Hello everyone. My name is Krithika, and welcome to From Crisp Design to Clean Documentation: An Introduction to Fusion 360 Drawings. It's a long name, but anyway.

OK, so this class is an introduction to the documentation component in the Fusion 360 application. And it's generally intended for beginners to intermediate users, but don't worry about it. If it doesn't sound like you, that's OK.

So what we're going to be doing in the next hour or so is to explore the full feature set that the Fusion Drawings offer. So that's everything including the user interface, the commands, and all of the functionality. We learn about the various access points and shortcuts in the drawings—basically, a couple of different ways to do things so that you can work efficiently. We'll also see how we can create an edit drawings, and once you've created those production-quality drawings, we also see how we can share them with downstream consumers.

OK, so before I go ahead with the content, a little bit about myself. I'm a computer engineer by degree, and a software quality assurance engineer at Autodesk by profession. I've been working with the Fusion 360 drawing steam for the last couple of years or so.

And prior to that, I used to work with AutoCAD Web and AutoCAD a little bit, so I have some experience there. And in my free time, I like to watch movies or listen to good music, and if I'm not too lazy maybe do some yoga. And I'm really excited to be here today to present drawings to all of you.

OK, so the agenda for the session is pretty straightforward. Firstly, an introduction to what Fusion 360 is and what Fusion Drawings is. Then we'll spend some time looking at the user interface and the features in the drawings environment, after which we'll move on to some aspects around working with drawings like collaboration and version management. Then we'll wrap up with some takeaways and open up the floor for some Q&A.

OK, has anybody seen this slide before? OK, I'm not surprised. So, I just borrowed this from one of our marketing decks, so I wouldn't be surprised if many of you are familiar with this.

OK, so before I talk about it, how many of you have used Fusion 360 or are actively using Fusion 360? Wow, that's a good number. OK, great.
So for those who are still new to it or getting familiar with Fusion 360, it's cloud-based 3D CAD, CAM and CAE application that's been designed to meet marketing needs end-to-end. So you can create your models in Fusion 360, and that can be solid models, or you can work with open meshes, or I guess you might have heard about organic display modeling in the keynote speech or yesterday's innovation forum, right? So you can do all of that in Fusion 360. And once you have that model, you can take it through rendering, animation, you can do simulation on it. And depending on how you want to fabricate it, you can set up your machine tool parts or connected 3D printers, create drawings out of it.

And the whole application has been designed to facilitate a lot of collaboration and rapid iteration. So there's tools for that too, and then there's the data management or BLM aspect of it, so there's a lot going on in Fusion 360. And in the next few minutes, we'll be focusing mainly on the drawings.

A couple more things-- this whole cloud support has a lot of benefits to it. You may have heard that simulation or rendering makes use of the cloud capabilities to do some of its computation. One of the advantages is that you get anytime, anywhere access to your content.

So to begin with, actually, Fusion 360 is a cross-platform application. So you can work on Windows or Mac. And to go along with that, you also have review apps which are available for iOS and Android, where you can view your designs shared and do all that stuff. If you're working from a browser, you have web access as well, from A360 Drive or Viewer, you may have heard of that too.

OK, so that's an introduction to Fusion 360. Now to introduce Drawings, I'll play a short video that the marketing team had put together earlier. I think it was created a while ago but it still holds good.

[VIDEO PLAYBACK]

[MUSIC PLAYING]

-You've told us that 2D drawings are still a critical part of your design process. So just as we were first with 2D on a PC, Autodesk is first at the cloud with Fusion 360 and 2D drawings. With Autodesk Fusion 360, you can create a drawing from any model in your fusion project. And I'm not talking about a watered-down, reverse-engineered, mini-me version of a DWG-- a real, Autodesk DWG file. Autodesk Fusion 360 has all the features you would expect to have
to quickly lay out views, create section views, add dimensions, notes, custom properties on the title block, all the things you need to create a production drawing with Fusion 360.

And of course, we didn’t stop at PARC drawings. You can create assembly views just as easily. Grab a parts list, slap down some balloons, and you’re done. And since nobody plays better with others than Autodesk, spit this drawing out as a PDF just as easily as DWG files. Just go visit Autodesk.com/Fusion360, and get started today.

[END PLAYBACK]

KRITHIKA

OK, there you go. So to expand on that, a Fusion Drawing is a document that's derived from a Fusion 360 model. And at this point, you can create ASME or ISO drawings. I think in the future, we’re also looking at expanding through GB, DIN and JIS. But at this point, we have ASME and ISO.

The thing about Fusion Drawings is it is built on AutoCAD’s drawing engine. In fact, I think you can even extend to say AutoCAD Mechanical's drawing engine. So you have the world's best documentation tool working for you.

At some point, documentation or drawings used to be the most requested feature for Fusion 360, and we were kind of the last company to become a part of Fusion 360. But today I think we’re at a pretty big and distributed team. I come from the office in Singapore. We also have development teams in Shanghai and the US, and a couple of members in other locations as well.

OK, so moving on to our next section, the user interface and features. So this section is going to be more of a demo. I really have the slides just for reference in case I forget something.

