

**PETER DOERING:** All right, we're going to get started. Good evening, everyone. As I said earlier, I hope you still have a little bit of energy left after this long day. All right, a little bit. OK. So Fusion 360 Modeling Tips from the expert. I'm Peter Doering. I'm a Fusion Expert Elite, Engineer. I've been an engineer for almost 30 years.

I've worked with CAD software for 25 plus years, all sorts of CAD software. And I design lighting systems. So the company that I run is Trippy Lighting. And that's also my forum name. So if you've ever been to the forum, Trippy Lighting, that would be me. Jeff?

**JEFF STRADER:** I'm Jeff Strader. I'm a Software Architect on Fusion. I've been with Fusion since it was first born. I don't know how long that's been, six years maybe, something like that. And then before that I worked on Inventor. So we're going to teach you some modeling tips.

Our objectives today really are to get you to optimize your use of Fusion 360. So this really is not the beginner class that Phil taught this morning. It's a little bit more advanced than that. Basically we're going to teach you to make your designs a little more efficient, more easy to deal with, easy to edit, things like that.

And our rough agenda here, we're going to have Peter go first and talk about modeling workflows. I'll talk about parametric modeling. Peter will talk about configurations. And if we have time, I'll talk about Sketch.

**PETER DOERING:** All right. So modeling workflows. So the first thing we're going to talk about is Fusion 360, rule number one. And what is that? It's basically a workflow. And you can see this symbol here on my chest is a component. And basically with one exception in your Fusion 360 design, you'll end up with several components in your design. And the question is, of course, how do you get to a component.

So the Fusion 360 user interface suggests, to some degree, if you read it from the left to the right, the first going to make a sketch. Then you're going to extrude that sketch or revolve the sketch so you end up with 3D geometry. Then you may want to modify. You may add a chamfer or fill it. And well, that's where many users that are new to Fusion 360 already start to have their body. But that's just the body.

So the first thing that we need to go over really quickly is you need to recap what's actually the

difference between a body and a component. So a body is basically just one contiguous piece of geometry that may really be a solid BREP. It may be an open surface, one contiguous piece of geometry. That's a body. That's it. Doesn't have any coordinate system, really, that you have access to.

So what's a component? A component is basically the collector for all that stuff that you use to make that body. So all the features, where the features are in the timeline, the construction geometry, joint origins that you may later use to assemble stuff, construction planes, sketches, all that stuff goes in the component.

So how do you get that stuff in the component? So let's start like a normal-- I wouldn't say normal, but like a new user often starts-- capture design history, I've never seen that.

**JEFF STRADER:** That's my set up.

**PETER DOERING:** Ah, special version. So I'm just going to start with a sketch, something really simple. And because it's so easy, I extruded. OK, I got my first thing. Let me make a second thing. I'm going to do the same thing. And I had to create a circle here. How-- and what's that? Red. Who developed that software? I mean, cut what? Oh, I can create a new body. Great.

So now I got my two bodies. And I really don't want this here. So I've seen this Move feature, so I'm going to move this out of the way. And then I can modify those things. Let me put a Fill it. On this here. OK. And maybe put a chamfer on this here. All right. And then I've read somewhere, oh, I need a component. If I need to assemble those things, I need to make a component.

So OK, fine. Let me go to the bodies here and I can create components from bodies. Great. Awesome. So now two problems already. If you take a look at the sketches here, the first sketches here-- did I not make a sketch for this here? No, I didn't. OK. Anyway, too bad. That's fine. If I would have made a sketch for this here, because I moved it out of the way, that sketch would be located at the origin. That's problem number one.

The second problem was that we saw interference between those two components. So the question, of course, you know, is that a good, is that a good workflow. We all know what activation of a component does, hopefully. So I activate the component. And activation of the component filters the time line down here for those items that belong to that component. Right? So there's not much in it.

So how do we get it in there? So are there other workflows that we could employ to get our items to show up there in that component? So as I filtered this component, or activated the component, you saw that all of those features that vanished, they're not part of that component. So how do we get it into the component? Let's create another design.

And that's Fusion 360, rule number one. When in doubt, first thing you should be doing is creating a component and activating it. So new components are automatically activated. Right after the creation of a component, it's activated. Now you can go ahead and sketch something. Do something else really simple and create another circle here that I extrude. And we can already see that extrusion shows up here in our timeline. So that's part of that component now.

