PHIL EICHMILLER: All right, good morning, everybody. We're going to get started. I'd like to welcome you to AU. Welcome you to the first Fusion class of the week. My name is Phil Eichmiller.

BRYCE HEVENTHAL: Good morning, everyone. I'm Bryce, and over there we have Aaron Magnin as well.

AARON MAGNIN: Thanks, Bryce. Sorry.

BRYCE HEVENTHAL: We'll figure out that. All right, well, welcome for the first Fusion class. Hopefully a lot of you joined us at the Fusion 360 meet up last night. We got to see a lot of customers, and we saw what they're going to be adding to Fusion over the next couple of months, as well. Today Phil is going to do a little demonstration of Fusion, but really this is for an overview for you guys to see what Fusion is.

We're going to point you in a bunch of different classes. If you want to learn more about Cam, for instance, we'll tell you where to go, what day it's on, as we get through this class today. But first, do you have anything else, Phil?

PHIL EICHMILLER: I just wanted to just briefly introduce myself. I'm a QA on the Fusion 360 project, and I've been there since day one. And so I know a lot about Fusion as a result of that. You might see me on the forums posting as Phil E., and I'm also an instructor at Portland Community College. I teach Inventor, so I have a passion for teaching, and I'm glad to see so many bright shiny faces here this morning.

BRYCE HEVENTHAL: Awesome. And then, we asked a couple of minutes ago, but we saw a lot of people trickle in. How many of you have used Fusion so far? Installed it, downloaded it, tested it a little bit. About half. That's awesome. Good to hear.

We'll show how to install as well for those of you haven't yet. It takes five to 10 minutes, super quick. But today what we're going to explain first is what Fusion is. So Fusion is what-- we call it product innovation platform. It's our theory of replacing the traditional method of a very linear process where it's broken between industrial design to mechanical engineers, all the way to manufacturer.

We see it as a cyclical process, all the way from designing to making. And then we're going to be able to use and collect that data and redesign. So that's been our vision moving forward

with Fusion. You'll see a lot of this going on at the manufacturing booth. I'd highly recommend going over there after the keynote today.

There's going to be a lot of different talks on the product innovation platform. But what we're here to talk about today is what Fusion is. And we're going to give a summary, overview of what it is, and how to do a couple of tips and tricks as we start with Fusion.

So Fusion's a lot of things. It's a huge, huge product. To come master at the whole entire thing is going to take a very long time. We have people who specialize in different parts of the product, whether that be the sculpting, Freeform stuff, the Cam tool path stuff.

So to start off, it's a Freeform design. So if you want those high curvature continuous surfaces. The cool thing is with Fusion, usually these are different tools that don't really talk well together. There's a translation that occurs. So if an update occurs, you usually have to rebuild those mounting bosses, those parametric ribs on the Freeform surface.

So it is also a parametric modeler, so your 2D sketching and all your eD features, similar to an Inventor or a SolidWorks is also in Fusion as well.

And then we get into 2D drawings because we still see a need for manufacturing drawings when you go to the manufacturing process. It also has a collaboration aspect to it as well. We can do things like live reviews, commenting, redlining, all on the file. So it travels with the file. We're only working on one file, which is awesome.

When I used to work on desktop CAD tools, I can't remember how many copies we made at my old company. We would just copy files all across, and produce a PDM nightmare.

Then we could get into some finite element analysis. And we just had one of our most major releases last Wednesday. And we'll talk about the update process a little bit later. But we added the NAStran Solver to all our simulation study types, as well as we added four of these advanced study types as well for nonlinear, explicit solver, and then topology optimization.

And there's some classes on specializing just on simulation, if you're into that type of work.

And then we can get into some cool things with rendering built in. We'll see some renderings as well throughout this class. And then what we're going to be working on very soon, we're going to be introducing things like sheet metal to the Fusion 360 to round out our product development process.

And then also integrated electronics, which is going to be able to bring in the eCAD board and visualize that in 3D, so you can really design close to that package.

And then we're going to be introducing several levels of generative. You'll see a lot of talks on generative at the manufacturing booth today. But we have one currently called simulation optimization, but we'll see some latticing as well throughout the year. That is the topology optimization that we have in the tool today.

And then we get into Cam as well. So we-- in that last update last Wednesday, we introduced four- and five-axis as well, to our three-axis and two-and-1/2-axis, as well as turning water jet, laser cut, and plasma.

And finally, we're going to be introducing over the next year a zero client, so you'll be able to access your data and edit from any device, whether that be a Chromebook, an iPad, or you're just one your-- you need change a dimension or check a dimension out at the shop. We'll be able to access Fusion through any device and make changes.

But I'm going to go ahead and give it to Aaron.

AARON MAGNIN: Thank you sir. All right, thanks, Bryce. Yes, I didn't really get a chance to introduce myself earlier. My name's Aaron Magnin. I actually work with Bryce. We do technical marketing. So if you go to our YouTube channel, you might have seen our faces before we like to pretend like we're famous, but-- anyway, more about Fusion and what we're going to be learning today in this class.

At the end of this class, we want you to be able to leave here with the knowledge of how to get Fusion 360 installed and logged into your account. You can access your data from anywhere right now. I mean, you can jump on any computer, log in, and it will bring all your files over. It will bring all of your settings over. It's a pretty amazing feature. It's a pretty awesome tool to take advantage of.

And at the end of the day, we also want you to be able to create a simple sketch, and part that's going to be Phil's work. He's going to be showing you how to make some models, to assemble some models, to do some of the Freeform stuff as well. I believe.

And we're also going to show you to make a simple assembly relationship. Here's a little animation, a little GIF of our saw that you might have seen. It's all over the place. We really

like this reciprocating saw. And then also we're going to show you how to create a rendering or to even print a design. And that image you see right there, I'm going to blow that up a little bit bigger. That is an actual rendering from Fusion 360. You can see the level of detail on that.

The background is actually an HDR image, but everything that you see on that part is done in Fusion, and you can see the tool paths actually created from the CNC side of Fusion, which is pretty darn amazing.

And I do also want to talk about Autodesk commitment to education. How many of you are in education or are students right now? How many of you know that all of Autodesk products are available free to you, OK? Good. You should know that. Because this is our commitment to the community. It's our commitment to startups and to new businesses. It's a great way to get in and start developing your products before you actually need to go and purchase anything. So make sure to check this out, not just Fusion 360. All of the Autodesk software is available. Make sure to get on there and check it out. And I'm going to pass it back over to Bryce. He's going to show you some of the stuff that you can use to learn this too.

BRYCE HEVENTHAL: Thank you. So we wanted to introduce-- give you guys a little high level overview of Fusion today. But we didn't want to leave you guys stranded right after this. There's a ton of learning opportunities at previous CAD tools that I've learned in the past. You basically have to pay a $2,000 fee to take a four-day class, and you have to take off work for four days.

We have so much learning content all over the website, all over YouTube. It's a different way of learning we see for CAD tools nowadays. Rather than you having to go ask an expert, you just Google it, and you'll get a small video.

So on our website, Autodesk.com/fusion360, excuse me, we have a learning page, which has three levels of learning, whether that be high-level learning all the way down to the feature level. If you want to learn something about the fill it, if you needed to. And we have certain things like foundational concepts, all the way to, if you're a SolidWorks user, how to transition over to Fusion 360 as well.

There's a whole slew of videos.

In addition to that, we also have different learning opportunities. Some people don't like to watch videos. They like a one-to-one touch, a person there. So we offer webinars, virtual labs. Deep dives are similar to webinars, but it goes deep dive into, let's say, rendering, or Cam,

where the webinars are more of an overview.

And then we also have several meet-ups. It's very group oriented, all around the world. We have them, I think, in several different countries. I can't even tell you now. It's pretty insane how big they're growing. But you can meet other Fusion users, see what they're doing. And they usually have a certain topic that they talk about, whether that be additive manufacturing and how they're using Fusion 360 to do that. I'd highly recommend going to one of those. And

Then as well, we have several-- I would highly recommend checking out our YouTube channel. We have several playlists, whether that be quick tips that you just want to learn. Every day we publish out one every week, all the way up to a really long, oriented hour-long videos to really get you a deep dive into rendering.

