JIM SWAIN: As I was saying to when a couple of people were here earlier, my goal is to let you know what my workflow is and-- is this up a little high on the volume? What's your thoughts in the back? OK, I'm going to keep talking, but I'm just curious whether it's too much.

I'm going to give you a feel for my workflow and why. And I think honestly the "and why's" are the most important thing about this because that way you know what my assumptions were, what my constraints were, when I was doing product design and how I would use the software that's available now to do this. I'm not going to go deep into the software-- just not time to go really, really far deep into everything. But there are a few areas I'll dive deeper than others. I'll let you know where they're going to be.

And this is a session right before lunch. If you're willing to give up some food time, some time down on the exhibit floor, I'll be glad to talk with you a few more minutes, maybe fire something up here and go that way. Does that sound reasonable to everybody? OK, good. If not, I'm up here anyway.

So that's what was put out on the catalog. I was a little bit annoyed because they didn't say anywhere that I saw on the courses as far as what the assumed level of user was. Was this an advanced class? Was this a basic class? Because I did put this up there as a basic, so hopefully it meets everybody. I've got another slide hitting that too.

So we'll take a look at some workflows involving both derived parts and multibody tools. I'm going to hit multibody harder than I am derived. I need to do something here?

I was going to try to fix your ratio.

Oh did it? I think I'm OK for inventor. Let me see. Nope, we had it fixed about 5 minutes ago, probably when I switched back over to VGA.

OK, so let me just put the PowerPoint back up. If the resolution dances while we're doing that, no big deal. OK, so hitting multibody solids, part design, type of thing a little heavier-- there we go-- and do that heavier than derived parts. Talking about Moldflow Design, that's the one piece of software I'm going to hit today that's not included in Product Design Suite.

And full disclosure, yes, I work for a reseller. No, I don't get commission on software. I'm not really trying to push the software down your throat with this, but I see value in it. I want you to

see why I say that, and you can decide for yourself. Say yeah, he's right or nah, we're not going to worry about it.

Showcase? Yeah, that is in Product Design Suite. I'll talk about how I would use that or do use that for doing part of my product design workflow. And then the 3D print environment.

So my name's Jim Swain. I like this. I don't have to wear a badge temporarily. I don't have that thing swinging around in front of me as I move around. So that's pretty good. I will put it on later so in case you're going, who's that again?

I'm what's called currently an Applications Consultant for a reseller with headquarters in eastern Pennsylvania. You could say Applications Engineer, Solutions Engineer, not really Software Engineer. I'm somebody that in the past, you would think of the person that goes out and would do the demo. Well, no.

I demo. I train. I support. I would consult with customers on implementations. They're basically my customer for life. So that's the way I approach things.

Before coming to Synergis, I was a design engineer for a lighting controls company. And in that job, picture a dimmer that you might go to a big box store like a Home Depot or Lowe's, and you see the one aisle that's nothing but switches and the first half, it's dimmers. Yeah, I've got some stuff in there with my patent on it. I'm kind of proud of that. We can talk later when it's not on the recording.

But a lot of plastic part, some sheet metal work, general design work went into that job. That was a good, what, seven plus years there. So design testing of the product before it goes out the door, both life testing and UL required testing.

We owned our tools, the molds for making the parts. We own them. We didn't create the tools. We didn't design the tools, but we owned them.

So as a design engineer, part of my responsibility was approving the tool design. Basically, it worked out, I wasn't the one that was figuring out gate size and type of steel to use and everything. But better know what's going into it because it was my name on the drawing. It was my name on the PO. And you can just picture what else was involved with that.

A group of us got together after a couple years and realized we're hiring a lot of people straight out of school. And those folks didn't know anything about injection molded part design.

It was very different than the machine parts that they had been taught and played with in the schools.

Now, remember this was back-- oh, I didn't say it earlier, but this is probably 20 years ago. So more and more schools are now having injection molding build into a design curriculum. Oh, there you go. I got engineers disease, never use a small word when a big word will do. I'll try and keep it to a minimum in this class, but it's going to slip out.

So we developed a course in-house for teaching incoming engineers the basics of designing plastic parts. I have a couple of bits of that coming into here, not a whole lot but just a little bit. I also took that concept, work with a local community college to do a plastic part design course at the community college level. The local college was setting up a plastics program-- designing tools, designing parts, running tools, all that.

Unfortunately, it kind of fizzled. The person that was heading it passed away. And so the program kind of died with them.

So quick feedback from the audience. The pace I'm speaking, is this a reasonable pace or am I talking too fast? All right, because I got a case of the jitters. I'm in front of an audience here, so I just want to make sure that I'm not just going right through it. I don't want to be that.

If at any point you have questions, I'd like it if you could hold it to the end. But if it's something that I just showed something-- hey, can you show that again, whatever? Bring it up at that point. Fair enough?

OK, again, basic level class. I'm going to go beyond scratching the surface, but I'm not diving deep on most of these tools-- I'd say all these tools. Again, the offer's out there. If you want to go deeper afterwards, we can spend some time.

How many of you have Inventor experience? All right, pretty good. Those of you who didn't raise your hand, are you looking to go into Inventor? Or yeah, maybe. OK. Again, I don't get commission. I'm not going to push it down.

All right, how many of you have plastic part design experience already? All right, so you're just looking for a good class to catch a nap in before lunch? That's cool. Give me fives. I'm happy with that.

Hopefully, I'll show you some things that are maybe a different way of looking at it. I'll help you

out that way. Besides this class-- first of all, the hand-outs got a lot more information than what I've got on my slides-- and I'm not going to death by PowerPoint-- and also more information than what I will show as far as setting up the different tools and so on in that handout.

There's also a lot of information in other AU classes out there. Inventor Flexible Modeling, I think that was on Tuesday. Getting Your Inventor Derive License, the derived part workflow was yesterday.

3D Printing with Inventor, also yesterday, and then Combining Inventor in 3D Printing in Early Prototype Development, that's this afternoon at one o'clock. So if you haven't explored deeply the 3D printing environment and you want to, there's an opportunity for you. And of course, you can go and download those handouts as well.

Help file within the Autodesk world, both for Inventor and for Moldflow Design. And there's a lot of good information there but also basic tools.

The resin manufacturers, we developed the course at Lu-- I'm not going to say the name, sorry. The company I used to work for. And even more so for the college course I did, just gleaning information from the different manufacturers' design guides for the different resins. Back then, it used to be General Electric had a wealth of design guides.

And you get a little bit of information from the VALOX. You get another little bit of information from LEXAN. You get another information-- and we just, OK, here you go. And we combined that with DuPont and with a few other manufacturers.

GE's changed, what, names three times in that 20 years since then. They don't go by GE anymore. For LEXAN, it's now SABIC or something. But they have design guides online, some really nice, interactive guides and things for the basics of plastic part design, as well as books and stuff that you can find, or other design guides or Wikipedia or YouTube and that for going beyond what we're hitting here today.

So with that being said, I'm going to talk a little bit about what the challenges are as a designer coming into having to do plastic parts. We'll take a look at the derived part workflow and the multibody part workflow. Is multibody new to anybody here?

Wow, OK, I'm going to have you guys out to lunch early because everybody knows it already. Yeah, sorry, you're going to have to sit through a little bit of it, but I'll keep it to the level that

seems appropriate. If you haven't seen multibody solids where you're working inside a single part but multiple solids, hopefully, I show you something that will pique your interest because I think it's a wonderful tool for the idea of maybe designing the next mouse or the next 3D mouse or that keyboard they have down there in the exhibit hall. Did anybody play with that, the split keyboard where it's a mouse in each hand and half the keyboard in each hand?