Let's create a second component. And in this case, I use another rectangle. I use another rectangle. There we see those bodies also don't interfere with each other because they are part of a different component. So now let's go ahead and activate this design-- the root of the browser, and add some features to those components.

And we see they end up here at the end of the timeline. So let me add a chamfer to this cube here. So that at the end of the timeline, but that's OK, because I activate the component. The features that are created on that component, Fusion 360 is smart enough when you touch part of a body to add a feature-- a face, a vertex, or an edge-- it recognizes that it belongs to that body and then it puts that into that component where that body is located.

So you might think, hey, you know, I just have those two pieces. That's no big deal. But as your design grows a little bit more complex now-- I have to find this here-- you want 35 to open.

**JEFF STRADER:** Keep going left. Left.

**PETER DOERING:** Left. We'll get there. We need a big monitor. Might have closed those. There we go. So this design was created with that first approach. The user created an enormous amount of sketches, extruded, and did whatever, revolved, and then created bodies-- components from all the bodies.

So if you go down here and activate any of those components, there's really not that much in there. There's just the component creation and the drawings that are associated with that part. But all the features, they're not in there. So if you want to edit a part, like for example, this

here, remember? Double clicking activates a component. Double clicking in the viewport activates a component.

If you only click a single time on an edge, on a vertex, on a face, that actually activates the body within the component. Don't use the Move command, then. You don't want to move a component within a body because you move it away from the origin. So anyway, if you want to edit this component, or this component, you see there isn't much in there. Their sketch isn't shown in the component because it was created before you ever created a component out of the finished body.

So you may think, that's OK. I'm just going to use that to look up the sketch here in my sketch folders. Good luck. So of course, that user also didn't apply Fusion 360, rule number two. That's not on this t-shirt. Maybe on another one. That's actually Name Your Stuff. And Jeff is going to talk about that a little bit later. So those sketches are here somewhere, if I find them.

There's a piece of that. And we can see this here is actually, that's the outline of that piece of geometry that sits up there. So they're dislocated from each other. Very irritating, and you'll have a lot of fun editing that component. Let's look at the origin of that component. So actually, the origin is also not where the geometry is or where the sketch is. So that's a certain disadvantage.

So what happens if you export a component like that? You decide, hey, this bracket, I love this bracket so much I want to use this in another bike design. Right? So you export it. I've done that already. And I've opened it here somewhere. Oh, there it is. So this is the export component. It looks all great with exception that you already have your first warning here for an extrusion. I'm not even sure why that does this in this case.

And there are sketches, right? So I have been lying to you. There are sketches. Those are reconstructed sketches. Where is Edit Sketch? You can actually edit those sketches because they're fully undefined. So if there would be any dimension in that actual sketch of we're able to edit right now, those sketches are fully undefined. Those are not the sketches that this component was created with.

So that's the problem. When you export a component that was created by creating a component from a body, you lose the complete design history. And that's not really so nice. So how does the design look that was created the other way? You could already see, going back to that other design, this browser tree is fairly long, and you know, with the sketches it's getting

hard to manage.

So this design-- that's one of my crazy lighting lamps here-- has 400 components. It's a very short browser tree and much easier to organize. And if you click on any of those components-- 400. So a lot of them are instanced, so they are equal copies. So that component that I just activated includes all the sketches that are needed for that compo-- actually, this one doesn't because I used another trick that I'll go over later. This one is an original component.

The trick that I'm talking about is a configuration, because you can actually make configurations in Fusion 360, to a degree. If you export this left mounting plate, you'll see that all the features, to sketch all the features, you can reuse that and modify that in a new component. So that's a certain advantage of, or that's that advantage of the Component First Workflow.

So the next question, of course, is that the only workflow? Are there other workflows? Would you always have to make a component? If you did, why would you have sketch in the first position? So there are alternative workflows. I'll go over to the two main ones. The first one is skeletal design. If you have a single sketch, and there's another unique feature of Fusion, you can extrude several bodies from a single sketch. And Jeff has a few tricks that he's going to show.

So in this case, for example, I use a single sketch. And that's basically it. Also in case you didn't know, you can actually, without having to edit the sketch, edit a sketch. Because right click on the sketch and show the dimensions, and then you can edit them. And what that does is basically, you go back in the timeline to the point in time where that sketch was created. But that's the whole sketch that I used to create this component.

