

TIFFANY

All right, so you are here for 60 Tips in 60 Minutes. So it is going to be a very fast class. So

BACHMEIER:

keep with me. We're going to be going through a ton of things. If I don't have time for questions at the end-- I've got a slide up there for it-- but I could meet you guys either, if no one's pounding on the doors to come in here, we can just hang out and chat and I can answer questions.

Or I can meet you in the lounge that's just outside of this room. And we can hang out there and ask-- you guys can talk to me about anything you want to. We can talk about electrical. I've got an hour between this one and the next electrical class that hopefully you're all coming to. So we've got time, we can do that.

All right, so you know what this class is about, you already read it. That's why you signed up for it. So before I get into the objectives, let me just tell you a little bit more about me.

Welcome. Come on in. Take a quick reference guide if you want to. There on the table right there. And I have some candy.

My name is Tiffany Bachmeier. If you've taken any of my classes before or if you took them last year, you knew I had a name change. So if they were-- I was previously Tiffany Tucker-- so if you've looked up other AU online classes, I do have some other electrical ones under that name.

And no, I did not eat too much at lunch today. I am pregnant. So as they're balancing this lovely microphone off of it, which is fun. My child's probably like, what is it happening right now.

I've been doing AutoCAD Electrical since it became a part of the Autodesk family, so back in 2004. AutoCAD since R10. So a long, long time ago. I've been in the controls design world doing conveyor design before I actually became a consultant. And I've been consulting for the last 10 years on various products.

But my main products that I focus and love on are AutoCAD and AutoCAD Electrical. So, like I said, if you guys have any questions afterwards, you want to talk about this, or moving forward with it. Or if your-- I actually want to see, in a second here, how many of you are even using it yet. But we can talk about all those things after this class.

And that quick reference guide, just so you guys know, we have tons of file paths, all kinds of

different options that come with Electrical. And a lot of key things that go with building blocks and having different attributes that we have, and all the different naming conventions. So I really wanted an easy access guide for you guys to, kind of, take back, have on your desk, pin it up somewhere. And you can easily flip to those different options and things on it. So that's what that is for.

All right, so on that note, how many of you are using AutoCAD Electrical today? OK. The Majority of the class. There's some of you. Those of you who aren't, are you using AutoCAD? OK, most of you.

All right, so you're just trying to absorb as much as you can about electrical. All right, cool.

So there are-- this kind of ranges from beginner, intermediate, and advanced tips. Those of you using AutoCAD, it's probably going to be a bit fast. Just to fair-- give you some fair warning there. But you'll pick up on a lot of the things that you'll see it's capable of doing.

So that'll be a good thing to see. Obviously some of them are probably going to be, you didn't know what that particular tool was before you even knew what the extra thing is that I'm giving you a tip on. But at least you can now see some of the different options and tools that we have in it.

So hopefully you guys will get great things out of it too. Most of this is geared, though, towards people who are already using it. So you can pick up extra tools, options, tips, things like that.

All right, I've kind of broken them out into project management, drawing creation, customization, and template tips. They could, kind of- some of them could fall into different categories too. But that's the general gist of how we're going to be heading through this.

And I will provide-- so if you guys went to the site to download things, I have a separate hand out that actually is about-- so those of you who said that you are not using it yet, it's all about getting started with electrical. It's about setting up templates. And getting going on projects and understanding that. Download that. It's a great tool.

I'm going to upload this presentation as soon as we're done right now. And then you guys will all have access to all 60. I wanted to keep them secret until today so that you didn't skip my class and not come to it. So that'll be uploaded and you'll have access to that.

And you'll see, as I get into this, I'm probably going to spend most of the time in the

PowerPoint. Just because I have everything imaged out, so we can talk about all those tools and tips. Some things I'll switch back to the software. But doing this last year, trying to go PowerPoint to software, PowerPoint to software, it just got a little crazy. And I want to be able to focus more on actually telling you about the tips.

So all right. So with that said, starting right out of the gate. Some of these things too. How many of you are using 2017? That said they're using Electrical. Good, that's good. It's a decent amount.

So some of these tips are going to be, specifically, about 2017. So I have that in parentheses, if it's a specific, new 2017 tool. So that you'll see that when you go through this. So some of these are only going to apply to 2017. Everything else is just tips and tricks for everybody. OK? All right.

So the first one, there is a new project node that's in our project manager. That you can click on if you actually click on the Location tab of it. So we got that a couple releases back, the location tabs, to go with the whole Inventor, EMX to AutoCAD Electrical workflow.

And that actually allows you to see a full project level detail and connections area of all the components, all their connections, all the information in the drawing. So it's a pretty neat extra, little tool there that we have. And there's also been some security updates with 2017.

So a lot of our libraries and other tools have been always located in the public folder. If you were to just download straight to your C-drive-- those of you that are in a network install, this obviously doesn't apply. But the C-drive ones, this has now been moved to the main support folder in a shared directory instead. So that you don't have to worry about the security issues of the public folder. So, just so you know, there's a new edition there.

All right, so the next, kind of, project management thing I want to talk about. And this is not-- those two we're 17, but this is now for everybody. Project utilities, this is kind of a hidden gem on the Project tab of the ribbon in AutoCAD Electrical.

And I always talk to my customers and students, when I'm teaching classes, that this is a great tool for actually doing as belts. So one of the great things about AutoCAD Electrical is it automatically updates every sheet, right? Rewire numbers, component tags, all that stuff to make sure it's following your standards that you've put in.