OK, so to access Fusion Drawings, we start with a model. And you can create drawings either from the model or from animation keyframes, and you can create either pad drawings or full assembly drawings. Let's take a look at how to do that.

So I’m switching to my Fusion 360 here. And I have the model of a stop valve. And what we're going to do now is to create a pad drawing from one of the components and lay down some views, annotations, see how we can edit things so they look better. And once we're happy with that, we also move ahead with things like exploded views, bill of materials with a full assembly drawing.
OK, so let's begin. This is our modeling environment. And what we can do is just go to the File menu, say new drawing, and from design. And you'll see that by default, it selects the full assembly, but you can uncheck that and select individual components that you're interested in to create the drawing.

The other way to do this would be directly from the browser tree. So this thing you have here on the left. So let's say we're interested in this component. You can just select that, you can select more if you want. Right-click and do the same thing, create new drawing.

So that takes us back to the same dialog. And you'll see that now the difference is one component. And underneath that, we have some configuration settings for how we want to create the drawing. So I can choose my standard and that can be ASME or ISO. The units can be inches or millimeters, and the sheet size, which really depends on the standard.

OK, I'll just leave it at the defaults and hit OK. And this takes us to the 2D drawing workspace. And by default, the base view command gets launched, which is a 2D projection of whatever we chose in the model to create the drawing.

I'll take a few seconds here to explain some of the important UI elements that we have. So right at the top, we have the Quick Access toolbar with controls like show or hide the dashboard where you have your files. Then file menu, save, undo, redo, that sort of stuff. Then we have the toolbar, which is divided into a few different panels based on the type of commands. And with each panel, you can click on the dropdown menu and that would give you more options underneath that category. And then right at the bottom, we have the navigation bar with ban, zoom, fit controls.

OK, back to the active command. Now we have the base view command, and we can specify some location for placing this. I'll just click somewhere.

To the right, we have the command dialog. And this is something that comes up with a lot of commands where you can specify certain properties that you want to change. So in the case of the view dialog, we have the orientation.

So when you expand this, you have the standard octagonal and isometric views. But right at the top, there are also these little blue icons. They present custom name view. So you can create custom orientations if you want. And I think there's a note in the handout on how to do that, so let's skip over that.
So I'll just choose right view for this, maybe a couple of other settings like the style. Just go for visible edges, make the scale a little bit bigger. And there, that's our first view. And once we have this, it's really easy to go ahead and create more views out of this, so I'm just going to go back to the views panel, create some projected views, so it's a top view and an isometric view.

So whenever you're done working with a particular command, you can hit Enter. Another thing that you can do is right-click contextual marking menu. And this is really similar to the other Fusion 360 workspace assessment, so if you've worked with it before, you'd probably be aware of it already.

So, the marking menu is contextual, and it contains options based on the command that you're working with, like you only have OK, cancel, a couple of other commands here. And this one right here is going to lead to a secondary marking menu with some more annotations. And I find this really useful, because you don't even have to click and work with it. If you kind of know where the commands are, you can invoke what is called a right-click gesture like that. Just do that, and that works.

OK, so we've placed a couple of views now. Let's quickly add some annotations. I'm going to zoom around the base view.

So moving to the annotation panel, the default dimension symbol that you already see here, it's a small dimension, too. So it's going to adapt itself based on what edge you're picking. So if you're hovering over a straight edge, it's a linear dimension. And even if you pick points, actually. Let's try that.

So if I pick two points, I can have an aligned dimension this way. Or I can just flip it around to get those octagonal dimensions. Or it can hover over a circular edge and get a radial dimension. So it's adaptive that way.

So maybe I'll quickly place one linear dimension here. And you can continue placing more, but if you are in need of your standard dimensioning tools, those are available here in the annotation menu. So here we have the linear, aligned, angular, radius and diameter dimensions as well.

So I'll maybe place some linear dimensions now. Like that. Again, you can continue doing it or end it whenever you're done.
So to create dimensions more quickly, we also have the baseline and chain dimension commands. So that helps too. So you can just do that. Maybe I’ll move this down a little bit. OK, so that looks better.

So back in the annotation menu, we have a couple of other annotations. Right at the bottom, we have the leader in text, and right at the top we also have centerline and center mark. So let’s create a centerline.

I tried to create an axis around here, so let’s do that. And you can just select it and edit the grips to extend it out further if you need. So I’m just going to let it go through the midpoint of the other side as well. And then you can continue doing that. You can place annotations on the center line, do what you need to.

Let’s move to the view on top, which is this one. What I’m going to do know is OK, we can quickly place some center marks here, like that, or place for the individual holes, too. You get the point.

We can also place some diameter symbols here. So just go ahead and invoke the leader command, OK. So diameter there, and maybe something for the holes, too.

So that looks fine, but what would be more useful is if we can add more meaning to these annotations. And you can do that by double-clicking and editing the dimensions. So for instance, this one right here, you can sort of make it look like a hole if you just double-click. And that brings up the in-place text editor. And that’s true for any text entity that you have, whether it’s a leader or text or title block. Just double-click it, and you’ll be able to edit it.