So I basically extruded my first body, patterned it around, and so forth. And then in the end, I have all these components that I derived from one single sketch. And so in that case, it doesn't really make sense to start with a component, make a sketch, make another component, make another sketch. Because it's so easy to do off a single sketch.

So there's one other workflow, and I call that digital carving, where you actually don't start with a sketch at all. You start with a body, with a T-spline, in this case. You might use, you know, the Patch features. You have Lofting, Surface Lofting, to do your industrial design, where you have several bodies. In this case, you'll start with one body. You have a T-spline. You have

one T-spline here and a second one for the stem. Let's forget that one.

Then I create a sketch and I use that sketch as a split body operation. And I split this body into two pieces. And then I make two components. So what all those workflows have in common, create a component as soon as you can. Because as you've seen, as soon as you have a component, once you start adding features, you add all those features to that component. They're contained in that component.

So I drag this here all the way to the end. That door, I added the Shelling feature and some filleting, and the [INAUDIBLE] will be contained within that component. And that's the first section about modeling workflows. Yes?

**AUDIENCE:** I have a question on activation.

**PETER DOERING:** Sure.

**AUDIENCE:** When you go into the ground to activate, can you do it graphically?

**PETER DOERING:** Yes. You can double click to activate a tool to highlight the component or to select the component. Right click and then say Activate. Do you see that?

**JEFF STRADER:** That was rule number one. So we'll talk about parametric modeling tips. I want to take one quick detour here and talk about our handout. So when Peter and I started putting this class together, we realized this was not a one hour class. This was about a one week class. So we put together a handout that is, I think it's 100 pages at this point. And it is published with the class. There's a PDF form of it.

But inside that PDF, there's also a link to a Google doc, which is this. And our intention here is to keep that as kind of a live document, add to it over time. So take a look at that. And you can even watch it to see how it--

**AUDIENCE:** [INAUDIBLE].

**PETER DOERING:** Sorry about that.

**JEFF STRADER:** OK. So let's talk about parametric modeling. You know, there may be people that come to Fusion from Fusion's own direct modeling routes, in which case parametric design is new. Even if you come from an existing parametric modeller, like Fusion or like Inventor or SolidWorks, there are some aspects of Fusion parametric modeling that are important to

know.

So the topics we'll talk about, first understanding the Fusion timeline-- Peter showed a little bit of that. The second is related, that Fusion assembly operations are timeline entries, too. And that they're history based. The third one is probably the most important, and that's understanding errors and warnings, preventing them, fixing them. And then finally, if we have time, we'll talk a little bit about organizing your timeline.

**PETER DOERING:** Starting with a component is a good idea.

**JEFF STRADER:** Right. So first, the first step is that the timeline in Fusion is a time travel machine. When you move that end of design marker back, you're actually traveling back in time in your design to when that feature was created. That's true not only for moving that end-of-design marker, it's also true for Edit Feature and Edit Sketch. So you need to be aware of that when you're editing your design, especially. And then as we said, this can also affect assembly operations. But let's start simple with a simple part and the time travel aspect.

So this is just a single body part. So it doesn't have any hierarchy to it. It's just a single linear set of features. And as we said, if you scroll this thing back, you can step back in time through your design and see the steps that were taken to create it. That's all pretty straightforward. It fits in with the sort of VCR controls here. You can step through the design and kind of watch it evolve over time.

So that's all that's pretty straightforward in a single body. But there are some catches here, too. So if I, for instance, want to edit this sketch, that operation also rolls back the design to that point in time. So if for some reason you wanted to say project that hole into this sketch, your natural instinct is going to be to edit that sketch. Well, that hole doesn't exist yet. So you can't project that hole into this sketch. It's just not possible. It's in the future.

So for anyone coming from Inventor or SolidWorks, that's all very straightforward. This is basic parametric modeling 101. But where it gets unique to Fusion is that assembly operations are also history-based as well. It includes things like component creation, component instancing, joints. Everything that you do that deals with components, creating new components, those are timeline operations as well. And this is something that you really need to keep in mind.

So let's talk about assembly modeling time travel. And since we're talking time travel, we're going to take us back to 1961. That's me. And that was a Christmas present I got, this old

pedal car. So I set out a few months ago to try to model this thing. You'll see I didn't, I never finished it. I didn't finish the body. I need some good T-spine help. But it provides a good example of some of the pitfalls of Fusion assembly modeling in parametric.