But if you've built it and you need to go back and add other things to it. And it's already on the shop floor, labeled, everything's done. Then this is a great utility. Because in here, you can come in and set wire numbers, component tags, item numbers, all that stuff to fixed.

Who knows what fixed means? There is one person. OK, so when you set something is fixed in Electrical, it actually means that it's not going to be updated or modified during any of those project wide refreshes that we do.

So it will hold its value. So if I have something that's, let's say, a control relay and it's CR403. And then I have CR403A. If I wanted to put something in between that and that A becoming a B, that's what the software is designed to do, right? It'll update, it'll refresh all of that.

But if it's already built, I don't want it to change those tags. So that's one of the great things you can do in here. You can also change all of your attributes sizes, font styles, you can even run a purge on just blocks.

If any of you guys know, we love purge in electrical. But not purging everything. You don't want to be purging layers and all of that. Because you might need those later.

So this is a great utility for just purging all that stuff out of your whole project. Or if any of you are into routines, you can actually run a routine, a little script file that you can run on your whole project, through here. So this is an awesome utility box for doing all kinds of different things to your project.

Of course my next one over here, number four, is a fantastic tool. It came out a couple releases ago where, when we published the whole project. And if you guys haven't seen that, our publish tools are awesome, in general, from even vanilla AutoCAD publishing.

But we've included the ability to do parent-child hyperlinks. So when you actually publish that project out, that project, then, you can click on it right in a PDF. And it'll auto link between parent and child relationships.

You can surf through everything, just like you are in the software. But you can be giving it to people who don't have electrical and they can easily see all of those connections right in a PDF. Which is awesome.

All right, folders. Who uses folders? And the project manager? A few of you, good.

So this was an add on a couple of releases ago, as well, that totally changed how we can filter and go through our projects. So we can actually easily add these project level folders inside of the project. And we can drag and drop drawings into each one of them. And you can actually run any of our project wide commands on a specific folder.

So it's a new filtering system, but it's also just an awesome organization system. So it's not Windows folders, it's not going to change your hierarchy. Your project and your drawings are all together still. It's just inside the project so that Electrical can handle it and manage it by that. So you could run, even a bill of material or any kind of, you know, wireframe 2 report on just those specific folders, as well.

All right, Surfer. I have a couple different Surfer options, here. Who has used the Surfer? Hopefully lots of you, the Surfers the coolest tool ever.

So it's what connects all of our drawings, right? It's one of the great ways that we can really easily, quickly see that we have all of our components connected to one another. Well if you actually go to the Surfer, itself, and you just hit Enter. You can trace-- that's with this dialog box-- on anything.

So a lot of people know they can right click and go to Surfer and see what's connected on that particular component. But if you hit Enter at your command prompt instead of right clicking on a component, you can search on many different component tags, catalog numbers, wire numbers, item numbers, throughout the whole project. And you can even use wild cards. So you can search for finding different things that way. So it's a great tool.

Some of the other ones that I think are, kind of, a fun little addition. You could-- and I've done this with a couple of customers who are not necessarily using it traditionally. And they can actually fill in the normally open, normally closed, attributes that we have inside of components.

For a component that wouldn't need that, but you can see that then in the Surfer dialog box. So you can see more information. So you could use it to filter more information that you want on your actual drawings.

And you can hold Shift down-- I don't know if you guys know this-- you can hold Shift down and actually have-- as you run the Surfer-- and have it look more like MDI mode. So if you've ever noticed when you surf, it closes the drawing you're on, opens the next drawing as you surf

through that dialog box. Well if you hold Shift down, it'll leave those drawings open.

So it will actually work more like your MDI mode. If I have time I'll jump back into there and I'll show you those Surfer tools.

Who's used the Drawing List Report? A Couple of you. So I get this question a lot. Is there a way for me to automate my index sheets or that table of contents page on your drawing set?

It's not in our standard report tools in Electrical. It's actually just a simple right click on the project. So that's why this is such an important one that I bring up. And just right mouse button on that project and go to Drawing List Report. Because that's going to actually provide that full index sheet, fully customizable, just like all of our other reports. But it's just in a different spot. So a lot of people miss it.

Drawing-wide location tools, so that is this guy right here. And if you guys have all-- do we know what the drawing and the project properties are? Did you guys go to Todd's class yesterday? He talked about them.

All right, so on our drawing properties, you can actually set-- and while this is designed for many of our IEC customers, it's a great tool that will auto place that location code right into all of your components. If you don't override it with something else. So you don't have to be manually doing that every time you go into the Insert and Edit component dialog box. So that's a nice tool too.

And then cut sheets. How many of you guys do cut sheets? When you end up-- how many of you have extra blank sheets throughout your projects just in case you have more things you need to add? I'm seeing some nods. No one's admitting it, but OK.

So if you don't want to have to do that anymore, if you don't want to have a bunch of blank sheets in the middle of your projects, one of the great things about electrical-- and this utility is fantastic, anyway. Most of you, I would think, who are using it, already know about this upper portion here. Of just automatically going through and having it re-tag, rewire number, update all those source and destination arrows. Hopefully, yeah.

But the other parts of it and what could be sometimes scary because some people have run this without pressing the buttons and altering some things. If you've ever checked marked everything, you've realize that it's probably re sequence to all your ladder's to just 12345678. Anybody done that? Anybody want admit it?