All right, with that we'll go ahead and throw it over to Phil and see how Fusion works.

PHIL EICHMILLER: All right. Oh, this way. There we go. So we just were talking about this, so I wanted to start off with what's what's your gateway to Fusion. And so, there's, I think, there's probably a couple of ways of getting there. I always type in Fusion360.autodesk.com. But you said the alternate one, which is probably correct better. What was it again? Autodesk.com/fusion360.

This takes you here, OK, well, either way you can find us. You know, right up front, you got the download the free trial. And you know, I'm not going to click on this, because it goes into a workflow. But all it asks you for is your email address. And next thing you know, you're downloading and setting up.

You know Fusion runs on Mac and Windows, so you don't really have many concerns there about your operating system, other than whether, as long as it's not a Mac operating system from eight years ago or something like that. So if you're up to date, you should be just fine installing and running Fusion.

AARON MAGNIN: Your hardware requirements are a lot less than you'd expect from like a 3D parametric model, so it's definitely something to check out.

PHIL EICHMILLER: Yeah, we strive to make Fusion run on what would normally be considered very anemic graphics and stuff like that. So give it a try, and see what you find. Definitely don't worry too much about that stuff. If you're doing business with your machine, you should be able to run Fusion.

And just scrolling down the page, you know, you can see there's access to some of these more specific learning and areas of interest, you know, so you can deep dive right there, from the main page. And then you get down to the bottom and you have links out to the gallery where people post up their designs and things like that.

One of the cool things about the gallery is that you can also post videos and models and things like that. So you can put a lot of information or get somebody else's information. You can download the model and look at how they built it and stuff like that. So it's another great learning tool, and also fun eye candy. For me, I just love renderings and things like that.

So let's see, where else? Also, right here from the page, there's the link out to our forum. And there's also the link to the gallery and a few other things. Idea station, where you can send us your ideas. If you're running Fusion and you're like, I really wish it did this. You know, post it up in our idea station. We've incorporated what? 300 something ideas?

BRYCE HEVENTHAL: 382. Our PMs have a meeting every week to discuss those idea stations. And then other users also give you kudos, so the higher the rating, the more likely we are to implement it quicker.

PHIL EICHMILLER: Yep, so don't be afraid to go to the website, and check it out and download your free trial. Now there is one key thing about-- how many people here have an Autodesk ID? And you know for sure you do? OK. You're going to want to use that, right? Because your data in Fusion, and in our world, your data is all attached to that ID.

So if you already have one, don't punch-- unless you want it to be your own personal Fusion account, don't create a new ID unless you really want one. Let's put it that way. If you've already got a business associated ID with Autodesk. So, any questions? All right, cool. You guys feel free to ask questions as we go.

I'm going to click over to Fusion right now, and just make it full size here. And let's talk about what we're looking at here. So this is when you first start in Fusion, I've got a couple of designs open right now. Up at the top, you can see those across the ribbon up here. Your first brought straight into the modeling environment, and there's some things that you need to know about what's going on here.

So this little ice cube tray up here is called the data panel opener or whatever. And it's where you access the data panel. And your data panel is where-- this is like your file cabinet in the sky, right? It's got all of your designs. Your organize by projects.

So if you're like a contractor, and you've got you know, like Nike and Adidas as customers of yours, you can have them in completely separate silos, and those are called projects. So if I bounce out to my project list, you can see that, whoops, there we go I've got a lot of projects I help a lot of people and customers and things like that. I get invited to a lot of stuff, so I can check out people's designs and things like that.

So that's one cool thing about working this way is that when I need to help somebody, I just send them my email address, and they invite me to their project. And I open up their data and figure out what's wrong with their model. I help a lot of customers that way on a just a nearly daily basis.

Creating new projects is as easy as clicking the New Project button. And then you can define, once you're in a project, let me go over to our class project here.

BRYCE HEVENTHAL: Yeah, and if you need more levels of permissions or high level control over your users, there's something called Fusion team that we have as well, to give you more access to tools to be able to control people's access to your files and data.

PHIL EICHMILLER: Roles and permissions and things like that. But inviting people is just as easy as just clicking on the People tab, punch in an email address, and it sends off an invite. And next thing you know, they can log into Fusion, and look at the project. So it would show up in their project list. It's pretty seamless.

Other than that, this acts kind of like what you would expect a File Explorer to do. There's new folders. You can put-- you can have folders and folders and things like that. However you want to stay organized. You can also change to a list view, if that's your thing. And right here is the Upload button, which is a very key part of your experience with the data panel.

Basically the whole idea is that, well, I've got to put some design up in here, right? Sometimes it's going to be anything from like Inventor or SolidWorks or any kind of CAD model that you want to put up there to work on. When you use the Upload button from the data panel, what we're going to do is we're going to translate it for you into a Fusion design. So you're going to get the results that you would expect from a traditional CAD modeler. You're going to get, essentially, it's going to turn Inventor and SolidWorks files into step files.

It will preserve the browser structure and the naming and some of the colors if it can. But it is a pretty much traditional CAD translation that you might be used to, if you've done that sort of

thing. And there's tools to help, once you get your assemblies up there, and we'll kind of go into that a little bit later. But just rounding out the data panel here, you can also search for things and search through all of your data for things, so it's a full-featured sort of file system over here.

When you're looking at objects in the data panel, you can also click on the little eye right here to get more information from them. So if there was a drawing for this, it would be listed as a related file here. There's also the concept of versioning. if you'll notice that my little demo file here is at v4, because I've saved it four times.

So that's another unique thing to Fusion is that if you find that you've gone down a wrong path or completely broken the model somehow, I mean nobody's ever done that, right? You've never broken a parametric model. You can roll back to a previous version, and just pick up where you left off.

Every time you save a version, you know, you click Save, like you're saving it. You're prompted to say, well, do you want to put a little note on here and say, like, oh, this is the one that I got ready for AU. So I can tell that if I'm looking at the version history, I can roll back to a milestone really easily.

Let's see, anything else here? And it also punches out to the web, so the other side of the data in Fusion 360 is out on the web and through the Fusion, the portal out there. Is it, do we still call it A360? I'm horrible at this.

BRYCE HEVENTHAL: No, it's called Fusion on the web. It's another way to access your data, through a web browser. You can view redlined markup, see comments. It's another way to interact with your data. And it's free, so you don't need to have a-- you don't need to pay for a subscription. So it's really good for CAD managers or non-CAD personnel to be able to still work in the product development process.

PHIL EICHMILLER: Yeah, you just send the web page out to somebody, and then they're looking at stuff, and they can tumble the model, and it's right from a browser. So--

AARON MAGNIN: I want to emphasize how cool that versioning is, because it's saved me many a times. If you make a mistake, you can go back and you can see whole history has been expanded out there now.

PHIL EICHMILLER: Yeah. It allows you to do some clever things too. If you make a decision about an older version, you can actually open that older version. It opens it. Either, well, you got two choices. One is to promote it, so it just puts it right back up to the top as a new version. Or you can open it as read only. And then save it as something else, and just branch your design right there if you want to. Not to be confused with branching and merging, but that's a very simple version of that workflow.

AARON MAGNIN: You've got a question over here.

AUDIENCE: Can you cherry pick out previous versions though? So you can go back, pull it out, and bring it to the latest version?

PHIL EICHMILLER: Yeah, well, so that would be a little bit more manual of a process.

BRYCE HEVENTHAL: [INAUDIBLE]

PHIL EICHMILLER: Yeah, so completely outside of the concept of branching and merging, which is coming. The way I would do that is I would open it up as read-only. I would find whatever was interesting in there, and I would have to manually save it out, and then insert it into the other design. So you can do a save as. It winds up in your project list, and then you just insert it, and it becomes a linked file. So--

BRYCE HEVENTHAL: But we do have something coming very soon called branching and merging that's a more automated process of that.

PHIL EICHMILLER: Cool, thanks for the question. All right, so I think that wraps it up for the data panel. So you get the idea. I tend to always shut the thing. Some people run with it always open. I like to have the real estate. But this is the best way to do it. There's also-- you can close that by the x here. So I guess if you're mousing sloppily, you'll hit something right there that will shut the data panel.