So I've got the mechanism all in place. And as you can see, there's components. They're fairly nicely organized and named. But if we look at the timeline here-- so I'll just pick this joint. And this is another tip. You can use the Roll History marker to here, so you don't have to go all the way to the end to find the spot.

So notice the first thing. By rolling the History Marker there, a bunch of components don't exist anymore. Right? Those are all created to the right of where we are. And you can see these little white block things are where we create components. So there's a bunch of components that just plain don't exist anymore.

Often in the forum I'll see-- a common post is, I want to project the edge of this component into my sketch. But when I edit the sketch, I can't find the component anymore. Well, that's because it was created afterwards. So you need to kind of think ahead about the kind of projections that you'll need to do, the kind of references you'll need to have, and make sure that your components are created in an order that makes that possible.

But it's more than just component creation. It's joint creation as well. So right now, I've got this assembly all joined together. I've got a joint here. I can rotate it. You can see the crank move. And specifically, this right side wheel has been joined. But if I roll back one more step, that was the joint that held that thing to the axle.

So at this point in time, that joint doesn't exist. So we can't count on that existing. It also includes things like the Position feature. So here, at this point in time, I've got my two wheels. But I created them, the instances on top of each other. So they're, you know, you can't tell one from the other. So this Position feature then allows you to move it somewhere else.

And even the act of creating the instance itself is a timeline feature. So roll back one more step and now we've only got one wheel assembly in our design. So I think the takeaway from all this is to know that these component operations are history features, too. And that may be different than you're used to. So keep that in mind.

All right. So tip four-- it's actually three-- we want to talk about editing. So we want to understand how designs can fail, what the differences between an Error and a Warning is,

understanding what can cause these failures, and then how to fix them.

So first, let's talk about Errors and Warnings. In Fusion, there are two kinds of failures that can happen. There is an Error. An error state means the feature involved can't compute at all. So something has gone catastrophically wrong with that feature and you're not going to get any result out of it. That's fairly straightforward and obvious.

The Warnings are interesting. Warning in Fusion means something's gone wrong with this feature, but we can still more or less compute the thing that we're trying to compute, usually with some cached geometry. And so let's look at a couple examples of that. All right.

So this is, by the way, this is the standard developer design here. And you'll see that we've got a Fillet that's failed. Both Errors and Warnings have the same review warning. There's a review warning that lets you-- another sign of a developer build here. Hide the text commands. So you can see that this fillet has failed. The reason is, kind of hard to see, but the edge reference is lost. And the compute failed.

And you'll see there's no fillet. Right? This is truly an error case. And then if we look at another complex design, this one has got an offset work plane that's failed. Again, I can review the warning. This is another reference failure. So in this case, you'll see that the offset work plane exists. And it exists because it has cached geometry. So Fusion caches enough geometry so that it can re-compute what it can.

And this is both good and bad. It's good in that anything downstream from that can continue to compute correctly. So if I had a sketch on that work plane and an extrude on that work plane and a fillet on that extrude, all those things would succeed. It's not like you get this cascade of hundreds of errors because of this one lost reference.

But it's something that you really need to be conscious of because it's using this cached geometry. So if I go back and edit the sketch that created that block, you'll see that we're using the plane of that face when that work plane was created. So even though that work plane is, quote unquote, successful, you've lost some design intent. Because clearly the intent was to make that offset from that plane.

**AUDIENCE:** But can you reattach it?

**JEFF STRADER:** You can, yes. We'll get to that in short order. So to understand failures, it really helps to understand relationships and references. Because in my experience, 80% to 90% of failures

that you get are from lost references. And so you need to be conscious of when you are creating these relationships. Because each one of those that you create is a potential point of failure.

And so anything you do where you select geometry in a command is probably creating a parametric reference that can then fail. So things like projecting an edge into a sketch, sketching on a face, offset a work plane from a face, those create those relationships.

Some references are more stable than others, and it's important to understand and use the more stable of the ones that you can. And then we'll talk a bit about what sort of edits can cause references to fail. OK. So this is an interesting topic. Some references are more stable than others. And in general, you want to use as stable of a reference as possible.