It happens because, a lot of times, it'll just go right to the re sequence and the setup will just be to one. But if you actually look at what this is designed for, these bump ups-- let's say I need to throw a sheet in between sheets four and five. If all my ladders start with 400 and then I'm going to 500, I need sheet five and up to bump up by 100, right?

And I can do the same thing by the bump up on the sheet number, as well. And then I can run this on just five through whatever I have left. And then I can add my new sheet in. And it's nice and easy. So you don't have to have all those blank sheets just in case you end up needing it. So it's a really neat utility for helping with that as well.

All right, any of you who have ever taken a class from me-- and I get to leave this in probably forever and ever and ever, because it's important-- do not explode, right? Or burst. I always get the gut person that's like, well, if I can't explode, can I burst? No, don't do it.

Because if you do that, you destroy the intelligence, right, of our blocks. That's what happens. Everything goes out to it's base forms. And we no longer have the ability for it to be able to be scanned, pulled into reports, done anything like that, right?

You think, OK fine. Can I just re-block it? Yeah, but you better make sure you're naming it properly, right? And that we have everything saved where we want it to. And everything is linked to the way it should be. So much easier to just go with the Do Not Explode.

There's lots of other tools to edit things. Most importantly, the block editor. So just very, very important. Number one thing because, if you've noticed, we've moved to drawing creation tips now. So that's the number one thing in drawing creation.

Next thing, so this Schematic List. Who's ever used the Schematic List, where you insert for your panel footprints from the Schematic List? Hopefully you do, don't do extra work. It's already there for you. You've already done it.

So the Schematic List, if you're doing that-- and what it is, for those of you who don't know, is it produces a list of all of the schematic components you've already inserted in your drawings, with all the manufacturer part numbers, with all the tools connected, everything. The location codes, all of that, all inside this dialog box.

Well one of the other things that I think a lot of people missed that you can do is you can, actually, have it insert your footprints based off your wiring diagram tables. As opposed to just

your footprint database. When we get to the customization tips, I actually have an extra tip in here on how to get extra wiring diagram tables in there.

But any of you who actually want to do true wiring diagrams where your footprints carry the wires, as well, and are doing wiring that way. This is a great utility for that. Because that way when it looks up your part number, it won't go to the footprint. It'll go to the actual wiring diagram symbol that has those wire connection options.

All right, connectors. Who's placed a parametric connector? A couple of you. So we have some really, really, really neat tools inside of Electrical that don't just do individual connectors. You can actually place parametric connectors.

And when you do that, one of my extra little tips here is if you hit the Tab key when you go to Insert, it'll flip your connector. And then if you hit it again, it'll actually reverse all of your pin numbering. So it'll flip all of those things live, while you're doing it. So you can do it inside the dialog box. But this is one of those little extra hints and tips for you.

OK, these two are both for 2017. So the Insert and Link to Inventor Connectors is an additional tool you can see right here, in this image. You have this option now where you can right click and insert your connector from a list. So you can actually pull it in if you've already done it in Inventor.

And the same thing, you can link your ACADE Components to their Inventor Parts, also now right from the project manager with the Location View Tab. So you're assigning from a componentry from one to the other as you can see over here. So pretty cool that we can keep all those things linked.

And if you didn't know this or you haven't seen Randy's classes in the past, there are some really, really cool things we're doing now with Inventor and Electrical connected to one another. So there's a lot of power to it. It's all automated now. There's no exporting and importing. So that's just another additional add on to those new tools.

All right so these I actually want to show you. So I'm going to jump in to do this. But Align and Toggle normally open, normally closed. So well, it's swinging.

All right, so some of the editing tools that we have and these are just-- these are for some of the newer users, those of you who have been using it have probably done this before-- but

this is just one of the most powerful things that I think is so handy and so easy. If I right click on a component and I want all of the rest of these to line up with this one, I can grab all of them. And you can window around them too.

But just to show these to you. How perfect was that? And they all just automatically lined up. So that the Align command.

But to take it a step further, those of you who do point-to-point drawings, that aren't necessarily ladders, I get a lot of questions about Align. Because it'll sometimes default to something that's opposite what you want. Obviously that was a vertical alignment, right?

What if I wanted horizontal alignment? Not logical in a ladder, but might be logical in a point-to-point somewhere, right?

So if you actually run these commands from up in the ribbon, itself, this will actually open up and allow me to toggle between horizontal and that default to vertical. So if you've never done that, it's a nice little addition to your Align tools.

And the other one that was next on the list is Toggle, normally open, normally closed. And just to show you that, this is a really neat tool too. It's not in the right click, you have to find it up in the ribbon. But if I grab this guy and I grab this component, it will actually swap this block out to it's normally closed companion.

So exact same symbol, has to be named exactly the same except for that Toggle for normally open, normally closed. But it'll actually swap it out. And it'll swap it at the parent. So in the parent, it will now say that that's actually a normally closed contact. As opposed to a normally open.

All right, pretty cool. It'll even do it for things-- let's see if I have a good other example. I don't think I added it to this drawing. I should have. That's all right, I'll show you in a-- well, let's do this one.

So I can do on that limit switch too. Normally open, normally closed. I was looking for a push button than I normally have on this drawing. But so, it'll work on anything that carries that Toggle.

