Any other slides you want to do?

SACHLENE SINGH: Do you have a class on Thursday?

PHIL EICHMILLER: Oh, yeah. I forgot about that. So if any of you are wide awake Thursday at 8:00 in the morning, Joel and I are going to be presenting. There may be just a tiny bit of overlap here. But my next class Thursday morning is large model management. And it's similar to this in that what I'm going to talk to you about is like how to use the UI and basically what's a large model? It's when the screen gets filled up with stuff. That's what I call a large model. When I have to start worrying about, how am I using the browser? How am I using the timeline? How am I using the tools inside Fusion to manage all of this information. So it's pretty cool. A lot of more workflows and stuff like that to show.

SACHLENE SINGH: Right. And I mentioned taking Assemblies to the next step. Once you've created those components, you start wanting to defining motion or function for your Assemblies. So there's a class on joints that Joel and I are teaching tomorrow morning at 10:30. It's called Joints, So Much More Than Just Rigid. And we'll basically be going through the Assemble drop down menu. So every different kind of joint you have in Fusion as well as the power of context sets, and motion links, and motion studies, and all that good stuff as well.

I think we had a couple other slides here. So just in terms of events going on around Fusion that you should be aware of. There's the Fusion 360 Design Slam which is tomorrow evening. And if you didn't go to it last year, I have to tell you it's potentially one of the best events I've attended at AU. It's where four users of Fusion face off with a design challenge. And basically they're given something to design, and you watch them design it live in front of you. It's actually really, really neat.

And then there's The Voice Of The Customer, which is Tuesday or Thursday 9:00 AM to 6:00

PM. And then Office Hours that are held right outside here, and you get to talk to a Fusion expert if you have any questions. Answer Bar, we've all heard about that, when to find Phil at it. And yeah, any questions?

AUDIENCE: Can you talk about mesh to solid [INAUDIBLE]?

PHIL EICHMILLER: Well, OK, so mesh to solid. I mean we could demonstrate it, or what's your question about it?

AUDIENCE: Just whatever you were going to say about it.

PHIL EICHMILLER: Oh. It was you. So, well, so the way to do it is to get mesh into the mechanical model is you have to do it from one of the history free modes. So either inside of a base feature or in a DM model. And it's just a matter of uploading it, and then using the convert tool. I could show you where the convert tool is if you want. It's on the modify menu.

AUDIENCE: Convert it to--

PHIL EICHMILLER: Yep. Screen. Thank you. So I created a base feature, and it's in here. The mesh to BRep and BRep to mesh command is done from within a base feature.

SACHLENE SINGH: Right. And that's assuming you're starting with a quad mesh most of the time. So there's other Autodesk products that will take a triangulated mesh and convert it to a mesh for you. And then you could bring that right in Fusion and finish off the rest in that workflow.

PHIL EICHMILLER: So we're getting better at mesh modeling. And I think there's going to be a lot of mesh improvements that are coming. This is really rudimentary compared to what we're working on. But there's a couple of caveats that-- the faces need to be planar. So triangle mesh, if it's got triangles all on one planar face, and it's within a certain tolerance, then it will combine those. And you'll get rid of the triangles. But it doesn't always work. There's some things can be just minutely out of plane because the way meshes really are. They're not as analytics solids like that. So you might find things triangulating unexpectedly.

So I still think that will eventually improve. I mean we would definitely have a good strong drive towards mesh workflows.

SACHLENE SINGH: Super. Anything else? Yes.

AUDIENCE: When you are working with sketches and you miss to activate a component, you have to sketch up into your higher [INAUDIBLE]. You want to get down, and you know that it's independent of anything else, how can you do that?

PHIL EICHMILLER: All right. So I'll do that right now. And I'm just going to just make a little widget here. So I'm going to start a sketch. Go ahead and just make a circle. The object I'm creating is not visible. There we go. Well, I was inside a base feature. Hang on. Let me back up till I'm not in a base future. There we go. All right. So if I've created a sketch, why am I still in a base feature? Let me back up until it doesn't say base feature.

There we go. Is it going to say based feature again and trick me? There we are. Stop sketch. Extrude this as a new component, right? So now I have a component down here that's got a body. I'm not in a timeline model. Well, in a timeline model, you'd still do this. You would drag the sketch down to the component. And if just so long as that sketch doesn't have any parametric ties to the root, like if you haven't used the sketch for six other components, then it should allow you to rearrange it that way.

So chances are you could probably, in that case, if it's immediately after the fact, just make sure that you just drag it down into the component.

SACHLENE SINGH: Anything else? Yep.

AUDIENCE: [INAUDIBLE] SAT file in Fusion. That's a function [INAUDIBLE]

PHIL EICHMILLER: So when you upload a STEP file, it's gets translated and converted to a Fusion file no matter what. If you want to actually export something as a STEP file just use the export option. So you can get SAT, SMT, and STEP and F 3D files out of this workflow. So it's just up here on the file menu and it's export.

SACHLENE SINGH: Super. I just want to make sure that we are at time. So thank you so much. Thank you for coming. I really appreciate the time. Phil and I'll be here for the next 10 or 15 minutes. So definitely feel free to walk up and get your questions answered. Thanks so much everybody.