I'm looking at that going I might be an old dog, but that looks like a good new trick to work with because I'm "ambimoustrous." I work the mouse left-handed or right-handed, and that's perfect because I already have a mouse in each hand in that case. So I'm going to look into that one.

We'll talk specifically about what tools are in Inventor for helping design things that are typical in injection molded parts-- hooks, snaps, lips, screw bosses, that kind of stuff. We'll take a look at that.

We'll look at Moldflow Design. This used to be known as Simulation DFM. Did anybody notice that I had Simulation DFM in the original catalog look-up when I was posting that back in June? I wasn't sure if that came through or if I got the correction in time.

OK, and then Showcase. 3D Environment, mentioned it before. And what hopefully is the most useful part. If I had all these tools back 20 years ago when I was doing this day in and day out for a living, or flip it around, if I was doing this now as my 8 to 12 job-- that's 8:00 the morning 12:00 at night-- what would I do? What would be my workflow?

And at that point, I'm assuming times for many, but if anybody has things that they would like to share as far as alternatives, I think that would be great for everybody. I do ask that you think about why you do it a certain way. Several of you were-- many of you were here when I was talking about that first thing with the class because that's just as important. Oh, I never use that tool. Well, why?

OK, yeah that makes sense. Or yeah, I understand it doesn't apply to what I do. So that's what I'm hoping that you can share. That's what I'm looking to share with you. OK, we're almost done with the first bit of PowerPoint, and I'm only- yeah, I'm right on time. That's scary.

So I just lined some pieces up on my desk before coming out here. We got a lid for a thermal cup. Ironically, the piece that it attaches to is an Autodesk Moldflow Users Group thermal cup that leaks because the seal doesn't hold. Hmm. I keep that just because of that.

My mouse and one of these things, as well as my keyboard and the laptop there. So injection molding, metal or plastics at this point, allow you to combine a lot of functionality within the parts. You can also get a wide range of geometries. You can do stuff in plastics that if you try to mass produce through a machining would drive you nuts.

Now, technology fast-emerging with 3D added to manufacturing, I think there's another revolution going on there. And I'm enjoying it. I'm enjoying looking at that. That's taking this further. It's going to change things that I think as much as what injection molding has done. So that's kind of fun being alive right now.

Part performance, it's got to get the job done. It's got to meet the design criteria. That might be aesthetics. The aesthetics might be we got a glossy surface on the mouse versus having that pebbly surface on the thermal lid and kind of a pebbly surface on the doo-hickey. That's the multi-charger.

Net Shape, yeah, parts got to fit together. Pieces for the mouse don't work real well if the upper piece is a couple millimeters wider than the bottom piece. Not exactly what needs to happen. That's not just with design. That's also hopefully you designed it to be manufacturable and stay within that.

It's got to be strong enough. I don't know how many times this guy's hit the floor. The battery door always comes off but nothing breaks. There we go, I think that's meeting the strength requirement.

And then the biggest challenge. I worked at a relatively small company. The owner was incredibly involved with the aesthetics, the product. Things for the first month would change rapidly. You'd be taking a different prototype to-- or two or three prototypes-- to see him every night for that time frame.

Meanwhile, you're also trying to start your design so that you could meet your overall goals. Just because he didn't sign off on the aesthetic, didn't mean the delivery date had shifted. Anybody else ever run through that experience? Yeah.

Anybody else put anybody through that experience? Good-- oh, leave now you.

[INAUDIBLE].

That's true. That's true. I'll have to give you points for that.

OK, from the manufacturing end of things, tooling ain't cheap. Both to design it, it takes a skilled person to design tool, especially something that you want to have last for more than a year or two. But also the fabrication-- things changed in 20 years obviously from the economy point of view. But still, it's a very significant part of the overall piece.

But as a design engineer, it's my responsibility to make sure that the parts are going to be economical. What I have control over that? Is this going to be easy to fill? Is it going to be easy to cool? OK, that means the cycle time can come down on the pieces, therefore, the price can come down on those pieces.

Is the geometry going to be forgiving? Do I need to inspect every single top coming out of the mold to make sure it's going to fit to have a hope of having this mouse be something the customer will accept? Or nope, we've never had a part that wouldn't go together and we're never going to because that's just how forgiving the design was.

And then, we did a lot of short runs, couple thousand pieces at a time, change color, do another couple thousand pieces, so on and so on. We are in a company that would run a million pieces at a time. So the amount of time it took to get a tool up and running at the beginning of a run was a significant part of our per piece cost. So we tried to design things, and especially the tooling, but run our systems and such that would allow it to cut down on that.

What I'm going to hit mostly today, though, are those three in the middle. What can we do from a design point of view to try and keep the cost of production down?

OK, so flexible design-- no, I think we need more of a scope on that surface. Really? The electrical engineers just took a larger capacitor in there because they said the performance wasn't working. And now I got to shrink down what's going around that capacitor. Thank you very much. Never phrase that that way to my boss. I was a little smarter than that.

So Derived Part Workflow. You've got an assembly, single part, something like. I'm going to make a new part that uses that as my parent. And I'm going to have my new one as a derivative of that.

Again, kind of quick poll, derived part workflow, how many people feel they're comfortable with that right now? OK, so 3/4. All right, I'll gear what I'm going to show on the screen live towards

that.

Multibodies. Multibody is great for-- again, we did one-person design teams, two-people design teams. Or if it was a larger group, each person had an area of responsibility. So I might be designing a mouse. I might be designing a handheld remote. I might be designing a faceplate or the faceplate system for a given product.

So the fact that I could have my entire design in the very beginning in a single piece yet still be able to design as if it's an assembly made up of batteries and circuit boards and battery covers and all that stuff. And that better work again now or I'm really in trouble. That multibody workflow, very handy-- and I got ahead a little bit on my slide.

What you can steal with the derived part, you can bring over information under the geometry, the solid body, the volume of the parent, whether it's one or more pieces in assembly, or just a single part that you're linking in. You can bring over sketches, have them available for you. So do you want the solid body as a dumb body, or do you want to be able to also get hold of information on the sketches?

Work features, plane-- work planes, work points, work axes, that type of stuff. Again, you can bring them into your new part, as well as the parameters. And I haven't done tests on them for a little while, but back in the earlier releases of Inventor, back before it was the year as the release name, it was definitely faster to derive something into another piece to get a hold of parameters than use Excel because the Excel links slowed down the performance. So you make one master part. Might have only parameters in it, nothing else.

And that would be derived into everything that you needed to use to get through it. I haven't tested that in a while, so I don't know if it still got that kind of constraint. But it kept Excel out of the mix.

Multibody solid parts. Well, essentially you are still working with a derived part. I'll come back to that.

Semi-independent, what I mean by that is I build this piece, it's rough shape, I split it out into separate bodies. If I come back and add some fillets or things like that, I can tell the software which of these parts that fillet's going to affect-- all of them? Only one? Some combination in between?

That's what I mean when I say semi-independent. No assembly constraints. OK, I was almost

expecting smiles, or maybe even a little cheer, when I said no assembly constraints. You don't have to mess with flushes and mates and inserts and that kind of stuff for pieces to hold together while it's still just a single part file. You're building it in the right location.

Then, when you've got things to the point you're comfortable with, you can kick it out to individual part files within an assembly. I will go through that. That, to me, is great because now, I could start doing the full out detail of the piece. If you want, you can keep it linked to the original solid body in case you need to do some late aesthetic changes, things like that.

But kick it out. You've got individual parts that you can now assign specific materials to, a specific bill of material information to, component, by component, by component. So that's what I love about that multibody solid workflow. What I hate about it is it's a real mouthful to say, especially in front of an audience.

So look at it another way. And this table is in the handout as well. If you want to compare what a assembly is like compared to multibody part, in assembly, each part already has its own material made that's probably generic, whatever your template file is set to. But each part already has it. Whereas multibody part, it's got the material of that master part.