So this list is not exhaustive, but it gives you kind of a feel for the things that are more stable. So the most stable thing that you can pick in all of Fusion is the origin work geometry. If you sketch on an origin plane, if you extrude to an origin plane, that thing I can guarantee you will never fail. Those things never get deleted. They never fail.

And then second, sort of working down from that, is other work geometry created from those things. So if you offset a work plane from an origin plane and sketch on it, that's also never going to fail. Sketch curves and points can be very stable as well. The only way that-- if you're projecting something into your sketch and you can project another sketch curve, that's not going to fail unless you go and delete that sketch curve.

The least stable references are unfortunately the ones that are the most commonly used, and those are the BREP references. So a face, edge, body. Even within that group there is some hierarchy. A body reference is more stable than a face reference is more stable than an edge is more stable than a vertex.

So let's look at another quick example. That, so this is an ever more complex design. So here I've got an extrude, a two-face extrude from an offset. And this is contrived, but hopefully you can get the idea. So the sketch is here. I picked this face as the termination face. All right? Pretty straightforward.

Now I'll introduce an error into this design. And this is my favorite way to introduce an error. If you edit the sketch, delete a line, and recreate it, OK, that looks like it's the same sketch. But to Fusion, it's not. And so what you'll see is now that extrude, that two-face extrude has failed

because it can't find that face.

If you're interested afterwards, I can explain why this is. But for now, realize it happened. So we don't want to do any of that. Undo all that. Let me edit this feature. And instead of extruding to the face, I'll pick the body and extrude to the body. Now if I go in and introduce my failure again, no errors. So that's just sort of illustrating this, it's easier for Fusion to find an entire body than a specific face. So remember that.

OK. So what can cause a reference to fail? The answer basically is you. Right? I see also lots of posts that say something to the effect of, my design started randomly failing. I can pretty much guarantee that that does not happen. Right? You caused it. Whether you remember causing it or not, you caused it. So really the only way that these things can fail are from design edits.

So let's look at some quick examples. We've seen one. If you edit a sketch and you delete and recreate geometry, that can cause a failure. But it's not limited to that. So here we've got another example. So this is another offset work plane where I picked this face. If I go back and edit this space sketch and make some sort of edit such that that face doesn't ever come into existence, now that work plane has failed.

So there's all sorts of edits that you can make that can cause failures. If you edit a fillet and remove an edge and you've got something, you've got a work access through that cylinder, that's going to fail. So it helps to kind of be aware of that.

OK. So how do we fix things? And these pictures are from my favorite website called There, I Fixed It. It's fantastic. So we don't want to really fix them like that. So the rules that I find helpful are, first of all, fix these errors as soon as they happen. And again, this is somewhat of the danger of the warning, right? You see a warning, you think, ah, it's just a warning. I don't have to fix that.

But as we saw, you've actually lost design intent. So as soon as you see these errors, you know, go back and fix them. The other trick that we've learned is to use Compute All as kind of a design checker. There's a little story behind Compute All. It was really added as a developer tool. And our goal was to not release it. Because you shouldn't ever need Compute All. If we've done our job right, Compute All is not a necessary thing.

But we've kind of found this design rule checking aspect of it. If you have your design that you

think is in a good position, a good state, and you do a Compute All and you get no errors and warnings, your design is good. You should save it at that point. I've got to the point of even putting a little label on it when I save, saying Compute All succeeded. Because then you can know you can go back to a good place.

All right. So once you have errors, how do you fix them? And it's mostly subsequent editing that can fix them. So we'll look at some simple cases and then we'll look at the hardest one, which is sketch geometry. So here's a case where we've got some warnings. OK? I introduced this using my Edit Sketch technique. But I've got three work geometries now that have failed.

And so the first thing to do is to review the warning, especially for these warning failures. Because they will give you hints. In this case, it's pretty obvious what was going on. But you can, hopefully, extend this to larger designs where it's not quite so obvious what geometry was picked. So this error, the warning message shows you that it was a face that was here. In this case, the fix is fairly simple. You edit the feature, pick a new face, and now that plane is fixed.

And similarly, review warning here. You can see that that one was from a horizontal edge. And so you can usually guess, Edit Feature, pick a new edge, and now you fix that. So oftentimes you can get out of these things just by editing the features that failed. And you know, at the risk of revealing that I'm a software engineer, this is very similar to Compile Errors.