All right, one of the other really cool things is the find, edit, and replace component text. This is kind of embedded. It's under this re-tag components option, right here. It's this guy.

And we'll just say I'm going to do this project wide. Here's where you can see where those folders come in, right? So I could actually do this by sub folder, that I was talking about earlier. But I'm going to do the whole thing.

What this dialogue box does is it lets you have access to every single attribute in all of the components. And you can do global find and replaces throughout the project. So it takes the standard AutoCAD find and replace and just kicks it up a huge notch, right? So really neat.

Once you get one good, solid project done, you copy that for another customer. You know that you need to do almost exactly the same thing. But you have one that needs different manufacturer option or a different location code naming convention. And you can just do those automatic find and replaces.

And they have-- they'll remember everything that's in there. So when you go to a find, you can get a list of everything you've already used. So you don't have to be manually typing. Which is pretty cool.

Sorry about all the wind. There's a lot of stuff hanging. Makes it a little harder. I'll try to hold on to it.

All right, back to the PowerPoint. OK, so-- oh, and I should have grabbed this one too. You know what, hang on. We'll do this one live too.

So this other one is underneath all of our attribute tools. So if you ever wanted to hide or unhide an attribute, obviously we have lots of attribute tools that you see just from the right clicks here. But there's a whole lot more built in to the ribbon.

And one of the cool things is this window multiple, if you've never gone into it. If I were to grab this whole drawing, it will give me every single attribute that's throughout this drawing. And I can choose which ones I actually want to be invisible or to make them visible again. So it's one of the neat-- it's one of the neat little extras, if you've never looked into it. As opposed to all the manual clicks and hides that we do just from that that right click menu.

All right. OK, so next couple of things. These are also great, just, global editing tools for just like I talked about before. With the idea of, you've created a template project, worked great for one customer, you're going to copy it, you're going to use it for another one. But you need to do a global update of something.

Well we have two utilities that will do some really nice things on your particular drawings where you can actually run this first one, the Copy Catalog Assignment. You can take a catalog number from one component Or do a catalog look up, either way.

And then, as you can see, it even gives you the steps. You select your part number, you click OK, and then you grab all the components that you want to have that part number. So it's a real easy override. So you're not right clicking and editing component for each one of those.

Same thing with this guy, this Copy Installation and Location Codes. You can easily populate a bunch of installation and location codes right through this utility. And these are all on that Schematic tab.

All right, Resequence Item Numbers. Who uses item numbers? Ballooning. None of you? All right, I've got one.

These are obviously connected to our part numbers, right? So they can be linked inside of all of our bill of materials. They can also be attached to our automatic balloons that we have in Electrical. There's a new update to Balloons I'm going to show you in a second. But in 2017, they've actually modified this pretty significantly with removing all of the, just, per drawing settings. Because there was a lot of duplication then and there was a lot of errors of reusing the same item numbers that way.

So what they've done is they've removed that. And they're actually processing all at a full project level. Also they're no longer-- this is a good one, this top line-- they're no longer opening every drawing as it goes through to resequence all those item numbers. So it doesn't have to filter and open and close and open and close automatically to do those. It's all happening in the background now.

So and there's also-- if you guys have ever been into the project properties-- there is a setting that allowed you to choose whether or not item numbers were going to be at the project level or per drawing. And that per drawing option, that reset, has been removed. Because they're trying to remove that chance for you duplicating all of those item numbers,

All right and there's the Item Number Balloon Update, the next one. So this guy now has a new little check mark, where if you change how you want your balloon to be set up in the panel configuration tools, you automatically can say that you want to apply it to all of the balloons that are already in the drawing. And then it will update all of them. Which is pretty cool. So

again that's a 2017 option.

Multi-Bus to a component. Who's ever used the Multi-Bus tool? Yeah, especially if you have connectors, it's pretty cool. If you don't really, kind of, dig down and read in the Command Prompt that there is an option to continue or flip, keep in mind that there's that ability to do that.

So you can actually-- and that's what this Multiple Bus tool is showing here-- if we attach two components and we want to zigzag a lot, you can keep hitting C at the Command Prompt. And it'll keep giving you turns to your Multi Bus, which is pretty cool. Or you can flip it if it looks like it's folding over on itself. So those are some extra little fun tools.

Who's ever used Wire Sequencing? Yeah, a couple of you. Really neat tools to be able to make sure that you're truly saying how you want components to be wired. So something like this, where you're seeing from this button to this connector down to this light, you can actually reroute how this looks and how it connects. And that's exactly what we're operating here.

So you can change the sequence and then you can actually run the show and it'll give you these nice little arrows, right visually on screen. So you can see that. So it's a pretty neat utility.

All right who's inserted PLCs? All right, very cool. More of you need to go out and do it. It's really awesome.

The parametric tools, they're all built in there with all the manufacturer PLCs that are pre-designed into it. So you can specifically see, like this one, you can tell I was inserting an Allen Bradley 1756.

But one of the really neat things is, if you hit Escape. If you've broken that module, you hit Escape, you don't want to insert the whole thing. Because you need to go to another sheet. Or you need to go on and build another ladder or you need to put something else in there first, it will save it.

And that's what this continue option is for, the second time you go back to insert it. It will automatically take you to this dialog box and say, hey, we saved half of your module. Do you want it now? Do you want to insert that? It's pretty cool. So it will save that. It will give you time to do whatever else you need to do. So it'll remembers that.