The colors, individual parts can have colors in the assembly. OK, that one is fairly obvious. You can also change the properties of individual solid bodies to have different colors while it's still within the solid part.

Constraints, as I said, you don't need them with a multibody solid. When you kick it out from being a solid into being a separate part files. They come in in that same geometric location. They're fixed. So there's still no assembly constraints, but it's a fix-- they're grounded, so they stay in their position at that point.

And then, finally, tool bodies. Tool bodies, if you're not familiar with this, the idea of maybe have two solids. And I'm going to cut away from the one with the other.

So at an assembly level, you don't really have that. I could do a derive part from the assembly and say, this piece is cutting away this one to get that same result. But at a pure assembly, you don't really have that tool body, that cut away, join the Boolean operations, don't really exist at the assembly level. You do have those at the part level. So I can make multiple solids and then cut one away from the other.

That might be how I make the cradle for the batteries in here. I have a solid body for the battery and just cut it away from the rib. And there's my support.

All right, so let's take a quick look. You're going to see these slides come up a bit. And yes, that was the platform creaking. That's a nice loose joint here, and I'm waiting-- last time I taught, I almost pushed the chair back too far. So if you see me doing a windmill, get ready to run for first aid help please.

So kick back over to Inventor. And here, I've got a part I'm going to use as a starting for a derived piece. I'll start a brand new part right off of the standard template.

Typical operation, we'll go and start a sketch. In fact, I wouldn't be surprised if some of you have that already sitting there in your template file, have that sketch there. Or you have it turned on to create one right away. But instead of that, I'm going to go to the derive tool.

And you can derive really at any point you want to. So that other piece that I'm going from is right hand of case. So notice I am allowed to go and derive from full cases or single parts. There's the right hand of case in my new part.

Right now, I'm bringing over the solid. If I don't want it, I click on it. The same thing applies. If I want to be able to get a hold of the sketch, I can engage that as well.

I do have work geometry in there, so I could bring over individual work planes-- maybe, that one there. And I could get a hold of model parameters. I can get hold of reference parameters. Reference parameter might be a reference dimension. So if you're wondering what the heck a reference parameter is, typically, the only place I've seen them come up is when I put a reference dimension in the sketch.

I don't have any user parameters. I don't have any linked in parameters in this particular file. If I did, those would also be available. Common use for this, you're making a mirror image part. So I can just tag that mirror, left hand part made, right hand-- boom, there it's done.

I can also use a scale factor, trying to do tool design before the tooling software was available for Inventor. This is how I would make a piece to allow for the shrinkage of the part, so I could do a core and cavity for my injection mold. Tedious is a good phrase for what that job used to be.

Up here choices. If you had-- and I don't have any here-- if you had a feature that was at the

exact same level as this, do I want to see a line between those two faces or do I want to just let it merge? And here, I'm just keeping it solid as a solid body. This is a single solid, so it's no big deal. If there were multiple, you'd see them listed right at this level. So a fairly simple derive.

I'm going to kill this for a second. I'm going off script-- always a wonderful thing to do in front of an audience that's going to be grading your performance later-- and go back to derive. And I'll just show you the difference if you derive an entire assembly into the part file.

Would I like to update the assembly? Sure, why not? Error occurred. That's probably a reason I should've said why not, but I'll go ahead and accept it and keep moving.

So here, do I want each of these three parts? Well, maybe, I don't need the speaker. I'm only looking for the cases. So the choices are include, subtract it, include just the bounding box, intersect it. So here, I'm actually subtracting the speaker from it.

If I had pieces where this was might be the line connection across there, right now I wouldn't see a line in my final solid unless I kicked over to here, then I would see a line. And that's if they were exactly matching. Do I see that seam along the match? And this is pretty much a yes or no.

Over on this side, for any given piece, here is the same stuff that I had when I was just deriving a single part into the new part. So you have the overall assembly, and then, you have for each individual part coming in, what do you want to do? And that's really what I wanted to show here. You also have the ability to get into design representations, positional representations, level of detail representations.

And then, you're starting to talk 100 piece, 1,000 piece assembly. Maybe you want to remove any part that's smaller than a certain size. And that's a percentage of the overall bounding box of that assembly coming in. Give me everything but the little nuts, bolts and washers from that assembly that kind of idea.

So I just want to give you a quick look at what an assembly style of derive would be like. And this is what I meant by scratching the surface. So this is the level except for some of the other-- a few of the other tools that I'm looking to hit for everything. Is that going to meet everybody's expectations for today? OK, good. You heard me when-- that means a yes, OK.

All right, different from that is a multibody part workflow. And I do have a piece that I'll go and

kick out into an assembly. But let me just start from a very simple beginning. I'll make a brick.

So new sketch, rectangle. It's going to be-- oh, got three by four stuck in my head for some reason. So I'll kick it out to a three by four. Not doing anything fancy with it.

And when you go to extrude, your very first extrusion automatically makes a new solid. You don't have any choice, same thing with the revolve. The very first feature of a part has to make a new solid because nothing exists. Well, OK, unless you to turn it to surfaces. There we go, pretty impressive part, eh?

I'll go make another sketch right there on this face. Project that face geometry. Finish the sketch. And now when I go back into the extrude, by default, it just wants to add to the existing solid. But here, I can go tell it it's going to be a new solid.

And I'm going to kill a couple of two birds with one stone here. I'm going to go to the More tab for extrude. I'm going to give it a negative value for the taper angle. Negative means the volume is reducing along the length of the extrusion. Apparently the mic can't pick me up and my arms are over there. And that way you can see the one piece from the other.

The green preview also indicates that it's going to be a new solid created from this. That's because of this being checked, takes these other three out of the picture. Drawing, cut, and intersect aren't valid when it's being a new solid at this point.

So OK, now in the browser, two solid bodies-- and I'm a real big fan, whether it's my extrusions, whether it's my solid bodies, that I give them some kind of name that makes sense. Because who wants to look through a list of 50 solids to try to figure out which one that you're trying to work with. So that way I can easily tell top versus bottom.

I was saying that when I do something like a-- well, yeah, I'll do that right here. If I do something like a fillet-- when I go to start picking edges for the fillet-- let's kick that up a little bit, a little more obvious. There I can pick at, and that's going to automatically apply it. Or I can explicitly say what solids are going to be affected by if I do this right now.

And I just messed it up. Of course, they both got the fillet. All right, I'll do it again on the bottom side, just a little bit slower.

OK, so right now, the only solid that's coming into play is that one. If I'd wanted to I would have to go back and edit this fillet. See how fillet two is only showing up underneath the second

solid. Edit that fillet, and for fillets, I just have to pick another edge. For other tools, you might have to pick that solid button and add it to the selection group.

Continuing along the theme of why this is similar to an assembly, here I've given different parts. I can also go and turn on and off the display of individual pieces. I only have two here, so it's not real effective. But there's also a tool that is Show All or hide everything else but the one I'm on, kind of like the Isolate command, when you're working with an assembly. Great way of just focusing in on the one.

So here we go. A couple of different solids within a single part file and really the last thing I wanted to hit there on that is if I right click and go to its Properties, there's the appearance. I can say I want it to be bronze. All right, it actually looks OK on the projector too, as opposed to the top that's still back with its original color. Everything is still the same material.

Again, it's all in one part for you. You can't deviate on the materials yet. But I've got features that are affecting some solid bodies and not others. And I'm changing properties of some and not others, as well as visibility.

So to go from there and kick it out as an assembly, here's one I've already made where I-- very similar concept where I was. You can see the names are just as creative. It looks-- trying to figure out if that's a period in there. No, it's just a glitch on my screen. Yup, it look like exactly like a period right in the middle of the name, so was getting a little worried there.