So, let’s add something here. So maybe something like this. And another thing is, this dimension here, though it looks like a linear dimension, we can actually see that this is a circular edge.

At this point, we don’t really have the support to get the diameter symbol working right away—that’s in progress. But until then, you can fake things by just double-clicking. And if you’re familiar with the AutoCAD way, you just go and do percentage, percentage, C, and that adds the diameter symbol. That’s a temporary workaround.

OK, so one other thing that you may have noticed is when you edit dimensions, you also get the command dialogue on the right with a lot more options that you can tweak around. So you
have the primary position, alternate unit, [INAUDIBLE]. There's also the presentation and the inspection values that you can play around with. For now, maybe we'll just add some tolerance.

Yes.

AUDIENCE: Could you be clear about what you're doing when you're moving [INAUDIBLE] around?

KRITHIKA SUNDARARAJAN: Uh-huh, sure. OK, I'll slow down a little bit.

AUDIENCE: What key are you pressing?

KRITHIKA SUNDARARAJAN: Oh, what key I'm pressing.

AUDIENCE: [INAUDIBLE]

KRITHIKA SUNDARARAJAN: Uh-huh, OK. So I'll do that, thank you.

AUDIENCE: [INAUDIBLE]

SUNDARARAJAN: So I think what I did was to just select the view and move it around with the center grip. So with each of these entities, if you can select them individually, that are these grips that you can manipulate. Like for the center line, I selected the center line and I just moved this grip out so you can extend it.

You can do that with dimensions, too. And for things like views, you have the center grip, which will help you move them around. Yeah, thanks for the tip.

OK, so at this point, let's maybe create one more view. So back to the views panel, we have a section view and a detail view as well. You can just place a section view. Or the other access point is when you right-click, we talked about the marking menu.

So when you right-click, you also get this thing called an overflow menu, which is pretty much a representation of the toolbar, so you don't really even have to go to the toolbar every time. You just right-click and you can choose the options from there. So I'm going to choose section view from here, select the view that I'm trying to section, and make a section line, maybe right through the center. Like that.
When you're done, right-click Continue. So I'll just place it somewhere. That overlaps with the title block a little bit, so again, let's move this up.

So I'm going to select the view. And things like projected views and section views, they're aligned to the section line or the view. So I'll just break the alignment using the Shift key.

You'll get a tool tip about that as well, the dynamic ground. So I'm just going to move it there. OK, so that looks better, I guess.

So at this point, let's go ahead and save the drawing. And when we do that, I'll just give it some name. So this is going to get saved to the cloud, pretty much like the design that we're working with. So when we start saving it, you'll see that new thumbnail is being generated in the data panel.

Another thing that happens is if you go to the File menu, there's an option for related data. And now this is going to create a link between the model and the drawing. So you can switch back and forth from this. So I'm just going to choose the model now. And that's same for the modeling environment, too. So if you go to the model and related data, all of the drawings that are created from it, you can see from that option.

OK, so let's close this. So the drawing that we just created is fully associated to the model. So when you make any changes to the model, you can update the drawing as well.

So let's take a quick example. So let's say we are going to modify these spaces. I'm going to add--I'm pulling it out and added a 1-inch thickness. So when you're done with your changes, you can just save it and switch back to the drawing. And when you do that, you'll notice a notification at the bottom right, which says changes have been made to a reference design.

You can update the drawing from the bottom in the application bar. So that's the same thing that you see here. So this one here is the reference menu. And when the drawing becomes out-of-date with the model, you'll see there's an option here called Get Latest, which will help you to sync the changes.

So I'm just going to do that, and immediately after that, the views and the annotations will adapt to the change. So right there, that becomes 1.87. So that was a change.

So at this point, I think we can do a couple more edits to make the drawings better and kind of polish it out. So what we can do now is to just double-click edit this isometric view here. And in
the style, there’s an option to make it shaded if you want to do that.

You can do the same thing with other viewers, just double-click and edit just as we did with the annotations. Moving on to the section view here, if you hover not anywhere else, but over the hatch instead, you’ll see the hatch gets highlighted. And when you double click the hatch, that brings up the hatchet it instead. And you can change some properties there, like the pattern. It's same as what you would find in AutoCAD.

So you have-- OK, maybe let’s just choose some pattern. You can change the scale and angle, those sort of things. Yes.

AUDIENCE: Can you have the custom hatches?

KRITHIKA SUNDARARAJAN: At this point, no. We don't support custom hatches yet. OK. What else? What else can we do?

SUNDARARAJAN: Yeah.

AUDIENCE: [INAUDIBLE]

KRITHIKA SUNDARARAJAN: Excuse me, could you repeat--

SUNDARARAJAN: [INAUDIBLE] whole circle, or a whole circle?

KRITHIKA SUNDARARAJAN: Whole circle. Oh like--

SUNDARARAJAN: [INAUDIBLE] line in the circle through your--

KRITHIKA SUNDARARAJAN: BCD sort of thing?

SUNDARARAJAN: Yeah.

KRITHIKA SUNDARARAJAN: No, not yet, again. That's again in the roadmap, but not yet. Yes please.