So our cross-reference tools. So on our cross-references, you can actually choose-- if you've

never dug into the actual settings, here, and again, these are in the project properties-- you can actually choose to display unused contacts, right on screen. So you can see how many you have left, what you still have to go with on them. Obviously we can see these things too when we get into the other dialog boxes for them. But this is a nice visual right on screen.

This is new for 2017. When you add source and destination arrows, they automatically update each other, just like parent-child relationships do now. So you don't have to go back and say, OK, go update source or go update destination. It's automatically doing that, so that one I'm really excited about. It's good.

Who's ever created a custom workspace before? Yeah. How many of you get into the software and say, holy cow, this is a lot of ribbon tabs? Yeah. Or you really want to change around how your project manager is set and all of those things?

I'm going to do this quickly. But if I were to right click anywhere, I can start unchecking what other different tabs I don't want to see. So maybe I'm not going to use Raster. I'm not going to use my add ins. I don't need conversion tools because I drew it all in Electrical. We love our Express Tools. Let's say that maybe I'm not using BIM 360 in this sense.

So I can start to trim this down, right? Makes it a whole lot less messy. And then all I have to do is come in here. Save Current as. And I have my own workspace.

And you can easily toggle back and forth between the ones that already exist with the software and your existing ones. Or your newly created ones, I should say. So it's a nice way to, kind of, streamline what you need there.

And you can also turn off different panels of the ribbons. I didn't want to take a ton of time to do that. But you can turn off those and just get the specific tools that you want as well.

All right so these are a bunch of 2017 tools. Inserting from the Schematic List. So, again, going back to that Schematic List that I just got that raise of hands if you guys have used it. You can - this Sort option, which is awesome. It helps you to organize how you're going to place all these. It's why you could Shift, Select and just plug-in a ton of motors or a ton of push buttons all in an operator station, something like that.

It allows you to save that sort automatically. So it's going to remember it every time it goes back in. You don't have to redo it. And then re-go in and pick all those again.

The installation, Location, and Tag attribute. So this is a big one. And just so you know, the quick reference guide, I kept it on here. I was torn about this. If you guys all flip to the back of it, you have a list of all of the attributes that every component can have on it. And not only what that attribute does, what it's intended for, but also the max characters that it can have.

Tag, Location, and Installation code have all been updated to support 255 characters now. So I was torn about whether or not I put that on here. Or if there were a lot of you who weren't in 17 yet, I wanted you to still have the original numbers.

But yet in 2017, you can get a lot more. And I think one of the biggest ones I hear that with is the Tag attribute. So it's good that you now have-- if you come up with 255, that is a big tag. I'm impressed. I want to see it. So that's a great support now as to how many characters we can have.

And the same thing, also in 2017, in the Terminal Strip Table generator, you can actually sort that. By actually clicking these headings. And then you could Shift, Select groups of them if you wanted to. To be able to have those all insert automatically, as tables. So that's kind of a new thing. This wasn't dynamic like that where you could click on those. So pretty cool.

All right, so this is kind of a fun fact thing. Do you guys all know what a Trap Distance is? Have you heard of it in Electrical? No.

Have you noticed that you don't really have to use object snaps? As long as you get close to a wire, components automatically snap in or other wires automatically snap in. Some of you are nodding.

All right, so if you didn't this, this is kind of an interesting little fun fact. That Trap Distance is actually done by an invisible crossing window that goes out and around the wires that you're attaching to. To be able to find that location to snap to it.

So it's all happening in the background. But it looks something like this. So if I'm connecting one wire to another, it actually throws that Trap Distance, in that crossing window, around both wires to be able to find that perpendicular to it. So it's kind of interesting how that happens in the background.

It also-- another little tip here-- wire numbers, if you ever just manually move them away from the component or the wires themselves. If they get off too far of that Trap Distance, outside of that crossing window, there, they will no longer connect to that. And that's how we get floating

wire numbers.

Have you seen that in the drawing audit? If you hadn't, we're going to talk about drawing audit in a second. So preview to another one of these.

So that's one of the nice things, too. So just keep that in mind, that that's how that's actually doing all of that. So I those are interesting tips as you're going through these.

OK so on that same note of wires. Hopefully you all know that you should always keep wires organized on proper wire layers. Not only is that going to make it easier in your drawing-- to process that it's properly a wire-- but it also will make reporting wire from two lists and all of your wire labels much more automatic, right? Because our labels can actually automatically be our layer names. So that's easy too.

Location boxes. Who's ever actually used a location box? Yeah. Good. Awesome.

So, right there on the Schematic tab, you can insert with a location box. And it will automatically, dynamically build a location box around your symbols. But one of the really cool things is-- and this is some of the things I just wanted to touch on about it-- if you move a symbol in or out of it, it will automatically update the component. To either match what's in that location box as you move into it. Or it'll go back to whatever your drawing settings are as you pull it out of it.

And it's actually the insertion point of this symbol that counts whether or not it's inside that box. So if you a very large symbol and it was, somehow, the insertion point was inside of it, that's how it would know that that's considered in. Or if you, by accident, maybe drew around a attribute that wasn't really-- you didn't include all the attributes in it, it's actually the insertion point at the symbol that counts. So it would still be included.

All right. Multi reports-- or, Multi-page Report generation. How many of you have built a bill of material in AutoCAD Electrical? A few of you. All right.