If any of you do software, usually you fix the first compile error, and then you re-compile because often that will fix downstream errors. Same is true here. Go back to the first error, fix it, and then see where you are. OK.

The hardest one, unfortunately again, is the most common. And that's failed sketch reference geometry. I would say half of the failures that I see usually involve sketch geometry that's failed. So why is this hard? If you just edit the sketch, OK, first of all, this yellow color indicates failure in sketch. This is something that we're working on. We're going to improve the UI here.

But your first instinct is to just go and delete that curve. Well, as we saw, you've lost even more design intent now because I've lost the dimension. So you don't want to do that. In this case, what I usually do is I'll break link on that projected edge. I should have first reviewed the warning so I could see where it came from.

So now this thing is free to go. It's free to move. But we still maintain the design intent. And then I'll project the edge that is where it should come from. And then in this case, this doesn't

always work because-- I mean, it works fine for lines because you can make them collinear. But now I've sort of re-established all the design intent here.

So if I go back and edit this sketch, everything is now correctly associated again. So you can't always use collinear, but you can use other sketch constraints to sort of put the broken one back with the newly projected thing. I think I'll turn it back to Peter here. And if we have time, we'll come back and talk about organizing designs. But I'll let Peter talk about configurations.

**PETER DOERING:** Yeah. And I think I'm going to hustle through that pretty quickly so we have time for the sketches. It's often discussed on the forum that you can't do configurations in Fusion 360. And that's not really true. You can do configurations light. I call it light.

You can't design, let's say, you can't design a screw and have it one length. And then just have a table defining another length that makes you another configuration. You can't do that. But what you can do is actually feature configurations. And there are two variants of that that you can do. Those photos are also, those images are also going to be in the handout. They should be self-explanatory on their own, but there's a little trick to that.

I just have to find those files. Did we open those?

**JEFF STRADER:** I don't think we opened that one.

**PETER DOERING:** All right. So you basically have your base component. I know this is pretty simple, but it can be more complicated. This is, you have your base component. That's here on the left side. Then you add some features to it to make your second component. And then you have a third component that also uses the same base geometry with different features.

And when I edit the sketch here-- let me do that differently because it's so nice. I turn this sketch back on, show dimensions. Change this to 40, and they all snap to the same position. So let's undo that so we can actually see what we're doing here. So they're all driven off the same geometry. So you can make configurations.

So how do you do that? I think that's one of the better kept secrets of Fusion 360. You would never discover that on your own, unless by accident. So the first thing is, you make a component. Right? Rule number one. You make a sketch. You extrude it. And you have your first body in that component.

And you make a second component and here's a trick. You take that body from the first

component and just Copy and Paste it into the second component. We all know what the difference between Copy and Paste and Copy and Paste New is? Right? Copy and pasting makes a, it's not a link component, it's an instance. That identical components, you edit the one, you edit the other.

There are always the same. Copy and Paste New makes an independent component. So you copy all the sketches from the first component, everything that's in that first component if you made a component first, and then you can edit that independently. I can't repeat that often enough.

So in this case, it's a different form of copying. You take a body from one component into another. And then in that other component, you can just add some features, add some features to it. I need to move those out of the way. Whatever. I think I screwed something up here. That's OK. We'll undo. Nicest function ever. So as you advance in your timeline, you add features to your body.

What did I do? I told you, I did exactly what I tell you not to do. I moved the body within the component. There you go. So you advance through your timeline. You add features to that second component. And then you create your third component. That's not together again. That's fine. We don't need them anyway. We'll just hide them.

And there's your third component, derived off that same initial component, just by copying and pasting that body. So there's a variant of that. So instead of copying the body from the first component into the second, and then from the first component into the third, you can also chain link them, so to speak.

And that's actually what we're doing on the left image there. And what would that be useful for? In this case, it's a machine component, goes through several production steps. Goes through several production steps and each production step needs its own sets of drawings. And maybe some of them might be outsourced.

And in that case, what do you do? As you see, you don't copy the component from the first into the second and then from the first into the third. You just copy the first body into the second component. And once you're done with that component, you copy that body into that first-- into the third component. So the question is, of course, if you forget something, what do you do?

How do you edit that? How do how do I edit the first feature, the first component? Well, as I said earlier, when you edit the sketch it's basically like time travel. So if you want to add a feature to that first component that you also want to have reflected in all the later components, what you have to do is you have to go back in the timeline to before when you copied and pasted that component. Right?