SUNDARARAJAN: Can you create a custom template?

KRITHIKA SUNDARARAJAN: Custom templates? No, not yet. OK. Hmm, our product owner should be here. Let's find him SUNDARARAJAN: with those questions.
AUDIENCE: So Meredith’s not here because she got [INAUDIBLE], but I can answer questions for her.

KRITHIKA: Sure. I think Andrew and Scott are planning to be here. But anyway, yeah thanks. Were there any more questions? Yes.

AUDIENCE: Can you create a hundred-page [INAUDIBLE] finish from this drawing?

KRITHIKA: OK, so that's multiple sheets, right? That's again, in the roadmap, which I think we can expect early next year at this point again to know.

AUDIENCE: Can you change the title?

KRITHIKA: OK, so you can edit the title block, but you can't tweak around what's available in the properties right now. So you can double-click Edit it, and you can just add values from the command dialogue, like in this case, let me just do something like this. So you can edit what's available, and it's configured according to the templates, but you can change much there, I guess. OK. Yes, please.

AUDIENCE: Is there a way to electronically add [INAUDIBLE]?

KRITHIKA: Electronically add a logo?

SUNDARARAJAN:

AUDIENCE: [INAUDIBLE]

KRITHIKA: OK. Yes.

SUNDARARAJAN:

KRITHIKA: Yes.

SUNDARARAJAN:

AUDIENCE: What [INAUDIBLE]? Are we able to take this into [INAUDIBLE] and bring them here?

KRITHIKA: Import the drawings?

SUNDARARAJAN:

AUDIENCE: Yes.

KRITHIKA: That again, is-- no, not quite possible. In this case, we work from the model. So you can export this and take it into Inventor, but not the other way around.
You can import drawings as canvases [INAUDIBLE].

Yeah. Yeah. That's right.

OK.

So one more thing. So here, we have the different annotations, but the one thing that I'd like to see which is not there is what units they are drawn in. And you can do what we did earlier to go ahead and edit them individually. But instead, if you want something that controls all of them, at the bottom here next to the navigation bar, we have some options there for the drawing settings.

Like here, there's a sheet settings, which will let you change the sheet size. Or next to that, there's annotation settings, which will help you change things like the font, text type, positions, annotation units, and trailing zeroes. So in this case, I'm going to choose annotation unit on. And that updates all of the existing annotations and newly added ones as well.

But if there are settings like this that you work with all the time, then you probably don't want to do it with every drawing. So you can change the global preferences in that, which is available here. So this menu here, you can go to preferences. And under the general section, there is one for drawings. And you can set those values here. And every time you create a new drawing, they'll make use of these values.

OK. So let's switch back to the slides now and see what we've done. So we just looked at how to access the fusion drawings. We took a look at the user interface. If you're on a Mac, trackpad gestures will work too. Then we went into creating drawings and model associativity.

OK. Yeah. So we saw how to get the latest changes to get the drawings in sync. But sometimes when you make drastic changes to the model geometry, it's possible that your annotations get disassociated, and they get that tiny yellow badge there. And when you click them, once again, you'll get a contextual marking menu. And you can choose what to do with them. Yes.
KRITHIKA: No. No.

SUNDARARAJAN:

AUDIENCE: [INAUDIBLE]

AUDIENCE: So you can't change dimension here and then change the [INAUDIBLE]?

AUDIENCE: Correct.

KRITHIKA: No. It's a one way associativity at this point.

SUNDARARAJAN:

AUDIENCE: [INAUDIBLE]

KRITHIKA: OK. Moving on. We also looked at how to edit drawings.

SUNDARARAJAN:

So the next section is about exploded views. So just as we created drawings from the design, we can also create drawings from the animation environment. So here back in the model, you can switch to the animation workspace like this, and that's the presentation model in Fusion 360.

So I've already done some things here. Maybe I'll just play this for you. Go ahead. Play. OK. So you can chose what to do with your components. You can go to the transform menu here and set things up. Or if you want something quick, you can also just right click. There's an autoexplode option to create that presentation for you.

So once you do that, we can now create a drawing out of this animation workspace. So it's the same thing. Go to the File menu. New Drawing. And from animation-- hmm. Some change. OK. Never mind. I'll just save it.

So that there said something was changed. So I just saved the change. Doesn't matter. So this brings up the same dialogue that we saw earlier, except this time the reference is not to the design, but to the animation storyboard. And the rest of the options are the same. So you have the standards and units and sheet size. So we're just going to create the drawing now. And this time, the base view that gets created is actually the exploded view. Let's place it somewhere and go ahead and maybe make the scale a little bit bigger. OK. So that's exploded
view right there.

So the rest of the options and everything remain the same. The difference is, when you create the drawing from the animation workspace from that particular storyboard, your base view becomes the exploded view. OK. And now you can quickly start placing some balloons. You can find that in the bill of materials panel right here. I'm just going to add a few balloons.

AUDIENCE: Is it possible to rotate [INAUDIBLE]?

KRITHIKA SUNDARARAJAN: No. We can't. But I think it's in the pipeline. It's not available right now.