How many of you have wished that you could put it on multiple drawings? Have you ever tried? Have you just thought, maybe it's going to work? Because it doesn't really seem-- from this dialogue box when we first see this-- that it's really going to happen on multiple drawings. It's not like the Terminal Strip Editor, where we see how many drawings it's going to create at a time.

But, rest assured, it will actually do it. And what it looks like is, if I go here. If I get to the table generation. If I actually set how many rows I want per section and then how many sections are on a particular drawing.

As I go and it sees that it needs to do a new drawing, it'll jump me to this page. And then it'll actually start to ask me what file name I want it to be and everything else. So it will start to create additional pages for you automatically. But it's kind of hidden you don't really realize that it's there unless you actually try to execute it first.

All right, so another cool tip from 2017. We can combine the AutoCAD Electrical and Inventor Bill of Materials into one. So, obviously, I said before that we started to get this EMX workflow. Where everything's automatically connected. There's no exporting now into Inventor and importing back in.

But now right inside of our Report Generator, we can choose to include Inventor parts. So we have a new little check mark there where we can say that we also want those Bill of Material tools to be included. Pretty cool.

Who's actually doing the EMX thing? Are any of you using Inventor and Electrical? A little. Testing it out, kind of. All right, it's very, very cool.

OK, the audit tool. So I started to, kind of, briefly talk about that when I talked about floating wire numbers. When I was talking about that guy down here. But the electrical audit-- if you've ever taken my classes I have a top three I always tell everybody. And I'm going to talk about those. But there's also been a new addition.

So the 2017 part of this is that this mark, as an issue versus mark as ignored, has been added. So a lot of times people-- yay, I saw the clapping.

So this was a big thing. All along I would tell people, don't panic when you see that you have 1,000 errors. They might not be errors, right? They could be just a red flag or a warning or it might be something that isn't an issue to you at all. But it just flagged something to pull up into this dialog box due to how we have rules set.

So that's why we can now do this. Last year this was just something if you had the subscription pack you could do. Now everybody can do it. In 2017. So it's a pretty neat thing.

And you can also choose-- that's why I have this highlighted-- to hide all ignored issues. So

every time you go into the Electrical Audit, it'll ignore anything that you've said is not truly an error.

AUDIENCE: Question about that. Is that per session? Or does that persist across [INAUDIBLE]

TIFFANY
BACHMEIER: I believe that that per-- I think that's saved into your project, itself. So it should be part of the database and it should remember it for all sessions. Now it's not going to jump projects. But it would be for that project entirely. Yep, that's a good question.

Now for Electrical Audit, in general, even if you don't have 2017. These are my top three and I just want to tell you why. So the top three things that I would always check as just a good best practice at the end of a projects.

The Wire No Connection category, which shows you if you're missing Source and Destination arrows. One of the biggest things. Because if it's just an unconnected wire, it's just dangling there, right? So you're missing those signal arrows.

Your next one, the Component No Catalog Number. It actually probably should've been moved to my top spot. That's your Bill of Material, right? If you're not-- if you put that component in there, but there's no Bill of Material information into it, it's not going to end up in your report. So it's going to get ordered. So that one's huge.

And then the last one. The Child No Connection And in other dialog boxes, it's called Orphan Children, very sad. But it can actually show you all of the child components that you have, that are not connected to any parents. And that's huge. Because they're not doing anything, then. They have no intelligence to them other than they've broken wire.

So those are my top three as just a good check mark for every time you run through a project. And you want to make sure that you've got as much done as you can in it.

All right, DWG audit. So my tip on this one. And if you guys have ever used this, this is just an awesome tool for cleaning up, kind of, sloppy drafting errors. Or people who have maybe used some regular AutoCAD commands, like manually deleting things. And they've left other stuff out there.

So if you've never done it, it's great because it cleans up all those little things. Like bogus wire numbers. I love that the word bogus is in this dialog box. I find it hilarious. Zero length wires, wire number rubber floaters.

But my extra to you, if you've only ever run this as a project wide command-- because that's logical, just clean up all errors through the whole project-- you've never seen this little extra check mark right here. And what this one does is it actually will show you, as a big visual on screen, all of your wire networks. It will point out, from wire network out, to all the connected wire networks.

And you'll get this vast visual on where all your wire networks are and how they're connected. And it's pretty darn cool. So you'll only see that, though, if you run it as Active Drawing. So just a little extra there with drawing on it.

OK I'm not going to spend a ton of time on this. Because there's a whole class on it, here, tomorrow morning at 8:30. So if you're not signed up for it, it's the-- I can't remember the exact, I have it at the end-- something about the Dewey Decimal system. But it's Rob Stein's class on the new SQL Server, inside of AutoCAD Electrical, as opposed to just the Microsoft Access Database.

And it's very cool. Go to that class. It's going to be awesome. But it's a new thing for 2017.

There's options for both the Deployment Model and Standalone Mode. And there's all kinds of cool stuff with it. But I'm going to save the rest of it for him to be able to show you. But that's one of the new awesome tools. And a big request point from lots of customers, that they really wanted SQL to be supported. So there you go.

All right, catalog browser. I know how many of you said are using 2017, but how many of you, at least, have 2015 or newer? OK, that was actually not a ton of you.

[INTERPOSING VOICES]

TIFFANY 14?