Before you copied and pasted that body into the second component. When you add a feature then, it'll be copied and pasted. So the body that you create is basically the output of that process here down in the timeline. If you think about it like that, and that same thing works for the second component. The body that you have there is the process of all the features that you have in the timeline for the second component.

If you take that and put that into the third component and so forth, you can create your configurations. So that's configurations. Then the next part I want to talk about is the limited, as I said earlier, they are limited to feature configurations. Now what I said earlier with a screw-- well, OK, you can make a Save Another Feature and combine that into that second component. So you can do that too if you want to. It's not as convenient as a different dimension, but can be done.

The second part that I want to talk about is Linked Components, and when they're needed and when they're not needed. So fairly often on the Fusion 360 forum, I see designs that look like this. I wouldn't design like this because it's a bit of a catastrophe. You can, I mean, there's nothing wrong with, inherently wrong designing it like this. It can just get very cumbersome.

Because if you have those staggered and deep update makes that a little easier. You know, you have to open all those files and edit them, save them, and then go back to the original design, click on Update. And you know, if you're in your design flow, that's very cumbersome. And as you've seen with that lamp design that I had earlier-- I'm not sure if I'll find that now. Well, I said I had 400 components and I didn't have a single linked component.

So one feature of Fusion 360 is certainly that it doesn't separate between an assembly file and a component file. It's all in one file. So you can build, obviously, you can build, as you've seen earlier, you can build a complete design with all, with complete necessary structure in a single file. So when do you need linked components? When a component is to be reused in another design. That's a little obvious.

And-- I made that and big because that's a logical and-- when modifications to it are expected.

And when the modifi-- when you actually want those modifications to be reflected in that design that you linked it into. That's not always clear that that's needed. So we also need to talk when is it not needed.

And it's not needed when that component is a unique part of that design. You can take a look at that image here earlier. Anybody that's ever designed machinery, you know there's a bunch of purchased parts in it. You don't need to link those. There's no need to. There's also a whole bunch of parts in there that are unique to that particular design that you probably would never use in any other design. You don't need to link those. It just makes your day harder.

If you make a component first, you know, you have the correct structure not having to do that. So of course, when is it not needed. As I explained earlier, when the design that the part is inserted to needs to reflect an as-built status. I've built a lot of factory automation systems. We needed to know what we put in the machine at some point in time.

We didn't want automatic updates because if we needed to exchange a part-- a sensor or even something custom designed-- if we needed to exchange a part, we've used lots of modules. We didn't want that to automatically update because, you know, you take your module out of out-of-stock and put it in that machine, it doesn't fit because you changed the module. They didn't want that to update.

So that's a little bit about linked components. In that case, that design would not use that many linked components. There's nothing wrong with linked components. They certainly have their use if you work in a team environment where a lot of, several people work concurrently on the same design. You know? Then you probably need linked components and linked assemblies. But in that case, it's more important how to define the interface between all those. That's a whole different topic.

With that, hopefully we have a little bit of time for--

**JEFF STRADER:** Yeah, sketches. OK. So I'll take a few minutes and talk about sketch tips. Sketching in Fusion is sometimes a source of some frustration, but there are some techniques that can definitely help make your experience better. So the first one is to keep your sketches small and simple as possible.

Look at a couple of cases. So this design came to us from a forum user. And what they noticed is, they got the sketch to this point and they were just trying to add this last fillet. And it

took a fairly long time to add the fillet. I don't remember how long. It's a little embarrassing.

But the point of this is that this is a fairly complex sketch. You'll notice there's a lot of symmetry constraints. There's a lot of equal radius constraints, a lot of tangents. There are other ways to do that that will make your life better. So first, I lost one. I lost yours. OK. So first, Peter was the first one to respond to this user on the forum, and came up with this approach.

So he realized that you could make this whole design without having to make all four complete parts of that cross. And instead, what this sketch does is make a series of extrudes of that section, and then uses a set of patterned mirror to build the design. And then combine in an inside fillet. But the key takeaway here is that this sketch is going to be much more pleasant to interact with.

**PETER DOERING:** It's also much less work.

**JEFF STRADER:** Right.

**AUDIENCE:** Can it be just one sketch [INAUDIBLE]?

**JEFF STRADER:** Right. So that's the other approach. So this-- good comment, [INAUDIBLE]. So this was my version of this sketch, of this design, is to start with just a single sketch that does the basic cross shape. And then extrude that. And then offset, sketch on that face, offset the profile, repeat that all the way to the top. And then add some fillet features to round out the design.