SUNDARARAJAN: OK. So you can place some balloons that way, and you can also place a parts list, which is already populated with the values from the model. So you'll see the columns here. Things like quantity, for example, represents the number of copies that you have in the modeling environment. Part number, description, and material also come from the model. So if you want to change that, just switch back to the model. Let me go to the modeling workspace.

OK. So let's take this one for instance. Just right click. And you can change the physical material from here, and you can do the same thing for the other properties. So there's this properties option. You have part number, part name, and description. So part name is usually what you create the component with. The rest of the values can be configured from here as well.

OK. So going back to our drawing, so those are the columns. If you look at that item number, that gets assigned by default based on the order in which you create the components. But if you want to change that, back to the bill of materials menu, there's an option called Renumber Balloons. And this will allow you to renumber the balloons as well as the corresponding entries in the parts list. So you can do that. Or there's an option to align balloons as well. You can quickly select the balloons that you need to align and just disperse them as you wish.

OK. So the exploded views was a feature that we just introduced with our release in November. And one other thing that we introduced in November is the GD and T symbols. So right here, we have the symbol panel, which has the datum identifier and the feature control frame options. Maybe let's take an example and create one.

So I'm switching back to the parts drawing here. Once again, it's notifying me because something got saved. But let's just ignore it. OK. So let's create a datum identifier now. So the
options that you get in the command dialogue, once again, are template dependent. If it were ISO, I think you also have the datum note and thread note options. So let me create a datum ID and a feature control frame.

OK. So let's specify that this is parallel, specify some tolerance. So that's a simple feature control frame right there. So you have more options. Just as you would in AutoCAD Mechanical, you have the all round top note, bottom note, and first frame and second frame options here that you can work with.

AUDIENCE: [INAUDIBLE]

KRITHIKA SUNDARARAJAN: Sorry?

SUNDARARAJAN:

AUDIENCE: [INAUDIBLE]

KRITHIKA SUNDARARAJAN: Oh. The right click one. This one?

SUNDARARAJAN:

AUDIENCE: [INAUDIBLE]

KRITHIKA SUNDARARAJAN: This one? I just expanded this one out. Is that what you're asking about?

SUNDARARAJAN:

AUDIENCE: How did you get to the [INAUDIBLE]?

KRITHIKA SUNDARARAJAN: Oh, OK. So this one right here-- so there's datum identifier and feature control frame options.

SUNDARARAJAN:

Let me do that again. So when you start creating this, you can right click Continue. I generally tend to do that. So once you're done pleasing that feature control frame symbol itself, it brings up the options that you can work with. So that's the command dialog there. And the one thing that I did want to mention was with each of the text fields, you get this tiny little dropdown with the symbol palette that gives you access to special characters that the standards has defined. So you can place some right there. OK.

So I guess that was it. So once again, let me switch back to the presentation and move on. So
we just look at how to create exploded views. We go from the animation workspace and can place some bill of materials. So we have the datum ID and feature control frame symbols as well. They're based on these standards. You can see from the screenshot. You can really go crazy with that and create complicated ones as you would in AutoCAD Mechanical.

OK. So I think that brings us to the end of this section, which is the user interface and features. I guess we're ready to move on to collaboration and version management. So speaking about collaboration-- yes, please.

AUDIENCE: [INAUDIBLE] user interface question here. If you're looking at a dimension, how do you delete it if you don't want it anymore?

KRITHIKA SUNDARARAJAN: Right.

AUDIENCE: [INAUDIBLE]

KRITHIKA SUNDARARAJAN: Step by step. I haven't used this since the last update. I tried to delete dimensions or move dimensions. It seemed to frequently result in moving in disassociation.

KRITHIKA SUNDARARAJAN: OK.

AUDIENCE: [INAUDIBLE]

KRITHIKA SUNDARARAJAN: OK. So let me just do a demo. Let's see if that answers your question. OK. So you have these dimensions here, and as with other entities, there are grips that you can work with. So these kind of grips will help you position the dimension. But these things might-- you're moving it out of the view. So that might cause it to become disassociated.

At this point, you can reassociate or delete it. That's one thing.

AUDIENCE: [INAUDIBLE]

KRITHIKA SUNDARARAJAN: Yeah. So I think it's available in AutoCAD as well. It's the dim reassociate command. So you just select that, and then you can specify the points to which you want to associate it back to. So that's one thing.

AUDIENCE: [INAUDIBLE]

KRITHIKA SUNDARARAJAN: Yes. Yes. Yes. So at any point when you do a right click, that brings up the contextual marking
SUNDARARAJAN: menu. And it's true for all the workspaces in Fusion 360.

OK. So the other thing about move and delete-- so here, we have the modify panel with move, delete, a couple of other commands. So if you're trying to move the dimension itself, I guess it would become disassociated like that. Yeah.

AUDIENCE: Because that seems like where you should go to move it.

KRITHIKA OK. So in order to avoid that, you can just select the view. And because all of the annotations are associated to the view, when you move the center grip, everything that's associated or aligned to it is going to move along with it. The other thing that I would do is probably just cross select-- I'm sorry. I just did a right click. OK.