BACHMEIER:

[INTERPOSING VOICES]

TIFFANY OK, so maybe not backwards too far. All right, the new Catalog Browser that started in 2015--
BACHMEIER: and then has obviously moved through 16 and 17, as well-- all of the editing tools are live, inside of it, just like this. So it's pretty cool how it opens up and does this. But this is my extra little tip is you can actually see that you can add additional 2D blocks to this.

So you can automatically assign a default symbol to a Particular Part number. This is for Schematics. This is not the footprints, that's a totally different thing. But you can actually pre-assign them. And you can add additional ones. And if I were to click in to here and actually get a third one in here, it'll give me a fourth line and a fifth line.

So you can keep adding default symbols that would go with that Part Number. So you get more than just one option. You can do the same thing with the actual Inventor ones as well. The 3D parts, this column, If you didn't know this, the symbol 3D, that actually goes for Inventor parts. Because we're all connected in that catalog. So it's pretty cool.

And then the other part, that is a big part of this new Catalog Browser, is that we can edit the Pin List and the Terminal Properties Database all inside of it at the same time. So those of you have ever done this before, remember we used to have to go out. And we had a separate area where you had to go out to the Edit the Pin List Database or Edit the Terminal Props.

Now it's all inside of it. And you can edit all of these right here. So you can not only see that symbol area, but the Pin Lists tools as well. And if you were in a terminal, the Terminal Properties. So that's pretty cool, too.

And it makes it a whole lot easier we don't have to remember all the codes for whether or not it's a normally open or closed, can be either, convertible. So that it's automatically doing it by us being able to pick drop down. So not nearly as much code memorization.

And along with all of those with the Catalog Database, who's ever used the Textvalue? Good. OK, so there's some of you. That's awesome.

So this is an extra little thing that if you've never really dug into the catalog database, you can actually have it auto fill out other attributes than just manufacturer and catalog number. You maybe you've seen it with ratings. We see it on fuses and motors. It'll automatically fill that out based off of the part number.

But the Textvalue, you can automatically tell it, this attribute equals whatever it is that you want it to do. So you can put that information right into the Catalog and have it prefill that stuff out. Which is pretty cool.

And then-- do I have another image? Yeah, that's what I thought. So the List Box definition table and the Catalog Database as well. This one's sometimes hidden.

I get a lot of people that ask me, how does it know the filters? Why is it when I go into the Catalog, it only shows me Allen Bradley every time I go to a control relay? Well all of that is what's inside of this LISTBOX_DEF Table, inside your Catalog.

So if you actually go into the Catalog-- obviously I'm in Access here-- but, in Microsoft Access. And you can see, if you look at the particular family code, what manufacturer, what type, what coil, so on. But it's going to filter by it to give you just those specific filters. So you can add and customize your own filters in there, if you want to, right in the Catalog Database.

All right and then that's what would relay back to here. So if this was set up, here, in the Access Database, this is what I'd be seeing filtering on the catalog itself.

All right, how are we doing on time? Oh my gosh, I have to go faster. OK, I thought it was going fast enough as it is.

So additional Catalog Database columns. If you've been in the new catalog and you've thought, man, I really want to search on the user one field? How many of you use user one? For your own part numbers? How many of you use your own part numbers as opposed to manufacturer part numbers?

So that's what the user one, two, and three fields are. If you want to be able to see those, what you can do is you can right click in the Catalog, go down to More. And it will give you a list of all of the different category types that we have from the Catalog Database.

And anything that's check marked is what you're doing this little Google search, basically, on. So as long as it's showing up in there, on screen, you can search on it. So add extra ones if you want to be able to filter and add buy more things in your searches.

OK, Wire Connection Attributes. So if you've ever wondered, or you've ever gone in to build a symbol, and you've, hopefully, used the Symbol Builder. Because it's awesome and it'll make it a lot easier. And it, kind of, does it for you by you saying, I want the wire to come in from the left or from the right or from the top or from the bottom.

But if you've copied and pasted a symbol and just, kind of, copied and pasted those attributes around, you might have realized connections are happening in the right spots. And they're coming in weird ways. It's because there are specific numbering sequences that go in with this for what a number in the actual description of the-- or what the actual attribute on the wire

connection point is.

So if you see at the very top there that X@ TERM 01, that means the wire is coming from the top. So the X2, that 2 right there, is for a very specific purpose. So can't just copy that and put it at the bottom. Because then the wire is going to cut right through this symbol, right? So if you didn't know this, this is here for you guys to go back and use. And look at all those different setups for how those are all associated.

And the number at the end of these, like the 01, is what your other Terminal Option Attributes go with. So the term description and the actual term number. And that's what the Pin List associates to.

OK, so all that's there for you guys to go back and reference. Same thing here. I'm not going to read all of this, but this is for your reference. And it's also part of what you have in your handouts on what the naming conventions are. So remember how I said don't explode? It's so that you don't have to memorize what you need to go back and remake that to do to make sure that that symbol works like it needs to.

So this is all of the different Naming Convention tools. And if you didn't know things like the third character of a terminal symbol actually says whether or not it forces a wire number change or not as that passes through it. Those are all specific things that go into that name. So it's pretty cool.

All right, Component Tags. So I talk about this again in they handout as well. In the Attribute Tools, you can split your attributes to go across two lines. So that's why you can have a TAG1_PART1 and a TAG1_PART2. So you can actually have them be able to be stacked. And it'll do things automatically.