Both of those approach will make your entire design experience much more easy to deal with. So keep your designs as simple, keep your sketches as simple as possible. And just to reinforce the point, this one came in this morning. This was yesterday. This sketch was taking, well, what the user was saying was that to change a parameter, he let it run for more than an hour and it didn't finish.

But if you look at this, it's kind of hard to tell but it's the same basic thing. There's four pieces of geometry that are reproduced. I would never do this in a sketch. Do it with Features instead. OK, so a few points about working with splines, and then we'll have some questions. So Fusion splines are another source of common questions. So there's a couple of tips there.

The first is to use as few fit points as you can get away with. And so this one is one that I ran into the first time I tried to do this as well. So this is a very common workflow on Fusion. You insert an image and you want to trace it. And the natural instinct here is to, obviously, it's going

to be more accurate the more points you have. So that's the first instinct that everybody does.

And you can certainly do it that way. But that spline will be very hard to edit. Right? If you want to make any changes, you'll have to move bunches of control points, or fit points at once. But it's not just that. The resulting geometry is not so good either. So if I turn off that, that's the result I got from extruding that. And it doesn't look too bad if you don't look too closely.

But if you turn on Curvature Analysis on that edge, you'll see what a mess, what a mess you've got. And that's because each one of those fit points is affecting the curvature. So instead, so instead, you can get that same shape with a lot fewer fit points. And the resulting body will be better. And if you look at the Curvature [INAUDIBLE], you can see it's a much smoother transition. So the last thing--

**AUDIENCE:** Can you just funnel these [INAUDIBLE]?

**JEFF STRADER:** Yeah. The bodies or the analysis? Yeah. So the last thing I want to talk about is using spline handles. Because there are some UI challenges there. So this is a fairly simple 2D spline. And this is the spline that you'll get if you just place fit points and let Fusion fit the curve. But you'll notice that there's these handles here. And you can interact with those, but they need to be activated first.

So you can activate them manually. Once they're activated, you can drag-- you're changing the direction and then the amount of influence, as well. You can also implicitly activate them by just dragging. You'll notice that there's another handle here and it's called the Curvature Handle. And you can activate the Curvature Handle once the Tangent Handle's activated. And then you can control the curvature at that point as well. This works on 2D and 3D curves. But this UI is not all that discoverable. So it kind of helps if you can see an example.

**AUDIENCE:** Should've worked out that handle then?

**JEFF STRADER:** So when you select the curves, all of the unactivated handles are shown in green. And once they're shown in green, you can select them and activate them. Yeah. And it's another thing that we're actually actively trying to work on the experience for. So that's basically what we had. If there's questions, we're happy to take them. I encourage you to look at the handout.

**AUDIENCE:** [INAUDIBLE].

**JEFF STRADER:** You can do that as well, right.

**AUDIENCE:** [INAUDIBLE]. Yes.

**AUDIENCE:** There should be no problems then.

**JEFF STRADER:** Right.

**PETER DOERING:** It's still more work.

**JEFF STRADER:** Well, yeah.

**PETER DOERING:** The reason why the user didn't do that, when you offset, that inner radii get smaller. And he made that design challenge just a little bit harder by wanting to keep those, the concave radii, the same.

**AUDIENCE:** Well, you need to do the small one first and then start by sending it out.

**JEFF STRADER:** Yeah. Right.

**AUDIENCE:** [INAUDIBLE].

**AUDIENCE:** But if the interior-- the inside radius [INAUDIBLE].

**PETER DOERING:** I need to grow a few feet here. Those inside radii are all the same radius. They don't grow. That's what presented the challenge to him.

**JEFF STRADER:** But you're absolutely right, with that exception. Because these are concentric, the outer, you know, the rounds are concentric. And so you can do that too. You can do this without the interior fillets. Yeah. So you're thinking the right way. Because there are lots of ways to do this that are better than this.

**AUDIENCE:** [INAUDIBLE].

**PETER DOERING:** Exactly. All right.

**JEFF STRADER:** Any other questions? Feel free to come up afterwards. But thank you for your attention.

[APPLAUSE]

**PETER DOERING:** Thanks.