I can just cross select or windows select whatever and just move all of them together. So just move the view alone or moving everything together should work, I guess. And if you want to delete it, you can select it and press the Delete key. Or the other option is, like this one, you right click. There's a delete option here from the modify menu.

AUDIENCE: The only option in the menu up there is designed to do what?

KRITHIKA The same thing. It's the same as selecting it. Yeah. So let's maybe look at an example. Delete.

SUNDARARAJAN: Select this one. You can select multiple ones if you want. And when you're done, just press Enter or right click OK. Yeah. That's it. Was that OK?

AUDIENCE: Yeah. I think it has been a little confusing because when you [INAUDIBLE] and you click on it, it doesn't delete unless you also right click and select OK. There are multiple steps.

KRITHIKA Right. Yeah. So it's still waiting to complete to see what else needs to be del--

SUNDARARAJAN:

AUDIENCE: [INAUDIBLE] you have delete as an option, it seems like maybe that's the one you should choose. But if you do that, then nothing happens because you just reselected the same thing.

KRITHIKA OK. That's an interesting feedback OK. Thank you.

SUNDARARAJAN:

AUDIENCE: Speaking from a user standpoint, it's a little [INAUDIBLE].
AUDIENCE: Sorry. Just on that, [INAUDIBLE].

KRITHIKA SUNDARARAJAN: OK. Thank you. Yes.

AUDIENCE: There was one situation where I inadvertently manually overrode the name of the view, and I didn't want that. Is there a way to return it to an associative [INAUDIBLE]?

KRITHIKA SUNDARARAJAN: You mean like overrode the name, meaning something like this?

AUDIENCE: [INAUDIBLE] cross-section, and I accidentally typed over the A-A, and I didn't want that. I wanted to [INAUDIBLE] label on [INAUDIBLE].

KRITHIKA SUNDARARAJAN: Undo would work.

AUDIENCE: [INAUDIBLE]

KRITHIKA SUNDARARAJAN: If you realize much later--

AUDIENCE: [INAUDIBLE]

KRITHIKA SUNDARARAJAN: I'm sure there's a way to do it, but I can't remember now. I can get back to you on that.

AUDIENCE: So you just want to reset your [INAUDIBLE] to [INAUDIBLE]?

AUDIENCE: Yeah.

KRITHIKA SUNDARARAJAN: OK. OK. Thank you. All right. So let's move back. OK. So speaking about collaboration-- so once you're done creating your drawings, there are a couple of different ways in which you can share them. One thing that you can do is from the drawings workspace, there's the output panel where you can find options to export your drawing as DWG or a PDF. But you can also take advantage of the A 360 collaboration tools that have been built into Fusion 360.

So you can invite project members and give them access to all of the files in your project. You can do that in the data panel. If you switch to the People tab, you can start inviting people to add them in. Or if you only want to share individual files, you can also do that from the data...
panel. Just select that file, right click, you get a menu to do that. That's it.

And moving on to version management. This again, drawings is going to take advantage of the version management that has been built into Fusion 360, and it works for all the files there. So every time you save, there's a new version of the file being created. And the drawings reference specific versions of the model. And we already saw how you get update the drawings to the latest version of the model using the Get Latest command. So that's fine. But if you change your mind, you can undo that too. Or at a later point, if you want to access old versions, that's also possible.

So what you have to do for that is we can go to the data panel. And if you look at the screenshot here, I'm accessing this drawing. And if you click on that Info button, that'll bring up some more information. And you can start seeing a list of the different versions that you have created in the past, and you can open one of them or promote that one to be the latest. Do whatever with that.

One more thing that you might notice in the screenshot here is this link here. So just as we had links in the reference menu back to the model and drawings, you'll find the same kind of links in the data panel as well. So you can see all of the files that are related to one file in the data panel.

OK. So we're pretty close to ending. So to summarize what we've just seen, fusion drawings is a very accurate tool with the best in class documentation engine behind it. And it has a growing feature set. With every release, we're actively working on adding more value to the customers.

And it's also end to end. Fusion 360 is that product that's helping us take our ideas from design to concept until production. And drawings is something that makes it possible, and it just fits into the ecosystem. And the other thing is-- well, the ecosystem, to access your content. So you can access your files anytime anywhere, and they'll always be in sync.

OK. So before we get ready for some Q and A, I'd just like to remind you that I think the first two are not relevant anymore, but we also have the voice of the customer and Fusion 360 Office [INAUDIBLE] that are located right here on level 3. So if you're interested in speaking to the Fusion team one on one, giving your feedback, anything, just feel free to reach out to us.

OK. Are we ready?
AUDIENCE: What file format is [INAUDIBLE]?

KRITHIKA SUNDARARAJAN: Yes. They are DWG files. Well, when they're saved, they are F2D files.

SUNDARARAJAN:

AUDIENCE: So they still are [INAUDIBLE]?

KRITHIKA: Yeah. But internally, it's a DWG file.

SUNDARARAJAN:

AUDIENCE: Can you open [INAUDIBLE] AutoCAD 360 right now?