This one I created, a little bit of a different technique. I made the second solid by using the Split command. I'll come back to that and-- oh no, I'll jump over that right now.

Here is a piece where I've run the Split command on. And what the Split command allows you to do is if you take this choice at the bottom, you're creating two solids from whatever single solid you're starting with. And here, I'm using a surface as my Split tool to break it out. So split is a way of separating existing pieces.

This is what I would typically do, say, I was designing that new mouse. I'd be constructing that overall shape, getting my rough shape, putting surfaces or planes where I know. I'm looking to later break into the separate parts.

I might be wrong. I might have to adjust things. Again, those aesthetics are pretty flexible early on.

But if I'm just moving that surface, that's fine. I'm still splitting the body using a surface. Life is good. Things still keep moving along smoothly. So that's why you'll see later on, I tend to keep these surfaces in there for a while.

All right, so going on and taking that solid part and kicking out to an assembly, over on the Managed tab, there are some tools in a panel called Layout. So one of them is used to make a part file from individual pieces. So I could take these solids right here and kick them out as separate parts. Excuse me.

AUDIENCE: I've got a question on that.

JIM SWAIN: Sure.

AUDIENCE: So you got a plane-- it was three planes there.

JIM SWAIN: I had a surface that was three.

AUDIENCE: Surface.

JIM SWAIN: Yeah.

AUDIENCE: So could you put a box in the middle of that, if you have a complex snap and you wanted to do a full [INAUDIBLE]?

JIM SWAIN: Yes.

AUDIENCE: It's not just there. There's a box in the middle.

JIM SWAIN: I'd have to see a sketch for what you're talking about. I think the answer is yes. What I'm using for that surface just have to be nice and flat. But I could use a ruled surface, a swept surface, a lofted surface. Loss can give you challenges other places. But I think from what you're describing, I would say yes.

Now, it might be that I do one split to make two separate pieces, then another one to get that boxed area out of there. That might be a reasonable way of going as well. If you want to come up afterwards and maybe sketch out what you're talking about, give it a try or at least look at it. How is that for a good deflection? That work? Yeah, come back to it.

So here what I'm going to do is that make component, which is going to take an individual part and take the solid bodies out of that part and create an assembly. By default, I once used the

exact same name. Since I probably have done two or three of these just to get through the practice, I'm going to give it another number on the end that I know I haven't used.

I can pick a template file. I can go ahead. And if I need to go to something a little bit more in depth to find my templates, I've got the button to get me there. Where am I going to put this?

And what I found, which is fairly annoying, I couldn't just go ahead and type in the name right here. It'd give me an error message. But if I went into the browser, yeah, I'm going to make a brand new target assembly. Otherwise it's going to push it into an existing assembly.

And I could do a new folder at this stage. And let's see, today's the third. So I'll just give it today's date so I know how to delete it later. I got tired of finding files labeled junk and folders labeled junk over the years, so I started just putting dates on them. Bill of material-- what type of bill of material conditions you want from the overall assembly.

Now I need to pick the solids. So I can either pick them on the screen, pick them in the browser, what have you. Get the different solids. They're going to be kicked out as new part files.

Next tab allows me to now go and give the name. Now here you can see the name of the part files matching up with what I've renamed those solids to. So that's another reason why I like going there in the beginning.

And did anybody else attend that Top 10 Inventor Apps session yesterday? That some of those tools were going and automatically renaming things and going back and hitting the iProperties. That would be awesome. Applied to this where there were some tools that are out on the app store that will go into a folder and push iProperties down to all the parts that you select in that folder that type of thing, so that here we're starting with very little information, but I can quickly fill that in later as we go.

Again, I've got the ability to give a bill of material setting at this point. File location, I can go ahead and browse down into that folder. And it probably would have been good if I had highlighted both of those when I had gone ahead and dived down. You can multiselect there at that stage and highlight more than one.

So I've got these pieces going down there as well. And I still have the mirror part scale factor. And the color, wow, color override.

So if I had applied the different color here, that color would be assigned to the new part. It's not affecting a material. It's only assigning a color.

Those parts in the new one, they're going to have whatever my material is here, and unfortunately, it's generic. I can include parameters from this. Again, this bottom half is a lot like a derived part because it essentially is doing derived parts.

So now it went out, did its thing. Notice I'm in the assembly file. And if I go and open an individual part, generic, it's got the name Top of Case. That's what came from the other piece.

At the assembly, those are fixed-- sorry, grounded. Keep using the wrong word there. If I ungrounded, I better use some assembly constraints to hold position.

Back at the part file, I did mention that these are built in place, so they're stuck there. That wasn't quite 100% accurate. There is a command under the Modify to allow you to move bodies, so you can type in and offset distance. Question?

AUDIENCE: Yes.

JIM SWAIN: Yes.

AUDIENCE: Because currently, I'm using the Vault and is it possible to take a part from another project and put it in a new project?

JIM SWAIN: When you're using the Vault, are using one project file for everything coming out from the Vault, or you're using multiple project files that are all Vault project files?

AUDIENCE: No, I'm using multiple project files.

JIM SWAIN: All right, the answer is yes. But I believe what you would need to do is reference those other project files. I'd have to think about the right phrase.

I'm not a Vault expert at our company. We had people that are. Unfortunately, none of them are in the room, at least those from our company.

So I don't know if anybody here has more that they would like to add to the answer after we're done with the class, if you don't mind. Otherwise I'll also try and think about it. If I have an answer for you after the class, I'll let you know. OK, but the short answer is yes. The derived part workflow does work within the Vault.

All right, so back to the PowerPoint. And we're-- oh, we're doing pretty good on time. [COUGHS] Sorry, that's not going to be good on the recording. Yeah, I know.

OK so from a design point of view, things that give challenges. Sufficient Draft. If you've done plastic parts injection-- well, injection molded parts of any kind-- then, you're already familiar with this. Draft being if the mold is going to open in this direction, it's going to open, in this case, vertically as the slide is.

Right here I'm going to have trouble because I've got a vertical wall of steel, that's the blue, and a vertical wall of plastic, that's the gray. And as you try to eject that off, they're going to be rubbing. You're not going to be able to have a good surface finish. Good's in the eye of the beholder.

You can't have any texture on the inside. You're going to get texture. It's just not going to be what you want because it's going to smear as it goes along through there.

So instead, you need to put some kind of taper on there. That way, as the mold opens, you're not doing this. You're immediately getting some separation.

And there's rules that the folks that will do, like the chemical etching for the different types of look, whether it's a fake wood grain or a leather or something like that. They'll tell you what they want to have for the draft angle before you give it a resin for their particular texture. So you need to have that.

The other thing that comes in with draft is not only how much force you're going to need to take to eject it because you've got to overcome that friction in there. But also, how long do you have to let this cool before you can push it off? If you could push it off a little bit earlier, again, you're going to slightly increase your part count for a given day, a little less cost per piece.

But those walls are a little bit softer, so when you go to push on them, they have a little more give to them. So you want to balance with that where they're cool enough to push off and get that as early as you can versus wow, we just deformed it because the ejector is pushing on the bottom of that wall. So you've got that balancing game going on. And I already mentioned the surface finish.

Also the overall shape-- now by this, we had some interesting conversations trying to convince customers that you can't have a perfectly square box unless you really wanted to have an

expensive mold because there was going to be some kind of taper. And we do everything we could to keep it as minimal as possible. But that straight push on push and them wanting some kind of texture, that combination was not going to work. So that was an ongoing discussion.

The other item that I'm going to hit from a plastic part design guideline, rule of thumb type of thing, is going to be trying to maintain a uniform wall thickness. So here, you can see a basic shelf for this box. And then, here's a shell where I've used a different thickness. If you expand the shell dialog box, you can change wall thicknesses for the different parts from it.