So, all right. So now looking at the modeling environment. OK, let's take a look at this. First of all, there's the kind of things you would expect to find in a regular app frame, right? You have access to a file menu, which gives you various commands, and you know, some of them like we just talked about, like save as.

Or you can, even if you've managed to lose your connection and you need to recover documents you can go in there. Basic file menu kind of stuff. There's the three-and-a-1/2-inch floppy save button That hopefully will be with us till the end of time. I just love that icon. I don't want it to go away. What are you going to replace it with?

This right here is the workspace switcher, and it's basically just a list of the different workspaces that are available in Fusion. So when you go there, you wind up with a different set of tools, right? So if I go to the patch environment, I'm treated to a bunch of surfacing tools, right? So this is not surfacing like you're going to find an alias. This is more like I've imported an IGES file, and it's got holes all over it. And I need to patch them up and stitch it. That sort of thing.

Or if you're if you're using surface bodies to do modeling, to make modeling bodies, and things like that, this is a great place to-- obviously where you would do that.

The rendering environment puts you into-- there's nothing really to see here. But you know you can see there's rendering tools and so forth. There's an animation environment that allows you to sort of tear apart the assembly, kind of like Inventor Publisher did. You can make animations, and fill out your timeline here with events and that sort of thing. And it's good for, you know, you can also record it, make a video, that sort of thing.

A simulation environment. All the sort of stuff you'd expect to find. Anyone here use Inventor Simulation? So you're going to find a basically the same kind of workflow. You know, you start off by creating a study and defining materials, constraints, loads. I mean it's almost exactly the same workflow, and it's-- which makes sense, because that's how you do that stuff, at least with our software.

There's a Cam environment, and you know, this is purely just for setting up the Cam stuff. You know, the whole time you're working on one model. That's the cool thing about this, if that model changes, all of this stuff has to react to that. And it's all automatic and seamless.

You, know, the only thing that you're going to have to do is if, well, like I said, hey, you can sometimes break a model yourself. But that's OK. There's lots of tools to patch and edit those things. And there's our drawing environment as well down here on the list. So you can do new drawing from design or new drawing from an animation. So you can put your exploded views into drawing form and so forth. And that opens up the 2D drawing editor.

So that's your workspace switcher. We're going to go back over to the model environment, where I'm going to get started in just a minute. Let's see. Hopefully, my toolbar isn't customized. I can't help myself, and so I always wind up with a non-stock toolbar, and I'm showing customers this kind of stuff. So I forgot exactly if this is a-- a couple of tools are added. But you can see as you're looking through the menus for each individual section-- I'm in the sketch menu right now-- you can actually add these tools to the toolbar as you're going. Or drop them, you know, drag them off of the toolbar if you're done using them.

So rather than sort of a static toolbar definition, you know you just basically set it up as you're going. When. I'm going through workflows, I'll put some tools up there, and I'll take them off. And every once while, it gets too crowded, and I just clear it all away and start over again.

So you have your basic sketch environment right here. You have create tools, like for creating extrudes and revolves and lofts and things. And your modified tools for adding fillets and chamfers and doing some things like split body, hacking up solids that are on the screen.

You have assemble tool. We have a pretty good set of joint tools here. So if you're familiar with joints in Inventor, you're going to feel right at home here. We have a cool thing called the As Built joint, and I'm going to go into that a little bit later, and show you how that works. It's basically a great tool for if you have stuff that's already, you know-- if you're doing a top down design, And the bodies are already relative to each other, you can just apply a joint between them without having to tear things up.

Lots of good tools here. There's also-- we also have the concept of contact sets. And you know designing with contact, right? Has anyone ever done that, where you're trying to define the range of motion of something, so you define contact. It's not something you leave running all the time, because it really slows things down. But you can then move the mechanism till it bumps into something. Figure out what the angle is. That kind of thing.

So it's a pretty robust set of assembly tools there. Nice set of construction tools here. So if you need to set planes up and build something, you know, the fun ones for me are when I find that I have to-- I drill down, and I figure out, oh, great. I'm going to have to put two points in here, and then put an access through those points, and then put a plan, add an angle to that access. That's always the fun ones for me.

And then you get to drive the whole thing, and that's my favorite part of modeling. But there's some good inspect tools. We have things like Zebra Analysis and Draft Analysis so, you know

this gets down into manufacturability, right? You know, you want to quickly find out that your plastic part is going to have the right draft angles. There isn't going to be an undercut in there or something like that.

There's an insert set of tools for putting in decals in canvases, so if you like to draw on top of an image of something, or sketch rather, or model, whatever you're doing. You have access to that as well. Or putting decals on, for like stickers. It works really well. They look great in renderings and stuff.

And a couple of other tools for bringing in data from other places, like SVG and DXF is a big favorite of folks who are doing commercial work for other folks. They've got to get the logo in from a company or some kind of something that is art that's not properly done by sketching with parameters.

There's also insert mcmaster-carr components. So this just punches right out to the Mcmaster-Carr site. Has anyone ever done that? You can get CAD models from them for a lot of their stuff. Be careful, be wary of their IGES components. I would say, never insert IGES from McMaster-Carr. Just take my advice. And--

AARON MAGNIN: What's the best file type?

BRYCE HEVENTHAL: Step. Step is absolutely the best. Does anyone know the difference between Step and Sat Import. When you have like an assembly that comes out of SolidWorks or Inventor, and it gets saved as a SAT file, everything gets converted into bodies. So it crushes the structure of it. And you wind up with-- you don't know what the assembly was anymore. So Step is definitely the best, because it preserves that assembly structure.

AARON MAGNIN: And the beauty of that McMaster-Carr insert is that you don't have to spend time designing bolts and things that you don't make, right? So it's a great way to make your designs a lot faster.

PHIL EICHMILLER: Yep, if you'r doing cabinet hardware, hinges, and things like that. They don't always have a CAD model, so you just have to kind of learn what they support. Let's see, there's the make menu, so this is obviously all about 3D printing. There's add ins, so there's-- we have an API engine in here, so if you're into scripting. There's people that are-- and there's even a whole forum page about it, and you know, there's some community around that. So if you're into doing that kind of scripting to make your tedious workflows more automatic, that's certainly

supported. And we like to hear from people who are using that.

And then there's a set of selection tools over here on the end. I always run with my selection filter wide open, but every once in awhile, you get into a complicated model. And you just find, you know, it's going to be worth it for me right now to just shut all this stuff off, and just let me pick edges. Or faces. Or whatever it is you're after.

So you can isolate stuff that way. I tend to-- you want to be in the habit of going back and turning it all back on again, or you'll go, why isn't anything picking?

There's-- you can do selection priority. So if you're working primarily with bodies, this is actually fairly useful here too, because if you're working primarily with bodies and moving them around, for instance, you'll want to have body priorities set, so that it's easier just to go and, without having to play around with this big long checklist of filtering, you could just make a priority out of picking bodies.

The Selection tool select by name, boundary size, and invert selection. Now this is a great one if you have a huge array of things on the screen, and you're trying to isolate just one. Anyway, we don't need to go into those workflows. But there you have it. That's basically your toolbar. And all the other toolbars in all the other environments with their tools act the same way. So when you want to throw something up to the top, you just put it up to the top.

BRYCE HEVENTHAL: And one of my favorites that I always use is the S key. It's also customizable.

PHIL EICHMILLER: Yeah, so this is an idea that SolidWorks was kind enough to show us. And we were smart enough to pick it up. Basically you can search for anything. So if I'm looking for a zebra, and I want to have it in my toolkit, you know, I just put it up there. And this acts like the toolbar, except it's a tool box. And it shows up, and you can have any list of things here. So as are working along, it's right where the mouse is, wherever you're at.

So if you're a hot key kind of person, there you go. And the--

AARON MAGNIN: Help you stay focused on what you're working on. And it's mobile as well. So if you're in simulation, it will show you simulation-related commands.

PHIL EICHMILLER: Yeah, you have a different tool box for every environment, which makes sense. You wouldn't want to be having a bunch of animation tools popping up right now when you can't even use

them, right?

OK, so let's see, what else is in here. I'm sure you're all aware of the view cube. How many people in here know and love the view cube? OK, so it's actually particularly handy if you're not well-versed in the kind of gestures that make things happen. You're running without a mouse, perhaps, on a Mac, and you're not very good at gestures. You can always mouse up over to this thing, and use it to twiddle the model around.