KRITHIKA SUNDARARAJAN: You can upload it. Yeah. So if you want to do that, you can just use the Export option, the one in the output panel. Or, you know, the F2D, it's just like a zip file. You can rename it and just open the DWG [INAUDIBLE].

AUDIENCE: I love that you're developing a drawing program [INAUDIBLE] 360, and it's growing. I get it. And it's nowhere close to being able to do what I need [INAUDIBLE] drawings [INAUDIBLE]. So in the meantime, I'm thinking of what is my countermeasure using another product to finish my drawings? And so my question was, is there a way for me to open up the Fusion 360, put my drawing [INAUDIBLE] in [INAUDIBLE] animation [INAUDIBLE] inside Fusion and then finish that drawing in AutoCAD without losing associativity?

AUDIENCE: [INAUDIBLE].

AUDIENCE: So export app. And what's the best work flow [INAUDIBLE] so that the [INAUDIBLE] still in the file but not indistinguishable from each other?

AUDIENCE: [INAUDIBLE]

OK. So it's going to the server I think [INAUDIBLE]. So as soon as we export [INAUDIBLE], [INAUDIBLE] should do [INAUDIBLE]? It would lose [INAUDIBLE].

AUDIENCE: [INAUDIBLE]

AUDIENCE: Is there a way to put the drawings [INAUDIBLE] AutoCAD into the Fusion [INAUDIBLE]? So some other drafter can then finish it.

AUDIENCE: [INAUDIBLE]
AUDIENCE: OK. So you could upload it [INAUDIBLE]. But I can still put it on AutoCAD A 360 [INAUDIBLE]?

AUDIENCE: Yes. [INAUDIBLE]

AUDIENCE: Is it possible to see an example of what [INAUDIBLE] looks like in AutoCAD?

KRITHIKA SUNDARARAJAN: I don't have AutoCAD installed.

SUNDARARAJAN:

AUDIENCE: [INAUDIBLE]

KRITHIKA: Yeah. That's right. If just a paper space. As he said--

SUNDARARAJAN:

AUDIENCE: You [INAUDIBLE] get it to [INAUDIBLE].

KRITHIKA: Right. So you'd be losing the associativity with the model, and it opens up only in paper space.

SUNDARARAJAN: The annotations can be associated to the views, but there's no model changes that can be accommodated after that.

AUDIENCE: [INAUDIBLE] in [INAUDIBLE] space [INAUDIBLE].

AUDIENCE: You can add dimensions and measure [INAUDIBLE] like that.

AUDIENCE: OK. So I can still [INAUDIBLE] in AutoCAD [INAUDIBLE] if I know the scale [INAUDIBLE].

AUDIENCE: You shouldn't even have to do that. [INAUDIBLE]. So you shouldn't even have to [INAUDIBLE]. [INAUDIBLE] associative work [INAUDIBLE]. [INAUDIBLE]. [INAUDIBLE] true [INAUDIBLE].

AUDIENCE: So [INAUDIBLE] it hasn't even [INAUDIBLE]?

AUDIENCE: It has [INAUDIBLE].

AUDIENCE: It comes [INAUDIBLE] as [INAUDIBLE] from [INAUDIBLE]. I'm sure if you [INAUDIBLE]. But it essentially means more [INAUDIBLE].

AUDIENCE: So it's not just [INAUDIBLE] down and [INAUDIBLE].

AUDIENCE: [INAUDIBLE]. And if you had [INAUDIBLE], you could create [INAUDIBLE] for [INAUDIBLE].
AUDIENCE: Yeah. [INAUDIBLE].

AUDIENCE: So when you go to [INAUDIBLE], you will have had [INAUDIBLE]. But it has [INAUDIBLE] points [INAUDIBLE]. [INAUDIBLE] dimensions. The dimensions will [INAUDIBLE].

AUDIENCE: [INAUDIBLE] turned off?

AUDIENCE: It's [INAUDIBLE].

KRITHIKA SUNDARARAJAN: Can turn it off. [INAUDIBLE].

SUNDARARAJAN:

AUDIENCE: Are there any plans to [INAUDIBLE] back rather than doing all the [INAUDIBLE]?

AUDIENCE: So for [INAUDIBLE], [INAUDIBLE] annotation?

AUDIENCE: Yeah. So being able to define all [INAUDIBLE] and [INAUDIBLE] in [INAUDIBLE] environment?

[INTERPOSING VOICES]

AUDIENCE: [INAUDIBLE] as where is it profitable. And as far as the timing of it, I can't give you a timeline [INAUDIBLE].

[NO AUDIO]

AUDIENCE: I don't know if [INAUDIBLE] that, but it's more to

[NO AUDIO]

KRITHIKA SUNDARARAJAN: Yes.

SUNDARARAJAN:

AUDIENCE: I had some issues [INAUDIBLE]

[NO AUDIO]

AUDIENCE: [INAUDIBLE]

AUDIENCE: I know that [INAUDIBLE]

AUDIENCE: [INAUDIBLE]
AUDIENCE: Sometimes the issue where they can only open up AutoCAD 2014 versus [INAUDIBLE]. And so that's usually the case, and we don't currently

[NO AUDIO]

KRITHIKA SUNDARARAJAN: Yes.