So here's a question for the audience. Which one of these has uniform wall throughout the entire part? And yes, it's a trick question. No. OK, and didn't have to shake your head. Now everybody knows who gave the wrong answer.

You actually don't. You have a uniform wall thickness here. You have a uniform wall thickness here. But where these two meet, that part there, that wall thickness is about 40% the size of what just the regular wall was.

Where you've got the three corners coming together, now you're about 70% of an increase over what those overall walls were. So you've got to keep that into account. I'll also mention it in passing. I'll come back and hit those.

Resin doesn't like going around sharp corners. It's going to give you a little bit of an eddy in there. Get a little bit of a swirl. And that starts affecting your aesthetics. So you've got to balance out going and giving a smooth flow for aesthetics versus what impact does that have on your wall thickness and start working things that way.

The increase in wall thickness, it can take the filling time. If you're trying to fill it, for some reason, from this end, and you've got a thick area far away from it, you're going to have to push that plastic all the way through. It's going to have to be staying hot till it gets there.

Boy, it'd be really nice if it can move where that gate was. You're filling in the thick area. Yeah, it'd be really nice. Can't always do that.

Cooling time. You now have a thicker area. That's going to take longer to cool. It's got that much more heat because there's that much more plastic there.

And the problem with that-- I just mentioned warpage with all kinds of things. Because the longer it takes the cool, the more it gets to suck in, so the more shrinkage you get in that area,

which can cause warpage. It can cause sink marks.

It can cause all kinds of things that makes QA a pain. And Murphy's law will say that that one area where you've got the really thick area is also going to be your really high gloss area that the customer wants to see, which is the wrong combination for hiding shrink, hiding sink marks, that kind of stuff.

So what do we have to work with? Well I already showed you the Extrude tab. I won't go further into that. It's got a taper on the backside.

Typically, unless I was doing something as simple as those boxes I had earlier, I don't normally mess with the taper on that because that tends to put your parting line right where the extrusion begins or right where the extrusion ends, depending on which way you taper it. And again, Murphy's law, I usually had to do some kind of step or something like that in my parting line because of other reasons. And if I do the taper, I'm kind of stuck with where the parting line was unless I really start mumbling under my breath about what an idiot I was when I started. So I tend to keep the taper at zero and later add draft, mostly because it's faster to put taper in the beginning, but I'm wasting time when I have to make changes later.

The Shell command, that seems to be the perfect thing for putting in the uniform wall, but as I said, it only gets you started on it. You're not going to have a uniform wall throughout your entire part unless you really got something simple. You get away with fillets and all the corners and so on and so on.

Fillets, in fact, there you can see up in that corner an even better example of now, I really don't have a uniform wall where that corner's now been filleted away. Is that a right word? I'm not sure.

But putting those fillets on the inside, the plastic's going to flow around a lot easier. I'm going to have much better aesthetics as a result of that because I'm not going to see a blush or not as likely see as blush or something like that on the outside wall. And a lot of the pieces I did, it was an enclosure around something. So I was always worried about my exterior appearance.

Split, already showed you that. I'm going to give you another look at it because I'm going to use Split to put some draft onto a piece.

Now, Inventor also has a series of plastic part features. I don't know if anybody has not seen these, but it's easy to miss them because they're not turned on by default. You fire up Inventor

and new installation, you don't see them. They're not visible.

What you have in there. Tool to make grills, such as speaker openings, fan openings, that kind of thing. Snap Fits, both to snap itself and the hook, if you want to. Canon lever being with special purpose. Bosses might be just a support but it typically something you're going to run a fastener into.

Ruled fillets-- and yes, I'm going in the order that the panel shows them. Ruled fillets where you'd set up a couple of rules that decide what given fillet radius is added at what part of your feature, what location of your feature.

Rests could be a intermediate staged height shelf where maybe you're having an opening for a USB connector or an opening for a power plug, something along those lines. That's typically where I ran into them.

And other than the Snap Fit, this is probably the one I would have loved the best to have back in the day because a Lip is a great way of if you've got a seam between one part and the other, if it's lined the line and one piece has some warp, you see that gap. But if you have a Lip and one piece is up a little bit and the other is down, who cares, you can't see into it.

Also, if we had to pass Underwriter Laboratory tests, you will. So it gave a longer path for SPARC, ESD protection. So now any SPARC had to go around that gap to get into the circuit board. So we used Lips all over the place.

Unfortunately, and here I'm showing why I've got gray hair, we're using good old AutoCAD release 13 back then to design the plastic parts. We were guaranteed a crash a morning, crash in the afternoon. And that was also trying to use Solids-- oh and Windows 3.1. I'm amazed I still have hair, let alone the fact that it's white.

The other thing Inventor has is over on the Inspect tab it's got a series of analysis tools. Now Zebra, surface, and curvature are really aimed at developing your surfaces. And that's not what this class is aimed at. There have been some good classes in this session. I didn't list them as far as Class A surfaces in Inventor, other things along those lines.

But Draft, as I'll show you, it'll give you a quick look at what draft angles you have in your part, real fast and easy to do. Section, what kind of cross-sectional thicknesses am I getting through that? So again, let's take a look.

Let's see, we don't need him anymore. Love that. OK, so I already showed you Extrude. Let's kick over to adding a Draft. Looks very similar to what I was running with earlier. I did a split. There's the extrusion. And that's going on.

So you've got a couple of different styles of Draft. And yes, I can be lazy and just pull the part marker down and show you the prefab ones. We'll do it live.

So the basic Draft, in fact, all the Draft commands, you're going to have to give it a pole direction. But the basic Draft command, what you're saying is pole direction here. One or more faces are going to be affected by it, and you're going to force an angle.

Typically, the problem with the basic Draft is you're going to get these lips popping up, these overshoots, depending on where you pick the draft from you might get an undershoot. So that can be a challenge for what you're trying to do.

You can also-- I don't really have a plane set up on here, but there's a fixed plane. So if you had a work plane, that was going to be your parting line, a nice straight parting line. You can draft both directions away from that work plane. Doesn't have to be parallel to x or y or anything like that. It's just a plane.

Relatively new for the Draft command is this-- got to wait for the tooltip to come up because I don't remember the exact name for. It is in the handout. There we go. Parting Line Draft, where I can set it up that I am going to fix the parting line-- well, it's actually default. It's going to automatically blend between the faces. It's going to allow me to chain the faces across tangents, which I don't really have anyway.

So here, again, it wants a pole direction. Pole direction, how is your mold opening in the tooling? All right, parting tool, it's got to be a surface, as far as I can tell here, surface or a work plane, not a sketch. Another reason to leave that surface sitting there.

What faces are going to be affected? Let's go with those. Let's take the one that goes both directions from it. Here we go.

All right, Swain, oh well. I've get that one going. Set it a little bit wrong.

And here you can see that I'm not getting any deviation off of the mating to the other solid body. It's keeping it nice and smooth. I am getting a draft across there.

It's good and even. It's going to definitely work for my purposes. I don't really need 10 degrees but giving me something that I can work with here. If I had had fillets connecting around the sides, I would have that -- the face chaining would have taken it around.

Now having said that about fillets, I keep fillets off for quite a while unless the fillet is running along an edge that is going to be in the same direction as the pole direction. And the reason is this-- I'm going to cancel that for a second. I'm going to go and hide the one solid body. And we'll hide that one.

And now let's go ahead and hide the surface too so you can see what's going on there. If I run the Shell command now-- So I'm giving this a tenth of an inch, just the default material thickness, putting that in there. I've got the basic hard corners.