I find that a lot of people that are new to 3D CAD in my Inventor class at the college, they respond very well to the view cube. The thing I like it for is isolating your ISO views. Right? That's the main thing I use it for is, like-- there's no better way to get straight to it than-- excuse me-- orthographic view right away. And down at the bottom, you guys can all see that, because the screen's really up high.

You have a bunch of navigation tools and then settings for your screen. So if you don't like the color of the environment, you can go and change environments. Or if you want to know, see what's another good one? Visual style. If you wanted to turn off the edges, so it looks a little more realistic, or look at it as a wireframe. That kind of thing is controlled there. We support multiview ports, that kind of thing. Whoa, made my grid really dense.

So anyway I'm going to go so right now I'm going to do some modeling. OK, so those are the basic things. Oh, yeah, I forgot to tell you. There this thing on the timeline that you guys are going to want to pay attention, to this thing at the timelines. We have-- in Inventor, you're used to just things stacking up into the browser, right? And that happens in a particular order.

So time and objects are mixed together in other Cad packages, and in infusion, time and the object you're creating are actually separate and so over here in the browser just think of this as the file cabinet for the design that's where all the bodies and sketches and components are going to show up. And all the things you do are going to show up down here in the bottom in the timeline.

So that's how you can quickly, you know, go in and figure out, oh, I need to put something before something else. Or edit something that's further back in time. Yes? Your question?

AUDIENCE: Could you go into a little bit of detail, just [INAUDIBLE] detail about the program [INAUDIBLE]?

PHIL EICHMILLER: Yeah, absolutely. So let's see, anything else to mention here? Just one last thing, while we're looking at the UI, is there is a commenting thing, so everybody that's on the project can get in

and look at things and comment. And anyone that's accessing your data through the web page can also drop comments on your model that show up in here. So the commenting is very robust for being able to do hopefully all your communication about a model without having to dig into email. Right? We're keeping all the focus on your data.

Now let's draw something. That's what you guys came here for. I'm just going to start a sketch line command. And the first thing it wants me to do is say, well, OK I need to start a sketch. I haven't actually even formally started a sketch. There is a formal Create a Sketch command, but I usually just dive right in and start drawing on something.

So I've started the line command, and I'm going to click on this plane to start drawing on. And the auto rotate gives me that orthographic view, and I can start drawing, and based on the origin, which is always a good thing to do.

So this should-- if you're familiar with parametric modeling-- this should make you feel comfortable. And if you're new at this, this should also make you feel comfortable. Everything you can see right in front of you is actually telling you exactly what you're doing. So right now, the heads up display is asking me, how long should this line be?

And as I'm dragging it, you can also see that there is a vertical icon, showing me that there is a sketch constraint being applied to this that's going to keep this line vertically. So if I type in two and hit tab, it locks in the dimension and allows me to start picking the angle. And I'll just hit Tab again. Oh, and hit Enter. There we go. And it locks it in, so at vertical and at two inches, that's now a parameter in the parametric table. I just hit L, and it starts the line command again.

So we do support hotkeys, so L. I got a line command. There's a short list of, I think we only have 26 hot keys right now. We're eventually going to try to make it customizable entirely. But not right now.

And for the rest of this figure, I'm just drawing a bracket. So I'm just going to go ahead and sketch in the rest of the lines, and worry about the dimensions later. And as I'm doing this, I'm getting this great feedback about what's going on the whole time. I can get sort of some tracking off of the origin there to make sure that I'm aligned with it. And then complete the figure.

What you see there is that yellow fill in there, that's actually the profile. So when you're

extruding things, that's actually what you're extruding, you're not extruding the outline around it. This outline makes a boundary that is filled with what we call the profile. And that profile is what you're extruding or working on when you're doing stuff.

D for dimension, and go ahead and finish out the dimensioning on this. And, of course, I'm working in inches because I'm just that kind of person. As you could tell. So-- typical to a lot of other parametric modelers, if I select another dimension that already is sitting on the screen, it's just going to use that same parametric value.

So you notice that this is dimension three, and I'm working on dimension four over here. So I'm just going to click on dimension three and dimension four is now-- equals dimension 3. And that shows up in the parametric table as well.

AUDIENCE: [INAUDIBLE]

PHIL EICHMILLER: So that's just-- it's just penciling in the values. So when you look at the parametric table, it says, D4, the value for D4 is D3, and then you could go look up it wherever that is.

AUDIENCE: [INAUDIBLE]

PHIL EICHMILLER: No, no, it does. Watch, so-- so, yeah that's always it's always following here. Let's see, Back up a little bit there. All right, so now that I've sketched my first profile, and I'm ready to make this cool bracket. So the question was: you know, bodies, you know components, things like that. What's the difference? Right now I'm working in what would be considered a single component, and so that's something that's unique to Fusion, is that it's the top down design.

And Fusion, basically starting at nothing, and just working your way into a fully featured assembly. I mean, you always start off as a single part file, but as soon as I add another component, this will become an assembly. So you have the best of both worlds. You have part features and assembly features, all in one place. And so it sort of is up to you to keep track of what is it I'm working on right now? Because you know, you're doing deliberate things to make this design. So you have to watch the browser and make sure you know what you're doing and what just happened.

So if I go to Extrude, and here's another cool thing about Fusion is that almost everywhere you go, you don't have to right click and beg for permission to continue. That drives me nuts. I've got-- why can't I go on? The result is on the screen? Oh, that's right, I've got to right click, and

say, OK. How many thousands and thousands of extra clicks have you done saying OK.

Watch this. It says, stop, Sketch. Screw that. Extrude. Pow, I'm extruding with no problem, right? So just go grab the next tool and just move on. I just find that to be-- it's particularly satisfying. So what's going on here, right? You have the-- I'm extruding that shape. And I've got a manipulator for distance, obviously. And also one for draft angle, so I can play around with the draft angle as I'm going.

So this manipulator is pretty ubiquitous to Fusion. You'll find it everywhere you go, and it shows up with either more or less features, but they all basically do the same thing everywhere you go. So you want to be familiar with it. So when you see the little wheel, that's for creating angles and draft and turning things. And then, of course, the arrows are for adding distance.

Over here in the extrude dialogue, and we'll expand this out a little bit, I don't need to, but I like it look a little bigger you have your basic options, you know, where do you want this to go? You can actually pick an offset from where you're starting, which is cool. If you need to build in paint thicknesses and things like that, or do your fits and different things for your design. Obviously you can flip the directions, you can pick a distance, or you can send it to an object.

And pencil in whatever values you want. Your operation. You have your Boolean operations here, and you have new body and new component. So are you creating? What is it you're doing on purpose here? Are you creating a multi-body assembly where you want it to all start off as a bunch of bodies that are relating together? Are you making bodies that are going to cut into other bodies?

You're going to wind up with these bodies over here in the browser, or you could just go ahead and go straight into a new component. I don't recommend this. And here's why. If I was to create a new component right now out of this, the sketch would be at the root. And the body and the extrude would be down in the component. That component is perfectly fine in this assembly, but if I ever want to send it out somewhere else as a parametrically complete entity, it needs to have been made from the start to be that way.

So I don't recommend new component unless, you know, you don't mind that workflow or you're never going to export anything, obviously it works just fine. So right now, we're going to just go ahead and default to new body. And I'm going to go to a symmetric extrude. We have two styles of symmetry. So depends on how your brain works. This seems to be the popular

one. People like to think of the overall width of something. I'll just go ahead and make this two inches wide. No, that's too big. It's more fun.

And what do we have? We have a body in our browser. We have a sketch that's associated with it. Down in the timeline, we have a sketch and an extrusion. So if I play the timeline, there's the sketch, there's the extrude.

There's your gateway to learning. Download somebody else's model that you trust, or that you think looks like something you would do, and see how they built it. That doesn't mean that they did it right, but you can certainly learn what happened, Right?

So and if you want to edit things, I can edit this extrude by just double clicking on it down there, and right back into the extrude dialogue, and I've decided that this is-- oops, 1 and 7/8. And hit Enter, and it updates.

