"All about the Libraries" Webinar (Part 1) March 2014

Questions and Comments from Libraries Series

The following questions were asked in the chat or question box during the webinar.

Comment / Question	Response
Will there be a URL link to the recorded meeting after the completion of today's presentation? Can I get an invite for next week? Can I get the PowerPoint Slides?	Yes! Request them via http://ninedotconnects.com/webinar-request-fundamentals-of-library-structure-052414
Will this session cover importing and properly aligning 3D models into PCB footprints?	Unfortunately, no. Our focus will be on libraries in general to ensure that our customers are using Altium's library offerings that best fits their situation. However, 3D is a topic of interest and we will consider doing a webinar on this topic at some point in the near future.
If you can do a basic overview of libraries before you get into specifics, please, it me be helpful	This was requested at the beginning of the presentation. Hopefully, we achieved this request in the first webinar. I would encourage all who are reading this to join us for the 2nd part where I will give an overview of all PCB libraries that Altium makes available. This will be a "tour" rather than a PowerPoint.
Is there a way to search in the Altium vault for parameters other than description or comment?	As far as I know, the answer is "no." However, I will take this as an action item to confirm it. This is something that has been asked by other customers and I plan on addressing it in 2nd webinar. I do know for a fact that there is no way to do a cascading of parameters in the vaults. Therefore, in the vault, the description column is VERY critical given this potential limitation; however, the vault's ability to handle purchasing data more than makes up for it.

LIBRARY QUESTIONS

Question	Answer
Do I understand correctly that every time a resistor manufacturer "goes south" I will have to go through *all* my schematics and find equivalent resistor from other manufacturer? With Altium ideology of "hard-wiring" part number into schematics, changing resistor in dozen schematics becomes a terrible work	It all depends on how you set up the library, regardless of it being a "symbol centric" or database library. If the purchasing data is being kept separately via PLM, you can use the .dblink file to push that information into the component on a design from the PLM. If your Altium libraries are maintaining the purchasing information, you will have to manually go in and make changes. The Tools » Parameter Manager in the schamatic library editor can help with the "symbol centric" libraries, while the user can make direct edits in the database library table. If you are going to use data from the PLM, you have to have a part number associated to the "symbol centric" library, or row in the database. This is the ONLY way that Altium can relate the PLM information to the component. The vault, if the user puts both their designs and libraries into it, has a "where used" feature that allows you identify the designs that are impacted by an obsolete part. In addition, the vault supplier links are based on manufacturers, not vendors. So if you have multiple manufacturers for the part, it's simply a matter of removing the link of the manufacturer who "went South". Another feature of the vault is that you declare which manufacturer you want to use when it comes time to completing the bill of materials. If later the manufacturer is no longer available, you can see which projects are impacted, you can rev the project, and make the change. Keep in mind that Altium libraries uses a static library methodology. I did not talk about this during the webinar because I wanted to focus on the intelligent data. However, the term "static library" (not static data) refers to the fact that when you place a component from an Altium library, it is a COPY of the component. Though the part provides information as to where it came from, Altium does NOT change the component on a design automatically if you makes changes in the library. The user has to update the components in order for the changes to propagate. This may sound limiting, but it's a far better approach than a dynamic library in which any changes to the library immediately result in a change to the schematic. From prior

	experiences, this is how Boardstation and Powerview used libraries and it was miserable!! There are several 2 MAJOR disadvantages to dynamic libraries: 1. If you remove a part from the library, it gets removed in ANY design that made reference to it. Unfortunately, you do not know who may have used the part and by removing it, you possibly compromise existing designs 2. If you change a part graphically, the changes will propagate to the design and may cause the primitives to disconnect, which in turn, compromises the netlist. What typically happens: You finish a design, put it away for 6 months, open it, and find that the design has been trashed due to library changes!
I know how to get the .designator in the footprint library for assembly layer, but do not know how to make it take on the ref design number without doing it manually. Is there a way to automate it? Will he cover it in this class?	Unfortunately, I will not have enough time to get down into the specific of footprint creation in the webinar 2. However, I may be able to answer this now. The .designator is a special text string in the PCB tool. This is denoted by the period (.) as a prefix. There are a number of these special parameters. Open the string dialog in the PCB layout editor and press the arrow on the text box. Altium will list them all. The user does not need to add .designator to the footprints for the silk screen. However, they can add this to other mechanical layers. When the ECO from the schematic to the PCB is performed, Altium should be converting the .designator to the actual designator, regardless of layer. If you do not see this, it may be due to the "convert special strings" function being disabled. This can be found in the PCB preferences.
Components may not live long enough. Didn't work out in prototyping, waste of effort to put it into purchasing until we are sure it's going to be used	This is a major problem, no doubt. In some companies, the validation process is so painful that you generally avoid that process until you know for certain that the component will be used in production. Even then, purchasing needs to vet the part to ensure that it has a chance of qualifying. Keep in mind that they have to deal with trade compliance issues such as REACH, RoHS, Conflict Mineral, and ITAR in addition to lead times, vendors, quantities, and Life Cycle status. There's nothing worse than to build a prototype that can't go to production! These are thorny issues and every company has to deal with them based on their existing situation.