If I go and put a fillet in that affects that bottom edge-- and typically, it's going to be more along the lines of taking every edge that's in there. It's very common for that to now, unfortunately, have locked the draft on that notice that all those faces get selected with one click. I've had very little success trying to add Draft once I've already got that fillet in there and having that tie into the rest of the piece. So I wait on fillets as long as I can. It's one of the last things that I add to a plastic part.

All right, so that was fillets, chamfers, the extrude where-- I didn't do chamfers the same as fillets. We hit the Extrude earlier. The split to separate it whether it's splitting it for making separate pieces, but now splitting it for doing the draft.

For the other features, I'm going to go and use the classic cooking show version. I've got this ready to go. I'm just going to pull the end apart marker down and talk through it.

So this is derive from another piece. And I apologize, I meant to mention earlier, when you have a derived set-up, you can go in either suppress or full out break the link to the original part. So it starts off as a child. It doesn't have to stay there, if you want it to be fully independent.

All right, so the bosses first. What I did was I put a sketch. And on that sketch, I put a couple of points, literally center points. They're showing up OK on the overhead. They don't show up real well on my screen. So one there, one down here.

Also, this plane is at a specific level. In this case, you can see it's at the top of the part right there. The height of the sketch plane and those two points are going to be what set up where

the boss is going to go. Because the boss itself starts at your work plane and goes to the rest of the plastic part.

For the boss, you can tell it whether it's something that's going to enclose the head of the screw, whatever fastener. I'll just use the word screw for that. Or is it something that's going to be threaded into? In this case, I went with the threaded into.

It's going to pick up the points from a sketch. And telling it which direction it's going to go, it picked up the centers automatically. And I can give it an offset from the work plane if I want to.

I usually just let the sketch plane do it that way. I'm avoiding confusion. Now why is this a half a millimeter above everything else? The sketch plane's there. What's going on?

I don't like throwing extra things on there if I can avoid it because I'd just forget about why it was there in a year and then confuses me if I have to change things. I added a very nominal fillet at the base of the boss. You'll see it when I finish the command. It's going to come right back in there.

For the thread, the next tab on the dialog box goes into what's the overall diameter of this at the top? What's going to be the opening for the fastener as it goes in? What's the draft on the inside of this? Because again, that's going to have a pin in it in the mold.

So it's going to have to be able to release. You can't get away typically with a zero draft. You might go small as you can, but you can't get away with zero.

You can also add ribs to it. If I add ribs, how many do I want on this? Two? Let's go with six just to be stupid and so on and then work it through. Now this actually won't work here because the ribs that are going to interfere with the existing walls don't get created. It's only ones that have a clear shot on the landing surface that get made.

You can set up the geometry. What's the width going to be of the rib? Any fillets, any tapers and so on? Here, you can see that 10 degree taper on it. We can all get rid of those before life gets too ugly.

So that's a tool right here for creating the boss. And I didn't mention it, so I better mention it now. If when you fire up Inventor you don't see this plastic parts panel, right click in a blank area of your rib and show panels by default it's not turned on. You've got to put that check mark there.

All right, next grill, that came from a little more complicated sketch. Series-- it's actually on the other side of the piece. Here we go. So series, in this case, concentric circles. This is going to be that internal island. This was a ring around that, and this was the outer extents of the grill opening.

And then I drew some cross pieces because you kind of have to hold that in place with something. So you need to put at least one part or one rib in there for it to go through. I think stringer is actually the word that they use.

So when I go and edit the grill, I can set up how big is the grill? Does it come above the surface at all? In this case, no, it doesn't it. It stays exactly flush. That's because of that value.

How deep does it go? I did have this one go deep into the part a little bit. You might need that because it's holding a speaker or things along those lines.

How wide is that rib worth of the grill? All I had was a circle drawing. These settings here are setting up what the outline for the island's going to be, what the outer ring is going to be, what any in-between ribs are going to be, and the spars that connect the rings in the-- I'm sorry, the ribs and the island and the boundary. All that's being controlled by these settings here. And then, finally, you can also put a draft on it so that you don't have to come back later with the Face Draft command because this would be tedious.

The other end of this is that's a really good reason for a rule fillet that come in and automatically fill in everything that's going on here. And what the rule fillet looks like is you set up a rule. Your choices are as this feature attaches itself to the rest of the part, or as this attaches to a given other feature, any free edges off of this feature, and all edges. And you can set up a condition for, maybe, as it goes against the part but come and add another condition for where it's just a free open edge, not touching anything. And that would really go bad real quick but give that something more like a reasonable value and so on.

So you can keep adding rules. And this one, I think I just used all edges and was lazy about it. Cancel out of there.

And you can see, there's the result with the fillet on that end. Oh no, it was only against as it met the rest of the parts. Sorry, it's the only fillet I did there and free edge there. So that's the rule fillet because again, you'd be picking a lot of edges if you tried to do this with manual fillets

otherwise.

The rest-- actually I cheated on the rest because I had to then come back and clear out that little vestige that was left on it. For the rest, all I did was I created a sketch that was again, on a work plane, up at some height. And then the software added the walls to connect it to the remainder of the part.

I gave it a thickness. I said [INAUDIBLE] 50 and that whether it was going towards the inside or the outside based on my sketch. Like I said, I ended up with just a little bit of a vestige there, so I did some-- I'm not a fan of cleaning up little things like that, but sometimes, it's a necessary evil.

The lip-- no sketch required to make a lip. Another thing I love about the feature, not only the job it does, but how easy it is to put on. You just need to have an edge of the part. This is nice and flat, could be a three dimensional edge going around.

But the three dimensional-- oh sorry, the lip itself. You tell it whether you're making the groove. Or are you putting a lip on so lips protruding off of the existing surface, a groove would be cutting in.

Tell it what the edges are going to be. So I just picked the outer edge of this piece. If this was a three dimensional path, I would also go and grab this face here and use that to keep the twisting to match what that face is doing as it goes around. Again, this is a very simple case, so I could get away without giving it a guide face. All I had to do was pick the outer edges.

The other tab, then, is where I tell it what they are going to be the dimensions of that cross-section as it sweeps around. And very simple from this diagram to quickly and easily build that trapezoidal shape.

All right, I'm getting close to the end here. I'm going to have to kick it up a notch because I'm talking too long and that, unfortunately, a problem of mine. So let me just go down here to the Snap Fit.

Snap Fit had a sketch, just had a point on it. From there, the tool itself also took care of all of the dimensions for the Cantilevers. And this is where those online design guides, the manufacturer design guides will have tools. It'll also give you advice on what you should use for these values so you don't overstress the base of the Cantilever, beam or overstress where the hook tip is.

And since I am doing a hook instead of a loop, I can go into the detail and give it the ramp angle. So maybe it's a snap that's allowed to come apart as opposed to one that's got a 90 degree, sorry, I don't care how far you pull on it. Till you destroy that Cantilever beam, you're not unsnapping them until it hits the floor, and then it flies over place. But anyway.

So a real quick look at those again. The handout goes further. I wanted to spend a few minutes talking about the analysis. And I didn't leave myself as much time as I want, so I'll be fairly quick.

The Zebra-- if you've never seen one, they look really weird. You're applying a black and white series of bands on your piece. And what you're looking for is where do these not blend smoothly from one to another?

These are faces that are joined by a simple fillet. So it's tangent. There's no other continuity, no other-- it's not a G2 or anything like that.

And you can see, you get breaks in the way those are. It's like going to look at a used car on a sunny day. Hopefully, there's a lamp post or a picket fence nearby that you can take a look at what the reflections of the body is to see if there is body work done.

Curvature, same kind of thing here, I'm only looking at the curvature of this face. But you get these whiskers to show you what the relative curvature is. The longer the whisker, the faster that curve is changing.