So to answer your question, and hopefully round it out. If you got more, just let me know. A body is a container for 3D information. The body contains some sort of shape. That's it. A component contains those bodies. It contains sketches. It contains everything else that you add to that component. So the component is just the larger, the next level up envelope for this stuff. Hopefully that makes sense.

BRYCE HEVENTHAL: And the main reason to use components is that's going to appear on the bill of materials, and when you go to drawings, as well if you need to do assembly motion, that also needs to be a component as well. And you can also isolate components, but you can't isolate bodies.

PHIL EICHMILLER: Yeah.

AUDIENCE: [INAUDIBLE]

PHIL EICHMILLER: OK, Yeah, we'll get there. I'll finish this part out completely in just a second. And just for those of you who might be taking the tips and tricks from the class with Jeff Strader and Peter Dooring from the forum. Rule number one, and I'm going to explain what rule number one is because-- well, anyway, I won't tell you why.

Rule number one is to create a new component and make it active. It becomes active on creation. And right now, I would be-- if I was making more components in this design, I would be now working in an activated component. Everything else is grayed out. And so if I add

sketches and things like that, you'll see that this component now has its own fully intact parametric history.

BRYCE HEVENTHAL: And you also notice the top level there automatically turn into a little assembly icon as well.

PHIL EICHMILLER: Yep. So just to explain, hopefully that explains what rule number one is. And there you go. Let's finish this thing out. I'm going to add a couple of sketch points to this. And use the whole command. Oops.

So I'm just working along to make this a parametrically complete sketch so that it updates when the model updates. And let's look at the sketch palette real quick. We haven't talked about that. The sketch palette hangs out every time you're sketching. It's a nice reminder that you're still in the sketch environment, in case you lose track.

And there's a bunch of settings here, but there's these set of constraints that are key to the experience of sketching here. So I want these two points to always be horizontal from each other. And so I'm going to go ahead and add a horizontal constraint. Now this is a very simplified version of this. Obviously two dimensions would be just as easy to add in this case.

But now one dimension will position that, right? So this dot is horizontal from this dot. So that's-- you know, you have these sketch constraints allow you to make these geometric conditions. I always tell my students when I'm sketching, I am always looking to satisfy conditions before I add numbers. It's too reflexive just to lock down a sketch and add in numbers that don't belong there, when a condition would easily get it done.

Why? You don't need a 90-degree angle. You could just say it's perpendicular. Right? I mean, that's the simplest example of that. I want to go ahead, and rather than stop the sketch, because I'm just that way, I'm going to go over to the whole command, and start the whole command. Pick the points. Obviously the whole command is rather enthusiastic about what it thinks you're doing.

Here's our manipulators. So here's another set of manipulators. They're a little bit different. But you can see they're positioned on the diameters, and I'm not I'm not even paying attention to what size I'm making this. I'm just trying to make a cool looking hole here. And I want the depth of this to be not all the way through it. And there you go. Two big lovely counterboard holes. I'd make them a little smaller.

BRYCE HEVENTHAL: We had a question.

PHIL EICHMILLER: Yes?

AUDIENCE: [INAUDIBLE]

BRYCE HEVENTHAL: Yep.

BRYCE HEVENTHAL: You'll see them around. There's shortcuts to.

PHIL EICHMILLER: Maybe at the answer bar they might have them.

AARON MAGNIN: Yeah, the answer bar.

PHIL EICHMILLER: Yeah, the hotkeys. And it's also-- so again, that's just a little sketch. And modeling workflow. And let's look at, well, here, I'm going to add another feature just because I want to.

AARON MAGNIN: So not all features require the sketch beforehand. Like Phil's doing now, he can just apply features instead of having to sketch them out first.

PHIL EICHMILLER: Wait a second. I did that wrong. Hang on. All right, so normally this is where I would tell my Inventor students, pretend that I just dimensioned this entirely, because you darn well ought to. I'm actually going to make this horizontal to the origin though.

There we go. And a cool thing about Fusion, that horizontal and perpendicular are actually one constraint. You don't have to guess which is which. Whatever it's closest to, Fusion is going to do that. So generally people Sketch in the way that would allow that to always succeed. If you've got good habits. At least my good habits help me out.

So there's our slot. So you mentioned that not everything is sketched, right? We have placed features as well. So I'm going to click on this edge, and the right click menu. So always explore your right click menus. They're are all context driven. I've got something selected, so I've got a different right-click menu up here than I would have if I had nothing selected. Fillet and

chamfer are on the list, because, well, what do you want to do when you're selecting an edge? You probably want to fillet or chamfer it. So I'm going to start that and just go ahead and add the rest of the important edges to this. And maybe I'll add some more in a minute.

Go ahead and just drag that down. This could obviously be, you know, all fully parametrically driven from your list of parameters if you're working that way. I'll do that in a second. Oh, wait, I'll do that in a second. So, we'll forget that for a minute. So there's our, you know, there's place features, sketched features.

And our timeline. Let's go ahead and play our timeline again. So you can watch the model as it's being developed. And it's cool for learning, but also, let's look at other features in the timeline. If I wanted to shell this, right now is not a good time. Or I guess it would be a good time to shell it. But, let's see, start the shell command. Pick the faces that need to go away. And drag it in, and suddenly it's a plastic or otherwise cast part.

Now, if I, right now I realized I made a mistake. This back corner shouldn't be sharp like this. It also needs to be rounded. So going back in time to edit that fill it is as easy as this. But what if I want it to be a different fill it, or I don't want to get this fill it mixed up with those fill its. This is just an arbitrary example.

Moving the timeline marker back is also how you go back in time, if you need to insert a feature prior to something else. So I'll just go ahead and add a fill it here, using the press pool command, which interprets an edge selection as a fill it looks horrible obviously it's been a while since I was a real designer. What's up?

AUDIENCE: [INAUDIBLE]

PHIL EICHMILLER: Oh, yeah, yeah, so I'll do that too. So you can see that I added this fill it, and the shell rightly b became confused. That was a new face. I introduced a new face. So this is what I mean by you break your own models, right? That's fine. I can edit this. Put that face into the set, and continue on. Because I had to include that in the shell.

AARON MAGNIN: To add that third face, you held that keyboard shortcut there that not a lot of people know.

PHIL EICHMILLER: Oh, yeah, I held down the command key. So it's-- here's a good example of it. So if you're in the fill it command, or something like that. And you want to add more edges, holding down the command key, I'm on Mac here, so I'm holding down the command key. It removes the preview and allows you to go ahead and pick more edges if you want to.

So that's something key to the fill it experience, certainly because you tend to add and remove fill its from your fill it sets all the time. So just remember there is a hot key that takes-- gets rid of that preview so that it allows you to pick more stuff. I can't pick more stuff right now, because there's actually a previewed size.

All right, so--

AARON MAGNIN: I was waiting for you to do that.

PHIL EICHMILLER: Right click, roll history marker here. So if you have a big time line, and you don't want to have to fiddle with sliding this timeline marker around, which could be tough if you're on a great big monitor, and you're trying to-- why drag that all the way across the screen when you could just send it there with a right click. Or send it to the end when you're done.

So there's all these controls all work for that. Now the question of materials. Let's actually go before the shell and add one more fill it. Here we go. Make it a proper part, there's no draft on it, you plastics engineers can beat me up later.

So there we go. And let's look at what's available for physical and appearance materials. So the workflow for adding physical material, this is, I tend to do it this way, is add the physical material and then decide what color it is, right? That's exactly how you would design the part of your building in real life. It comes off the machine as aluminum, and then gets anodized to be red anodized aluminum at some point, right?

In this case, we're going to go with plastic. We'll make it black resin, shows up horribly on screen. OK, hang on, wait a minute. ABS. There we go. So it's now, this body, now has the properties of ABS plastic. It's got, you know, what we list as density for this. Obviously you can go in and tweak these materials. Have your own materials, if you have different densities, and things like that.

It's got the center of inertia and the center of mass, and all that stuff. So a robust set of analysis tools to say what's going on with this body here, if you're at the analysts level. Let's look at our appearances. Appearances, you know exactly what it is. It's just the appearance of the thing. So every physical material has an appearance. But then you can override that.