In your personal opinion which library works best	I am a big fan of the database library, especially when you have the ability to use the .dblink file to pull additional purchasing data from your PLM. I am also a big advocate of the vaults because it addresses issues that the database simply cannot address. This is especially true when it comes to dynamic (i.e., purchaser's data.) In fact, the vault treats dynamic data as a LINK, rather than DATA. This allows the vault to only display or use data at THAT moment in time. Any data we put into a component or database that is dynamic in nature starts to go stale rapidly.
I use outjobs to automate BOM generation	I certainly encourage it! The .outjob file is a very powerful capability and allows for the same configuration to be used for all projects; however, this feature only works when the intelligent data exists in the design.
Is there anything that NineDot that covers PLM integration with Altium? In terms of exporting directly from Altium into a PLM system	We will not really cover this in the webinars, but I can give you an idea of what is needed to be done. The first thing is to extract the data. This can be done using the parameter manager. I think that you could also use the Tools » SVN Database Library maker with the intention of only using the database part. Once you have extracted the intelligent data from your symbol library, it's now a matter of getting it into the PLM. Second, you are not going to abandon your schematic and PCB libraries; they will be reshaped into generic components. For example, instead of having 1000 resistors, you will delete 999 and keep one of them that your PLM will refer to when you are calling out a resistor. In essence, you are already doing this with PCB library, so, most of the work is on the symbol side. Third, you are going strip out all of the intelligent data in the symbol library. Once you have this in the PLM, the only purpose of the symbol library is to have a graphical representation. All parameters and all model links are removed. In fact, the name of the symbol should be generic enough for general reference. With ICs, they tend to have a specific name. Think this out logically before proceeding. Save file copies before you make monumental changes. Keep in mind that links in the database have to match with symbol and footprint names.
Is there an automated way to "sanity check" and clean up parameters	Not really due to the fact that Altium does not know what would be "correct." Keep in mind that a parameter is simply a name and value. How you use it is up to you.

We sometimes get confused about which library the component is coming out of. How do you recommend we manage which libraries are loaded in a project. Should we keep all parts in one library only	I generally recommend that they are kept in the same library. If I have to make changes (especially in a database) it's a lot easier to deal with one table than open separate tables and make the same edit several times. If you add a column to one table, you need to do the same for the others and you have ensured that you kept the naming and value convention consistent.
All components shown are one manufacturer to one part number. What if you wanted to have 3 or 4 approved manufacturers for a single part	This is can be done in both the "symbol centric" library and the database; however, it makes for a messy bill of materials. You will need to have different parameters for each manufacturer (i.e., mgf1, mgf_pn1, mfg2, mfg_pn2, etc.) Another way is to literally have each of these as a separate line item (in the database) or as separate parts in the "symbol centric" library. This will resolve the messiness of the BoM, however, you will have to be specific about the part option you want. For the vault, you simply link all of the manufacturer's part numbers to the component and then reconcile the "flavor" in the bill of materials.
Is the schlib a different view of the database library (i.e. are they the same library)	The schlib is a separate editor from the database library. In fact, the database aspect is handled by tools outside of Altium (i.e., Excel, Access, Team Center, Arena, Agile, etc.) The .dblib file simply points Altium to the type and location of the database. I will demonstrate this next week.
We use different schematic and layout tools based upon customer demands, What would you suggest for handling libraries?	Definitely a database library, assuming that the other EDA tools can read a database library. You can have a universal database in which you have it point to different symbol or footprints. Altium has its own parameters that it needs to see; however, those could can be ignored by other tools (unless they require the exact same column names)
Any suggestions for small company of 3-4 engineers with no librarian to keep up with library	Step 1 – appoint or elect a librarian. It does not matter how small the group. Everyone has their own way of doing it, so someone needs to be the gatekeeper. As for type of library, I would recommend a database library using MS Access. It's the easiest way of maintaining and adding parts to the library. If your group is also handling the purchasing of components, then consider the vault. This will easily handle the supplier data for you, especially if you are buying direct from vendors such as Digikey, Mouser, etc. As mentioned before, the vault deals with real time supplier data, therefore, you simply set up the supplier links and you are set to go!

Our company uses pcad currently and we are transitioning to Altium is there a way to convert our library from pcad2006 to Altium designer	Altium has a transition wizard for the PCAD. If you do not see it in the import wizard, then make sure that the extension for that translator has been installed.
This is how I describe a resistor footprint RESC1005X40N(0402)	The only addition comment I would make is that in addition to the footprint dimensions, you may also want to mention the height. Different resistors will have different heights, which, in turn, will require different component bodies or STEPs. You may have 12 dozen different footprints with the same pad dimensions, but different 3D representations.
Any thoughts on connectors to PLM applications such as Agile, and others	Altium can interface with these PLMs through ODBC; however, you are at the mercy of the tool as to when Altium can interface with them. We see a need for the ability, for example, of being able to press a button and simply have Altium upload the BoM to the PLM. We are having those conversations. It's just a matter of resources.
Does Altium provide an option to treat case insensitive treatment of parameters in database	Unfortunately, no. Based on the support cases, it looks like this is also an issue for net names as well.