Another route to go instead would be the surface. Again, the brighter the color, the faster the surface is changing. Yeah, I'm right here. I'm looking at what's called the max curvature. So red is that middle ground, green, wow, it's really changing curvature quickly in that area.

Where I really take advantage of these when I'm designing parts is I'll come back and take a look at the draft. And if I look at the feature itself, I said, show me anything between three degrees and minus 3 degrees. Wow, you get to the reds, you're either straight up or you're into a negative where you can't pull it apart with a simple mold. You're going to have to have a fancier mold with the pieces pulling out of the way. I'll do the same thing with Moldflow too.

Now cross-section's probably the most useful of these. So cross-section, you got two choices on it. One of them, if I make a new one, gut simple. I'm going to pick a plane. I can use the xy plane in this case.

Select the plane. There we go, xy. And what it'll do is just slice right through it. It's like having a sketch and turning on slice plane with a sketch.

It's a quick look. There is no sketch created. It just gives you a good look at what's going on as you ran a bandsaw through your part.

I'll delete that one. This other one, what I did was I had a series of planes. And each of those planes, then-- I used, let's see, the xy again, and I had a couple other work planes that I created.

You can go and set up conditions. Hey, I want to know whenever my wall thickness is greater than 60,000 or less than 40. I've got no 50 nominal. What's 20% over? What's 20% under? That's where those numbers are coming from.

And I'm going to select, let's say, the xy plane. I could select others as well. I'll just do the one for now. And it's going to say, whoa, you violated the max material condition. Because right in here, I drew a circle tangent to that tangent to that tangent to that, it would probably be about of, I don't know, 078, 080, 085, somewhere in that area, would be that immediate location's wall thickness. We're going to get sinked there. We're going to get a high stress area as that part cools.

So you set up what are going to be your boundary conditions. You can have it show values for every plane you pick, only show the ones that violate the rules you set up. And that would be the way I would run it.

AUDIENCE: Can you do it without cross-sections, the part?

JIM SWAIN: No, it needs to have a cross-section to investigate because I think under the covers it's creating a sketch and projecting it. That's what it feels like as far as how it behaves. Now, I'll use that as a segue, thank you. By the way, I have some tchotchkes. You just earned, whatever you want from that for that segue, thank you.

Moldflow design. Now, Moldflow design automates a lot of the stuff, makes it real-time checking from some of these. Again, this is not included in Product Design Suite. So Moldflow design has already been running on these. I'm going to go back over to-- that's not the one I wanted. Here we go.

So here is a part that I've added some features to, very similar to what you've seen. I just added that boss in the center. This widget, and that's the technical phrase from Autodesk, this widget here is Moldflow design. Saying I've got a pretty good part right now, manufacture abilities at 82%, cost efficiency, 93%, plastic material impact on the environment, 95-- I got a heck of a part going on here.

But I've got that little warning right here. If I click on that, wow, we've got some areas that are not drafted at all. I've got some areas that maybe I want to check and see where those weld lines are going to be, places where as the flow comes into the piece to where it's going to meet. I've got some knife edges, just a couple, but I got us some knife edge areas in this. That's going to be hard to push plastic into.

By the way, what is my variation in the wall thickness? Wow, the wall thickness is really bad right through here. It's not so hot through this area. You know I'd love it if everything was green.

But I'm only going from 0.12. And I can't read what the bottom value is with this color scheme. But it's not too bad. I think my minimum was 0.040. Or no, it's probably 0.030.

So Moldflow design allows you to take a look at different functions within your design. Have it monitor everything for you while you're going on. If I made a change to this piece, it would update immediately. You see a little wiper motion going on, if I got rid of that extrusion right there. See how it's recalculating.

And now it's come up with a little lower manufacture ability level. And I can go through all of-- now, I've got a high wall thickness variation. Not sure I agree with it.

Other things I can take a look at that I really appreciate is I can go and turn on the Injection Location toolbar. I've got a cone here. That's my injection location.

I never had to tell it of the gating properties. It's just a fictitious gate. Move it wherever I want, put additional gates on, delete other gates if I want to. But once that's been put on, I can right click again and go to the Animation toolbar. And that's the expected flow pattern for this part with that key.

And this is a great thing to do when you're going to go get a fresh cup of coffee. Just let this loop while you're away from your desk for a few minutes. Oh guys, I was letting it think for a little bit. OK, now, I'm going back to the design now. I shouldn't give away all the good stuff,

but it gives you an idea of what's going on. And you can see why it was saying that there was going to be a weld line here because that flow front separated and is going to be coming back together, definition of a weld line.

The Moldflow itself is set up by this configure rules. Position will put the widget at a different spot on the screen. I'll go in the configure rules in just a moment. You can send stuff out. I don't have Moldflow advisor installed at the moment.

You can also go with what they call a finished part view. And before I do that, I need to look at this from the other side. Remember there's a screw boss right here. So I'm going to come in here and show the finished part view.

And what it does is take a look at the current orientation of the piece, runs the filling analysis, takes a look at the variations in the wall thickness, and gives you an estimate of how it's going to look. And you can see an indication-- doesn't show up real clear on the overhead, sorry-- that I've got a depression forming there. That's likely to be in the area of sink. So it's giving me a good guess on things that are going to go wrong. That's what that finished part will do for you.

So to configure this, it's very simple. The help from Moldflow design also goes further through this. If I wanted to configure the wall thickness variations, I get a score of 100. If anything's between 80% and 120% of the nominal, but if I have areas that start dipping below that or going above it, the score for those areas are affected, and then the average scores obviously are going to be affected from there.

So I like this running continuously. Yes, it does impact my machine because it's running this continuously. But I like the feedback of wow, shouldn't have done that, because that's going to have an effect on my design that I don't appreciate.

Once I assign a specific plastic to it, then I will get some more realistic numbers for things like the environmental impact and the manufacture building. For the generics, those numbers aren't so hot. OK getting down to the homestretch.

Showcase. Here, I've already imported something in the Showcase. Showcase, I used to show quick and easy, look at something before it exists, even in a rapid prototype. This is just imported from Inventor. So I brought over a file. You can bring in more than one file, move them around once they're on the screen, set them up.

When you do take a look at first pass at what these units are, every now and then I've had to come in and be wrong because it's given at a scale of centimeters instead of inches or vice versa. So take a look at that when you bring things in.

Things you can do here. If you've got positional representations in Inventor, you can show different positions. You can show a transit, essentially an animation from one to the other, or just a snapshot from one to the other. What I tend to do with my stuff is I'm usually worried about taking a look at the materials. So I can take a look at materials down below or just go and say I want to bring up the material library.

And right now, everything is this dull white. If I select my entire-- it already was selected. That's what those blue lines are. Let's see what that looks like, if we give it a black. And that's really hideous on my screen. Yup, even worse on yours.

AUDIENCE: [INAUDIBLE]?

JIM SWAIN: Yeah.

AUDIENCE: [INAUDIBLE]?

JIM SWAIN: Yeah that's-- yeah, I appreciate you saying that because that was one of those things where I know I'm going to not spend a lot of time on it here. Because what I wanted to explain why I'm using it. So thank you.

Loren does some very good stuff. Rob Cohee does a lot of talk about that I appreciate from what the-- all right, did I kill that, fine-- workflows and stuff as well. I would take a look at both of those. So I--

AUDIENCE: [INAUDIBLE].

JIM SWAIN: Yeah. And what I'm showing here and what I'm talking about here is what you can do in the version that comes directly with Product Design Suite. There is a professional version where you can start taking these things and adding them to your web pages, setting up turntable rotations to try and sell your product and that kind of thing.

This is a little bit of a clearer view where I've gone and said, look, that's smooth. Is that the effect you want or did you want this rippled effect? And you can adjust, make your own materials, and so on within the Showcase to-- instead of a black, I want that as a bright

orange. So it's as ugly as these. So that type of thing, you do have that capability.