So I wanted to draw your attention to-- I know this is not a wood part. But it's just so cool, our solid wood. We have a very unique moment, and this is going to be great for a live demo. I'm

going to download this material right now in front of you guys.

So if you open up your parents materials or your other materials, there's a stock set of stuff that shows up right? There's no upload button here, because it's already in my fusion world. Or it came along with Fusion when I installed it. Right now, I want to click Download material it's already here. Drag it onto the model.

And we've got this stuff called solid wood, that does not require UV mapping to go around all these edges and fill its and stuff. How many times you put on a texture somewhere, and it's like, the fill it just looks stupid. And you're like, Ah, this sucks. And then throw the coffee cup. Well,

let's change this. Let's do the texture map here, because nobody would ever cut this out of wood, right out of the middle of the tree. Whoops. Hang on a second. Let's try that again. Texture map controls. I get this manipulator. I'm just going to move this off to the side so it looks cool, and you can see all of that stuff maps across all of these surfaces.

And by the way, it just renders spectacularly. So this vertical face isn't helping, but that's OK. So you get the idea. That's like with all the other colors, right? There's different powder codes. There's paints. I mean, there's glass. There's just all kinds of cool stuff that--

This is another one of my favorite ones. Oops. Let's get that on to the body. There we go. Glass with bubbles. Well, that doesn't look very bubbly. Anyway, you get the idea. So, let's see, where do we go from here? Any other questions? Did I answer your questions? Everybody satisfied? Yes?

AUDIENCE: [INAUDIBLE]

PHIL EICHMILLER: Yeah, there is a thread option. Let's go ahead and do that. I want to turn this back into something that looks reasonable. And so, here's a cool thing that you can do with appearances. So you're just-- that's a great segue way, so this ought to be-- let's just say this ought to be aluminum.

If I drag this on top of any of these swatches, it actually replaces it. So if you have a complex model, and 47 out of 150 things are the wrong color, and you know where that is, just drag the new appearance on top of it, and it changes all of them.

So I'm going do this. We'll go back to this nice light color. that's Easy to see.

AUDIENCE: [INAUDIBLE]

PHIL EICHMILLER: I think it's internal right now. I mean, you might be able to expose it through an API or something like that. I'm not really sure. That was a great question. Whoops, hang on. I'm trying to grab the menu here. My glasses need tuning or something. There we are.

Thread command. So Fusion comes with a powerful threading tool. And let's see, you can have modeled threads. I might be blowing through the shell right now, as I'm doing this. Modeled threads are great for 3D printing. Do not put them in your giant assemblies. You're going to wreck the performance of your assemblies. Look at that helical edge there. I mean, 200 or 300 threads in an assembly for every single bolt is just thousands and thousands of things that Fusion doesn't really need to think about.

So you can have modeled or just cosmetic threads, and obviously go through and pick all the different thread types and stuff like that. There's another cool thing about this. When you import a model that's a step model, for instance. I think I've got a sketch visible, and it's bothering me. There we are.

When you import like a step model, and it's going to show up with, who knows, right? If it's modeled in Inventor somebody could have modeled it with the tap drill diameter or the major thread diameter. Or whatever, right? Who knows what size that cylinder is that used to have some graphics pasted on it like the other one does over there. Right? Maybe you want to switch to metric. OK.

I'm going to start the thread command. And I want to pick this face. And I want to pick something really just ridonculous. I want to go down to number four. It just resizes the hole. So instant joy on importing models and going, that's this. That's this. That's this. That's this. And all the holes resize. Obviously within reason, if you ask for something that doesn't fit on the model, you're going to break things. But I just think that that's fantastic. So let's have round of applause for the thread command.

All right, I think that's it for the model demo here. Is there any other questions? We've got about a half hour left. Boy, this took a long time, but the modeling parts the best part.

AUDIENCE: How does Fusion let you know that you can't do what you're trying to do? [INAUDIBLE]

PHIL Well, let's see if I can invoke an error from the fill it command. Hand on a second. It's acting up

EICHMILLER: on me here. So you get this error over here. And you can click on More Information. And it did something. Hang on. There we are.

So thread face references lost. Oh, I wiped out the thread. So there we are. The fill it wiped out the thread, and it told me what was going on. But you'll get an instant warning as you're doing something that you drag it past the limit, or whatever. You've just got to pay attention. So it will stop you.

AUDIENCE: Does it Auto Save in the background?

PHIL EICHMILLER: If you have your preferences set to-- so automatic recovery time is set here. And you can have-- so that's basically where it's handled. We don't want to make versions of everything, right? Versions are very important. They're specific milestones. But you can set this to automatically recover, and so between those two, I think you should be fine.

AUDIENCE: [INAUDIBLE]

PHIL EICHMILLER: Oh, I'm sorry. So this is in preferences. So under your name, let's, yeah, this is a nice way to wrap this section up here. Under your name, it's preferences, right? These preferences go with you from any machine. I go in and install on a completely alien machine to me. It's going to find my preferences and pull them in for me. And it's just basic stuff for running the program. But right up here in the front, you have your recovery time. Whether or not z up is awesome. And you know default units, junk like that.

Over here on the Help menu, you have a little bit more, you know, sort of what's outside of Fusion, the next layer outside. Like getting to the community forum or the gallery. Or tell us what you think. Graphics diagnostic. If you think you have graphics problem, you can run that. It will like sniff all of your graphics stuff and tell you if it's up to snuff or not. I will ask you for that if you're showing me pictures of obviously bad graphic experiences. I'll say, well what are you running?

Well, yeah, 256 megabyte card is not going to do this, perhaps. Or perhaps it will. I don't know. I had a great Mac that could handle-- that could swap out all that memory really easily. So that anemic card was terrific.

Anyway, all right, let's go back to-- Oops, there we go. I just zoomed off into outer space, and I'm trying to ham handedly switch back to this. There we go.

AUDIENCE: [INAUDIBLE]

PHIL EICHMILLER: So what you're saying is something like a named view, like if I really care about this detail, and I don't want to have to come back to it by manual navigation?

AUDIENCE: [INAUDIBLE]

BRYCE HEVENTHAL: He's talking about look at. Yeah.

PHIL EICHMILLER: Oh, yeah, the look at command. So if I had an oblique face, I would, and that's an orthographic face. but, yeah, look at look s any face orthographically.

AUDIENCE: [INAUDIBLE]

BRYCE HEVENTHAL: Oh, you could hit zoom to fit, and it will it right into your screen.

AUDIENCE: [INAUDIBLE]

PHIL EICHMILLER: Also, the view cube home takes you-- should bring it back.

AUDIENCE: Double-click the [INAUDIBLE].

PHIL EICHMILLER: You know, if you're on Mac. Where is it? Come on, do it.

BRYCE HEVENTHAL: Two fingers.

PHIL EICHMILLER: There we are. Two fingers zooms to home. Two finger tap.

AUDIENCE: [INAUDIBLE]

PHIL EICHMILLER: Yeah, that's where the name views would be. So if I'm really focused on some area of detail that's deep down inside some machine, I could be there and just say, right click. And say, New name views.

So it's the kind of thing that you capture on the fly. You're like, oh, this is the second time I've had to go find this particular little crevice somewhere. I'm going to save that this time, and then I can also name it, and say, you know, hard to find detail.

AARON MAGNIN: You can also right click on components in the browser, and say, find in window, and it will go and find it in the window.

PHIL EICHMILLER: Yeah, we have lots of navigation tools.

AARON MAGNIN: Yeah, no shortage.

PHIL EICHMILLER: Yeah, where is that? Oh, there we are. Find in window and find in timeline too. So if you have a timeline that's several meters long and runs off the screen, you can get this, you can get it to focus on the timeline where you're at.

AUDIENCE: [INAUDIBLE]

PHIL EICHMILLER: Yeah, anyway. I got coffee hands. Man, I'm right clicking badly. All right, so let's move on. Sorry. Any other questions, please just shout them out?

AUDIENCE: [INAUDIBLE]

PHIL EICHMILLER: Yeah, yeah. No problem. It's right here, under parameters. I'll just throw it up here. Pop it open. Yeah, except we do it better than Inventor because we put the user parameters at the top. And the model parameters, they are all collected under the various labels for what they are. So there's all this stuff in Sketch one. There's the parameter for fill it one. That kind of thing. Yes?