AUDIENCE: This might be a little bit of more general question, but many times I don't have my computer with me. I do some kind of sketching design working [INAUDIBLE], but it's quite difficult to get the [INAUDIBLE] file in order to turn it into [INAUDIBLE] geometry of it and then upload it. Is there any thought that has been given to making the space [INAUDIBLE] design files go also available in Fusion whenever [INAUDIBLE] explore it taking further in the drawing [INAUDIBLE]?  

AUDIENCE: A little bit of [INAUDIBLE]. I don't know if you saw him [INAUDIBLE] this week, but it has to do with quads and triangles and [INAUDIBLE] and how you get that in surfaces and [INAUDIBLE] the way that you do in [INAUDIBLE]. So even though they're all [INAUDIBLE] other products, it doesn't mean they're based off the same hurdle, the same meshes. So it really has to do with meshes.

There is a [INAUDIBLE] to maybe get it in. Right now [INAUDIBLE] data and [INAUDIBLE]. But that still-- I'm going to be very honest-- isn't quite where, as a user, I would say I [INAUDIBLE] to add this mesh, but the mesh is still so dense that I can't [INAUDIBLE]. I want to actually crop this whole area [INAUDIBLE]. So it's coming. We're doing a lot of development in [INAUDIBLE] areas, most certainly [INAUDIBLE]. It should [INAUDIBLE] over which [INAUDIBLE] just converting particles to the mesh, the individual [INAUDIBLE].

AUDIENCE: And I think that's the work around that I actually found is when [INAUDIBLE] data [INAUDIBLE] file, there is a [INAUDIBLE] and a folder structure that's what [INAUDIBLE]. But I just say that just because I do a lot of the sketching [INAUDIBLE] iPad-wise and later on for [INAUDIBLE].

AUDIENCE: By the way, [INAUDIBLE] Fusion 360 entering new data.

[INTERPOSING VOICES]
KRITHIKA SUNDARARAJAN:

AUDIENCE: Can you get the [INAUDIBLE]?

AUDIENCE: [INAUDIBLE]


AUDIENCE: [INAUDIBLE] as [INAUDIBLE] printer, and we are still [INAUDIBLE] little bit intelligent. So that is something that we are focusing on. [INAUDIBLE] priority [INAUDIBLE]. But we have [INAUDIBLE] that are [INAUDIBLE] how [INAUDIBLE]. So it is something that's [INAUDIBLE]

AUDIENCE: [INAUDIBLE]

KRITHIKA SUNDARARAJAN: Yeah. I can delete that.

SUNDARARAJAN:

AUDIENCE: [INAUDIBLE] once you removed it, you can't [INAUDIBLE].

AUDIENCE: Ever? For drawings [INAUDIBLE].

KRITHIKA SUNDARARAJAN: The drawing.

SUNDARARAJAN:

AUDIENCE: [INAUDIBLE]

AUDIENCE: [INAUDIBLE] in that drawing

[INTERPOSING VOICES]

AUDIENCE: So did you just want me to [INAUDIBLE]?

AUDIENCE: Yeah. You can delete [INAUDIBLE].

KRITHIKA SUNDARARAJAN: Yes.

SUNDARARAJAN:

AUDIENCE: When you have smart dimensions, why did you take away the explicit ways to do that?

KRITHIKA SUNDARARAJAN: We didn't exactly take away. It's there.
SUNDARARAJAN:

AUDIENCE: No. Why is it still on there?

KRITHIKA: Oh. Why didn't we take away?

SUNDARARAJAN:

AUDIENCE: Yeah.

KRITHIKA: That's a good question. OK. For things like-- yeah, why didn't we take away? There's no explicit need. OK. So when we give smart dimensions, why are the old dimension tools still there?

AUDIENCE: So smart dimensions sort of [INAUDIBLE] in a lot of ways, and there's a lot to [INAUDIBLE]. But there are certain times when they basically [INAUDIBLE] exactly what you want. So we wanted to allow [INAUDIBLE] especially because [INAUDIBLE].

AUDIENCE: Wouldn't that just mean smart dimensions are [INAUDIBLE]?

AUDIENCE: Maybe it's easier to place a linear dimension [INAUDIBLE] in a certain sense. Or [INAUDIBLE] where it's acquiring the edge instead of the [INAUDIBLE]. So there [INAUDIBLE] case smart dimension for what the [INAUDIBLE] wants to pick. There's the possibility that [INAUDIBLE].

KRITHIKA: Also, things like for the radial and diameter dimensions, that are defaults for the smart dimension tool too. So you have to flip it around. If it's a full circle, it's a diameter or arc if it's a radius. So I think giving access to the standard tools just makes it more convenient when you know what exactly you want in that scenario.

OK. Are there any more questions? Any feedback?

AUDIENCE: It's good looking [INAUDIBLE], and I'm glad you're developing it [INAUDIBLE].

KRITHIKA: Cool thanks.

SUNDARARAJAN:

AUDIENCE: It has a long list of [INAUDIBLE].

KRITHIKA: That is true.
OK. Thank you.

[APPLAUSE]