Like if you look at, say, a control relay, like CR402, it'll put that CR on top. And the 402 on bottom. So it'll split it automatically.

Who's ever heard of that ACE_Flag attribute? That was [INAUDIBLE]. No, no one?

All right, so these are some cool things too. There's different values that will flag different things, but-- I actually thought that was me that was beeping, you scared me. The value of two will identify it as a parametric connector. Or a three as a parametric generated twisted pair representation.

So there's some different things that can go in to those specific attributes that actually fire different things to be happening inside the software. And same thing, the WD jumpers attribute, it's one of the ones that's automatically in the Generic Attribute Template tab. But you can automatically add internal wiring for internal jumpers right inside of that. So that you can forcibly do that. And I have examples down here you can go through. So there are some extra attributes that are kind of hidden that a lot of people don't even realize are there.

The default WDN file. So this file will actually determine, for the Electrical Audit that we were talking about, what things it should ignore, inside that audit, for terminal filters. So if you want it to ignore more so you don't have to go in and right click Ignore, you can actually define these inside this file. And it is one of the files listed on the locations in that handout I gave you. So there's lots of information you can read on that there.

The search sequences, as well. So if you guys have ever realized that we have all these different support files we can create between our title block support files, our lime labels, the installation and location code options, all of those. This is for your reference of the filters that it goes by. That you can either name it with the project name and put it in the project folder. And it will only work specifically for that project.

You can do the default name in the same folder as the project. And then it'll live with it, but it could also work for any other project if you were to share them into the same folder. Or out in your support directories. So let's just keep that hierarchy in mind whenever you're building those customized files.

The wiring diagram tables that I was mentioning earlier. The way you get them is if you actually create the footprint database. If you go in and create the same exact part number, like you see the Allen Bradley, right here. And then Allen Bradley underscore WD. That's how it will flip to going to just your wiring diagram symbols.

So if you want to have both, that's how you build them in there. So that your Schematic List will find those as well. All right, we're almost done.

Modifying the Symbol Library. If you've realized that, man, I really need all of these to scale down differently, right? If you need them to be a different set up or maybe you want a different height for a bunch of, just, your tag attributes or something like that. As opposed to manually go through all the symbols, this is kind of hidden. It's under the symbol Builder. There's a drop down under the Symbol Builder.

But you can go in and you can rescale everything. You can change specific text types for the attributes. There's all kinds of-- you can even run an auto list for [INAUDIBLE] if you want to do something specific to all of your symbols and the whole library. My advice, always make sure you save a backup of that original one. So you don't have to go back into the install folders and reinstall them.

But you can do that globally. So that you don't have to be manually doing that throughout all the symbols.

Spreadsheet to PLC tips. Who's used-- I think already asked you guys this-- who's inserted a PLC? But who's inserted a whole drawing set through the Spreadsheet to PLC [INAUDIBLE] utility? All right, if you haven't, come to our class, right after this. It's going to be awesome on the automation of that utility.

But there's some really neat things when you're building your spreadsheet of codes, like, new drawing breaks, space, or skip. That will do specific things in that Excel file as your auto building all of those drawings.

Last project fill, this little text file-- and, again, I think I have the path to it on that handout-- this actually remembers this is how it knows all of the projects that are in your Project Manager every time it closes. And what your recent ones were. So if they're active or inactive, as you can see from that list, it actually knows all of those Toggles.

So if you're ever having issues with one or you want to get one out of there or clear it out so that it's not reopening. I've had people have corrupted project files that they need to get out of there. Go into this and you can actually fix that. You should never have to, but if you did it's good to know.

This one I'm just going to leave for you guys to read. But I get the question all the time, like, why do my users need Admin rights for electrical? Well there's a big explanation here of why. But a lot of it is the automation that we have that's happening in the background.

And some of the auto commands that are fired when we do certain things. And that's why. So I thought I'd include that for those of you who are dealing with CAD management on that side.

The Environment file. Those of you who know it, know that it's kind of the daddy file of all things for AutoCAD Electrical, right? So my tips on that are, make you've saved a copy of the

existing one before you start editing. And if you move it, make sure you make the path that it's going to be at the top of your search path in your options dialog box. Your standard AutoCAD options dialog box.

All right and we're the last ones. So the last ones are for templates themselves. The WD_M Block, if you guys have never actually gone into it, there's some things you can hard code into it to edit. Which is kind of cool.

But there's some tools that you can do, even from just upgrading versions and having really old project files that don't have all the new stuff from the new drawing properties and project properties. All of these commands are located in the drop down of the Other Tools panel on the Project tab of the ribbon.

And I have notes to all of that stuff when you guys get this PowerPoint. But all of these give you updating options that are automatic to cleaning up those files. So if you've been in electrical for a long time and have really old projects, using some of these tools will help to refresh those.

And I give you an extra one, a number 61. When you're saving your templates, good, best practice is always set layer zero as your current layer. That way when you go into building your blocks and you're in the block editor, you don't even have to think about the fact that you go, wait, I need to make sure that I'm actually still on layer zero when I'm building this. So just a good best practice.

All right, woo, with, like, two minutes to spare. So I knew I probably wouldn't have time right now. But if you guys want to hang out, I can answer any questions afterwards. Or I can head to the lounge if we get kicked out of here, just outside this room, and we can-- I can answer questions.

[APPLAUSE]

Please fill out surveys. Thank you for everything.

[APPLAUSE]