AUDIENCE: [INAUDIBLE]

PHIL EICHMILLER: To what?

Yeah, yeah if you're working in Cam, you should expect a fully parametric experience.

AUDIENCE: [INAUDIBLE]

PHIL You're asking the wrong guy. Yeah, I'm not a Cam dude at all. I'm a modeling guy.

EICHMILLER:

AUDIENCE: Anybody?

PHIL EICHMILLER: Anybody know? How easy is it to export?

BRYCE HEVENTHAL: --tool pass to Mastercam?

AUDIENCE: [INAUDIBLE]

PHIL EICHMILLER: I mean, you might wind up just, it-- you know, there's our traditional workflows where you would export the model as a step file and go machine it somewhere else if you can't get it done in Fusion with your current tool set. So ask the Cam forum. That's where I would go. There's a whole page for Cam on our forums. I'm sure somebody there knows for sure. I wish I had a better answer for you.

All right?

BRYCE HEVENTHAL: All right, so, yeah, I'm going to breeze through this pretty quick. You have a lot more to show, right?

PHIL EICHMILLER: Yeah.

BRYCE HEVENTHAL: So, this as you can see, is just an introduction to Fusion 360. There's a lot of other classes you can take. This one in particular, I'm going to highlight. This is Alex Lobos, a really great speaker, and Colin Smith, one of the PMS on the Fusion team. They're going to be showing you how to use T splines to make photorealistic in Fusion 360.

There's this integration to textiles in Fusion 360, so it's going to use another app. And that's a big part of fusion is, if you need to do something niche, you go to the app store. You'll find something that will help you to get your job done, for whatever it is that you need to do.

Fusion 360 modeling tips from the experts. That's going to be a great one. Peter [INAUDIBLE], the guy who developed rule number one. A big help in the community, was just recognized last night actually for his involvement. He's going to be doing that along with Jeff Strader.

And a couple more on here. Designing for the circular economy, so cradle to cradle, green design. I think everyone's hot on that these days. It's a good way to be. Use direct Modeling in Fusion 360. This is a part of Fusion 360 I absolutely love. I mean you can do some amazing things with direct editing inside of Fusion 360. I'd recommend that you take a look at that one. The guy who taught me and Bryce, Brad Tallis is the guy running that show.

And this one is also very interesting. This is this is a story about how we use Fusion 360, 3D printing, and the like to help a paralympian, Denise Shindler, and a really good class to take a look at. And I think we have some more demo. I'll just stay over here.

PHIL EICHMILLER: All right. So I did want to show you some stuff about assemblies. And that's what I'm going to do. So we'll talk a little bit more about some of the stuff that we just discovered. So I know this doesn't look like much. It's a makeup compact. I had to come up with something that was simple to do. I make all my own models. And so, right now this model is fairly well developed.

If I start to move it apart, you can see I have both halves of this thing, and they're not jointed together. A word about what I just did. In Inventor, it's called free move. You can tear apart your assemblies, and then hit update, and rebuild the thing. This is that button.

So we're always keeping track of where components are. Bodies don't get into that, because bodies travel around inside the envelope of a component, right? But components are the ones that are jointed together and are the full part.

Well, we have a positional modeling tools, and it's called capture position. So if I move this out of the way, Fusion assumes that I'm moving it into some position that this mechanism is allowed to go into. And that I need to do maybe a sketch, figure it out, I need to make clearance for this object as it's passing through something, as I'm moving it. So that's why it's asking me to capture position right now. It's like, do you really care where this is? Because we're going to capture this, and it becomes part of the parametric history.

So if I roll back before that, before and after that, you can see that it's there for modeling purposes. And I'm going to go ahead and back up here and revert that. So let's look at, how would we solve this problem? And I'm going to solve it with an as-built joint.

Now how did we get here? This is a top down design that starts off with a T spline body, and then I do a bunch of modeling to it. I'll show you that in a minute. But let's put the as-built joint on. I referenced this earlier.

When things are modeled in one place in a top down design, And you want them-- they already have a relative position to each other that has to have some motion to find. You need to add a joint. So clicking on the as-built joint command, you see that you have a list of all the regular joints that we support. You know, rigid is like glue. Revolutes like a hinge. Cylindrical is just like something on an axis together.

We want a revolute joint. And I want to pick the two components. And then it starts looking for points of interest. And right now there is a little shaft that runs on one all the way across. And there's a little pocket for it. And so it's saying, well, where do you want the origin of this joint to exist? And I want it to exist on that axis.

So the mouse is pretty smart. It's only looking for specific stuff down in there. And it's easy to find that. And it puts in the revolute joint. Now I have a more realistic product definition here. Now I can go through the joint. I'm not going to go into-- we don't have time for that right now, but I can set joint limits and say, this is 0. And this is as far as this opens. I could turn on contact, and figure out where the plastic tries to pinch and figure out if I needed to work around that in my model, that kind of thing.

And of course, I can always revert this or capture the position. I mean I might actually care to show this in an open position and go ahead and capture this, since I'm pretty close to the end of this design. Maybe I want it open and shut for renderings or something like that. So it's let's talk about how this model was created. And I want to play the timeline on this thing. And it starts off as a T spline, and then, I go through and pretty much hack it up, and add the stuff. It's not very compelling actually. I wish I could stop it at the moment, but it will go all the way through. And it plays it all the way through.

What's at the beginning of this is our sculpt tools. So when you're in Sculpt, we have these T spline tools for modeling. It's basically what you would find in Rhino if you're a Rhino user. That's how this initially was created. So you know, I use it for creating plastic stuff, and that's why I was doing this. But if we edit this thing, add a little bulge to it there, hopefully doesn't break anything. And finish the form, the model now has a bigger bulge on the top, right?

So this is an example of how direct modeling works inside of Fusion. Everything that falls after a direct modeling node, whether it's a T spline, or if you come up here and you go create a base feature, which is a parametric free feature. Where it just is basically working without parameters. Everything that falls after that, obviously listens to those changes.

So you can go back into that history free node, make some history free changes. And then everything that follows after will update. Any other questions about assemblies? This isn't the most compelling assembly example, but the problem is that I don't want to just sit here dazzling you with complexity. Those of you who are going to work with assemblies, you'll find that these tools, I think, get you where you want to go if you're used to using Inventor and that sort of thing. Yes?

AUDIENCE: [INAUDIBLE]

PHIL EICHMILLER: So different parameters for each instance?

AUDIENCE: [INAUDIBLE]

PHIL EICHMILLER: You're talking configurations and stuff like that? Or--

AUDIENCE: [INAUDIBLE]

BRYCE HEVENTHAL: Yes, we have a component pattern. So we can pattern faces, bodies, components, features. So you'd be patterning components, and then everything would be driven off of that first seat component.

PHIL EICHMILLER: Yeah, so-- it's not going to let me. So now I have two instances of the same component. And then I can go in and edit, you know, the original sketch, and they're both updating.

AUDIENCE: [INAUDIBLE]

PHIL EICHMILLER: So what I would do is then, I would copy the component and supply new parameters for whatever the height would be that would be different, and use, re-use the parameters where it was appropriate. So that just gets to a deeper use of the parametric table, right? There is a value for the length and width and height of this thing. And so the second one would just have a different height.

So you could refer all of the dimensions and parameters back to the first one, if you want. So it's kind of just depends on how you want to tackle that if I want this to be independent, then I would just actually copy it, and paste new, and it becomes a new thing. It instantly broke a couple of things, because it's not a parametrically complete model.

I didn't build it using rule number one. This is why you use rule number one. Because you'll get sick features in your time line when you copy parametric stuff that's incomplete. That's just telling me that there's cache data in there, is what it's saying. That it had to make some assumptions about that sketch and what was happening there.

So use rule number one, and paste new. And then you have a completely independent thing, and you can point parameters back and forth, or just build another one and point to some parameters and not others. So

AUDIENCE: [INAUDIBLE]

PHIL EICHMILLER: Yeah, yeah that would be a direct modeling thing. So hopefully that answers your question.