3D Print, let me put this to the audience. Do you want me to fire up 3D print and see it, or take the last 10 minutes for question and answer and discussion? How many would like to see 3D print for about two minutes? OK, we have enough people to do it. And I keep putting the wrong button coming out of it.

OK so under environment with 2016, we now have 3D print. 3D print-- come on you, thank you. I talk to the people in front of me on the freeway too. That's just life.

So there's the part. I can change its orientation and position within this work environment. That work environment-- unfortunately, this violates these, start at the left, work your way to the right that goes along with the ribbon. The work environment I would probably set first.

Here is the default 3D printer machines that you could pick and choose whatever you happen to own or whatever you know your service bureau has that you want to go with for maybe your given material or the size of your piece. If you go into the-- oops, wrong button. You go into the other printers, you'll see the full list. And you can even specify your own here. So things like the Stratasys, Fortus 360, hey, here's the working area for it, the volume that it can deal with, and I can set it as my default printer.

Print options. How smooth of an approximation of your surface do you want the STL file to be? Because prints are faceted. It's not a machine smooth piece. It's not a molded smooth piece. It's going to be facets, so how much surface deviation are you allowed to go?

Are you going to go with a fairly low resolution? So it's going to be allowed to move pretty good to a high. That's an order of magnitude less deviation being allowed right there. And then, you can also customize as you want to.

Other settings that are coming into how big are the facets that the STL file are going to create. As I said, I can set my orientation. Let's see, so say that that one's going to line up with the printer wall and that flipped it over because picking that face that's now on the bottom set the position. I can move it within the- I'm going to kill Showcase here, I think that's killing my speed, there we go-- its position within the piece.

Partition, don't have time to show it, and it's not really a good part. But Partition-- you've got a work area that's this big, and I'm trying to 3D print my laptop. I can cut the piece, partition it,

and also add locating lugs and openings for those lugs to fit into and have two parts be printed, ready to be reassembled, poxy, whatever, at the far end of things.

I like the picture showing, I would call that a cam shaft of some kind. Does a pretty good job of showing you what's going on there. And again, there's a session coming up at 1 o'clock.

When it's all said and done-- at this point, I am only doing one of these. I could kick it out to an STL file, but Print Studio allows you to go further deeper. Duplicate, put a bunch of these down. It's like a preprocesser, preprocessor-- there we go-- for your printer. So Inventor gives you a start, gets you the basics of how you're going to orient, but then to finish it off, use the 3D Print Studio.

And here, unfortunately, it keeps going back because on my machine I've got the Autodesk MBR as my default. And this piece is just a little bit big to really show it. Because you can do things like, duplicate this, have four of them laying next to each other, change the orientation of it, kick it up at an angle, see what the supporting structure it's going to be built for that particular machine that it's going out to. Again, it's a preprocessor to go out to a specific machine. Now, since I keep killing this, the PDF-- there we go.

So here to me is really the wrap-up for it. I do have a couple more slides, but this is it. Starting at the top, so you heard me say I would develop the rough shape, say I'm doing this mouse again, using multibody solids. [INAUDIBLE] part file, split it, another split for the battery cover, another split on the battery cover for the slider for covering the optics, so on and so on.

But rough things out. Get the basic shape. Architects would call that the massing study.

Then, go on and develop my surfaces to do the splits. So, OK, let me phrase it differently. Get my basic mass shape down, then start adding the surfaces that are going to split into the separate bodies. I got ahead of myself.

Either shell it at that point, or draft it. And when I say it depends, it really-- what's the end result? Do I need to have something as close to vertical? Am I really worried about having enough draft that I want to kick it out early and allow for it?

I've gone back and said, no, I need to change, and I need to do my draft earlier now and then did the shell afterwards. It depends on the individual design, really what works best. Typically, I'll start with the shell first, then add the draft.

Develop the details, final aesthetics. Start adding those internal fillets for the molding. Start adding the different outside pieces to give it just that right look that we're looking for or the strength or filling in the snaps and the relief that's going to take the snap, the bosses. All that detail gets added.

And I will keep that at the individual part as long as I practically can, again, one-man design group. If I was working with the design team, I would take that solid body, split them out, and have you do the battery. You do the top. You do the bottom. And you guys do all the internal detailing at that stage.

Moldflow design. Did I screw up the draft? Where does it look like it's going to fill? How are things going? I'm going to keep that in my head from day one. That way, it's not something in the late end of the game where I have to now come back and redo things because my part is not going to be economic or even manufacturable.

Showcase. All right, Mr Chief Engineer, here's what I think it's going to look like. Are we ready to go to the owner? No. OK, fine. Do some more stuff, give him another alternative, or maybe have three alternatives ready to go right then and there. And then go to the owner. Yup, the crickets are telling me the class is over.

Then, the owner always want to hold it. That's where the 3D printing comes into play. So if I was in my old job today, this would be the workflow I'd go.

So I'm going to ask you for another minute or two of your time. Does anybody have any recommendations for the group of alternatives to consider for a workflow? Because that's what this class is really about.

AUDIENCE: Instead of Showcase just use Studio.

JIM SWAIN: That's certainly an option. You have both in the product. What I like about Showcase is I can show alternatives very quickly and easy. Studio, you're pre-building what you want there. If the boss said, no, I want to see a slightly darker gray, I can do that probably in about 10 seconds in Studio-- oh sorry, in Showcase. Whereas in Studio I have to go back and re-render it.

But I do like Studio, especially for showing motions and things like that. It's awesome tool for that and other things. I get in trouble for saying that, but I like Studio. Other suggestions for the group?

OK, I've got two of these batteries, the phone charger things. So I'm going to make this really simple because I'm getting hungry too. What was one of the general design guidelines that I mentioned earlier that I always keep in my head, I'm trying to keep that as an ideal? Anybody? Was its own separate little picture. I was pointed--

AUDIENCE: [INAUDIBLE].

JIM SWAIN: No. It wasn't, but that's close. Going once, going twice. I was thinking uniform wall. And what was the other one? What was the other one that was on that same slide? You're already getting one.

AUDIENCE: I got one.

JIM SWAIN: Oh OK. All right, so let's do it differently. Who would like one? I see one hand there. I see one hand there. Sorry. You ready?

AUDIENCE: Sure.

JIM SWAIN: Up. Hey, look, drop test. OK, the other thing is I got a few of these, plugs into USB. It's got a for a current generation iPhone. I have no idea what the other two do. Actually, I think this one can be used to charge up those phone batteries. Yeah, I know it's an ugly orange, but it has our logo on it.

AUDIENCE: [INAUDIBLE].

JIM SWAIN: It doesn't have the old connector for an iPad. You probably know why I know this. Anybody would like one of these? No, OK. I'm going to stop throwing these. I'll actually walk out.

OK, so I have to by contract-- slight lie-- show the last couple of slides. And if anybody wants to see what I meant for Moldflow inside of Inventor Professional, I'd be glad to show you. But if anybody would like, I think these are two gigabyte thumb drives. Again, you want one? OK, you're welcome to it.

Any other questions while I'm-- there's one. Anybody else? I don't want to carry them home.

By the way, go on to the show floor during lunch because everybody's trying-- I'm not carrying this back in my suitcase. He had all kinds of good stuff. I've got two left if you want them.

Please fill out the surveys. Please lie and tell everybody that this was fantastic and perfect.

And tell them why because he gave me good stuff at the end. No, but they judge things based on the number rating and also what the comments are. So please be fair but also be honest.

All right, if you have any questions, I also got some business cards and so when you hit that. Otherwise, I'll put the last one here and see what questions everybody has. Thank you. Enjoy the rest of AU. I know it's not--