BRYCE HEVENTHAL: Awesome, that was a riveting assembly. So if you're looking to get more advanced assemblies out there. We have several assembly classes. One of our customers, Eli, he does a racing drones. There's like a whole series now, it's pretty awesome. They have a camera and you wear these like VR goggles and you race these drones around the like a football field. He's going to talk about how he designs those. It's going to be an awesome class.

Then we have the master assembly class. So if you're really diving into assemblies and need to do top down, bottom up, different various assembly methods, we have a master assembly class. And then finally, we have a future proofing products class with Fusion 360. This is for customizable components. We see a lot of companies out there getting into customization up front in the design process, for instance, like for prosthetics or wearables. So this is a class of how to think about that up front in the design process.

But now we'll jump in again.

PHIL EICHMILLER: There is my hero. All right. I don't know if I can say that in mixed company. But anyway, let's talk about-- the last thing we wanted to show is just getting something out of Fusion, right? So we have the drawing. We have 3D printing. And we have rendering.

And so I'm a big fan of rendering just because I've seen what it can do for business over time as it's developed. You know I've been with Inventor for a long, long time. I've been teaching it for eight years. And I've seen companies that didn't take rendering seriously at all suddenly thrive because they connected up their engineering and sales departments just with pictures.

And so, that's almost like a ubiquitous and just-- it's almost like taken for granted these days, right?

Well let's see what it's like to get a good rendering out. So go to the render environment, and hit go. Now there's a lot of stuff going on here. There's, you have, this is just local rendering. Right now it's using my GPU and CPU to figure out, where is all the light bouncing around in this thing this is done with that solid wood, and it's just going to sit here and bake until I stop it.

I think there is a way to limit that, in case you're not, if you know, you know go out to lunch and come back, and your machine is on fire or something. You probably don't want to do that, but you just let it go until it thinks it's ready, and you stop it. Now obviously your mileage may vary. There's you know, if you have an anemic video card, it's going take a little while to do all the figuring out and stuff.

But eventually this gets to a nice rendering, and I can hit Capture Image. That's just the simplest way to manage this. We've added some new controls. It's not-- maybe I'm accessing it in the wrong spot. But-- and it'll save it here. You can save it to your computer. And it also winds up in your data panel. So if I hit Save, it'll drop it somewhere, and you'll find it eventually over here when it's done uploading, and it shows you that it's uploading.

Now I'm wrecking the ray tracing here. So we're torturing this machine pretty good. It'll show up eventually. Anyway, if you just want to quickly figure out what's going on, you have different levels: quick, normal, and advanced. If you look down here at the rendering gallery, you'll see that it has a stock list of top right front views of your model.

If you have named views, it will render those. You save named views. You can have specific views. That's how you get them into your cloud rendering. So--

AUDIENCE: [INAUDIBLE]

PHIL EICHMILLER: These things are being done in the background for you. So you can do your own point and shoot kind of stuff using named views. Or you can just rely on the cloud. You can send more complicated jobs up to the cloud to let it work while you're busy doing other stuff. So I'm not a huge expert on the cloud workflows because I just love rendering right on the screen because that's the satisfaction I get out of it. I don't like to let robots do that kind of stuff for me.

But that's-- if you don't want to tie up your machine for two hours doing a nice turntable

animation or something, then use that.

AUDIENCE: [INAUDIBLE]

PHIL EICHMILLER: The environment library, these are all HDR images. It's all image-based lighting. It works the same way as materials. You can download new ones. You can see that some of these have an image in the background. So if you want to use that image instead, you can do that. Let's see, let's go ahead and put this chair outside.

Now it's outside, and we're going to go back over here, and instead of a solid color, I'm going to use the environment. Go ahead, and unflatten the ground. And zoom it up, so now it's a giant chair sculpture in the middle of some courtyard somewhere.

But you can say it's taking all of its shadows and everything from the background. Look, it gets really big. Now it's like, now we're in Denver. Anyone here from Denver? Really? Go to Denver, and look at all the giant sculptures they have. You'll see why I'm joking around about it.

But you can also set cool things, like depth of field like you know you want to have-- a little blur that shows that your camera is working. This is a horrible example of it. I should have practiced this a little more. But you get the idea. You've got full camera controls over here. You can fish eye it all you want. That kind of stuff so.

All right, I think we can move on from this. Let me get out of rendering so it doesn't consume all of my battery.

Oh, I forgot to show you where the 3D printing thing is. Hang on one second? Where's the-- file menu, 3D print. So you can just go straight to your 3D printer right out of the model, and it'll you know whichever one is your favorite, it will fire it up and use it. So just wanted to show that last little thing.

BRYCE HEVENTHAL: That's chair, by the way, was made by Arthur. Is he over at Pier Nine? Yeah, he's he makes some amazing stuff. I pulled it off the gallery just to use in the example today. So get on the gallery. Check out those models. It's pretty cool. It's a nice way to, like Phil said earlier, it's a nice way to learn how to make different products, so--

There are some additional simulation classes I wanted to point you to. We're having a Kegger. Sounds like fun. I'm going to probably try to attend that one. I hope there's free beer. It's about

simulation. I think it's by James Herzog and a couple of other people. Great example of using Fusion 360 for simulation.

Don't get stressed out. Another one Wednesday. I don't have a lot of details on that one. Sorry about that, but--

This last one is something that I think everyone should pay attention to. I mean contacts and simulation is something that a lot of people struggle with understanding which ones to use at what point, and what advantages there are to using different types. We recently released a new way to manage your contacts in the November release that just dropped last week. So take a look at the What's New videos on our YouTube channel. It's a nice way to learn about these new features and to learn about other things.

AARON MAGNIN: Yes, so we're constantly updating Fusion. That's one of the different things we've done from other CAD tools out there. We update every week to two weeks, whether that be the back end, the front end, the desktop client. So you're constantly check in to see what's been changing. We have a couple other classes I'd like to talk about.

The first one is for 3D printing. If you're using like a maker bot or small desktop printers, great class to talk about best practices for 3D printing using a couple of different tools. And then we get into the CNC aspect, the making.

We have a couple-- a whole bunch of classes on CNC. There's only a couple left open. But any of those out there who are interested in CNC, we actually have a class by NYC and CNC John Saunders. It's an all-day class tomorrow. A few spots just opened up. If you guys want to get really-- it's a hands-on class where you go from start to end, and you actually machine some stuff on the Tormachs. Feel free to come up to me afterwards, and I'll get you guys more information on that. I think there's about three to five spots left.

And then finally, we have things for taking scans in. And then using, excuse me, recap and remake and Fusion 360 to take in those scans and redesign it and re-engineer it. And turn those into solids. It's going to be a good class. And then finally, we have a class on branching and merging. We have branching and merging coming to the Fusion 360 very, very soon. This is for collaborative design. You can branch off, do some other things to your design, and then merge it back into the master branch. Some really cool workflows with that.

And then finally, if you're a developer, I see tons of developers out there. And you love staring

at scripts. We have an API class for automating a lot of things in Fusion 360. And we also have another one to create custom commands within inside Fusion 360.

And finally tomorrow, Wednesday, there's a main stage presentation right about 10:00. Fusion's got a big, big portion of it. I would highly recommend going to see it. Our colleague Stephen Huber is going to be presenting on Fusion 360 and showing what we're going to be, where our direction is going for Fusion over the next year. It's going to be an awesome keynote, I'd highly recommend going.

And then we're going to all be at the answer bar or in the manufacturing booth, so if you have questions, feel free to come up to the answer bar. Give us feedback. We have tons of tips. You'll get stickers and a little swag over at the answer bar. And we'd like to thank you guys for coming today. If you guys have any questions now, we could take it. As well, we'll be here after this presentation if you want to come up and ask questions as well.

PHIL EICHMILLER: I'll be at the answer bar Thursday afternoon.

BRYCE HEVENTHAL: And he'll be at the answer bar Thursday afternoon if you want to complain.

AUDIENCE: [INAUDIBLE]

PHIL EICHMILLER: No problem. I hope you guys enjoyed this introduction to Fusion. There's obviously, there's many more classes that take you much deeper into it. And please don't feel shy about filling out the feedback form for this class. And come up to me and talk to me anytime you want.

BRYCE HEVENTHAL: Thank you very much.