Symbol Library Creation Q & A

Question / Comment	Answer/Reply
Design rules are so powerful, so why are people still using courtyards?	The design rules in Altium Designer are very robust, especially the component clearance rule. However, there is still a need for courtyards. 1. They are a visual that allows the user to place components in the 2D mode. 2. The courtyard can encompass more than just the component body; it can also encompass any clearances for human fingers, rework equipment, test probes, etc. 3. In Altium Designer version 16 (and prior), there's a bit of a catch-22. If one uses a STEP model, Altium will adhere to the STEP model dimensions; if the STEP model is not used, it will adhere to what is known as the 'bounding rectangle.' The bounding rectangle in Altium Designer includes anything drawn on mechanical layers. The concern is this – if one uses a STEP model, the bounding rectangle (and the courtyard) is ignored in the component clearance rule. Therefore, there needs to be a visual way to ensure of providing the proper clearances. 4. Courtyards provide the board assembler some useful information about the placement of the components. As a side note, we addressed the component clearance rule in our webinar titled, "3D in Altium Designer" http://ninedotconnects.com/video-request-altium-designer-3d
What's the purpose of the SCH library?	The SCH library or symbol library or schematic symbol library is a library for symbols that will be used on the schematic sheets. Where things may become a bit confusing is the library methodology that is employed. By default, when a user creates components in the schematic library, they are creating a "symbol-centric" library. That means that the component symbol

	contains links to other models such as the footprint (for the PCB), the SPICE model (for SPICE simulation), and the IBIS model (for signal integrity). The symbol also contains “intelligent information” such as the manufacturer name, part number, etc. that can be used to instantly build a bill of materials just by placing the component onto the schematic. The other type of library methodology is the database library. In a database, the row of information represents the component. Within that row, there are links to the models, including the symbol and footprint. The symbol itself is simply a graphic. It does not contain the intelligent information nor connections to other models. Regardless of which library methodology is being used, in Altium Designer, the symbol can only be created in the symbol (SCH) library and can only be used in the schematics.
Is meta data, like manufacturer part number, only used in the .CmpLib?	If we are understanding the question correctly, we’re assuming that “meta data” is any text data that would be pertinent to the component. We at Nine Dot Connects generally refer to this data as “Intelligent Data” since it is necessary for identifying the component in both the library and in a bill of materials. Assuming that our understanding is correct, then it suffices to say that this data is can be added to the symbol directly. Note that the .CmpLib is not a true library; rather, it is a staging area for uploading components and their “meta data” into the vault (if one has purchased the vault from Altium)
How do I switch the grid in the schematic editor?	The shortcut key is the “G” key. It is a toggle key which will toggle the user from DXP units of 1,5,10 and back to 1. Note that when one initially uses the Altium Designer schematic library editor for the first time, the “G” key seems to go out of order initially and then picks up its toggle sequence. As mentioned in the webinar, one can draw on a grid less than 10; however, the pin “hot spot” which is used for net connectivity in the schematic should be on a grid of 10.

How do you determine scale when drawing a symbol outline? For example, the op-amp example is drawn 40 mils high on the input size. Are there "standards" for scaling the symbol?	The scale is not defined. This is due to the fact that schematics, by their nature, are dimensionless. As for setting a scale, that is left to the user to decide. One may want to base it on components provided by Altium Designer. FYI - The ANSI / IEEE standard 315A shows symbolic representations for every type of discrete component known up to the year 1986.
For alternate mode views, once the symbol is placed in a schematic can you toggle back and forth between views? (e.g view logical symbol during review, view package symbol during debug)?	You can flip between the views in the schematic; however, Altium does not have the wherewithal to reconfigure the nets.
Can component properties be loaded from a table?	Yes, they can be loaded through the List panel. You can use smart grid paste to paste over existing properties and smart grid insert to introduce new primitives with their properties.
How many alternate symbols can be made for the same part?	A quick experiment showed that one could create at least 15 alternate views.
Is there a script to automate the creation of a resistor or capacitor library with different values?	For a script, there may be such a thing; however, before one ventures down the path of scripts, they may want to consider the following: 1. Master the global editing commands, especially the Find Similar Objects (commonly referred to as the FSO), the Inspector panel, the List panel, and the Parameter manager panel. As a side note, as of AD16, there is now a symbol creation wizard under the Tools menu. 2. It is highly recommended that one consider a database library. The database is all text, and can be quick copied, edited, etc. In addition, in a database library, one can have dozens of different resistors in the database, all of them pointing to the same footprint and symbol graphics.

Where can you hyperlink the part number to a distributors website	In Altium Designer, there are 2 special parameters that provide a name for the link (ComponentLink1Description) and the link itself (ComponentLink1URL). Note that these parameters can be used for several links by incrementing the number found in the parameter names. For more information, check out the following link: https://techdocs.altium.com/display/ADOH/Component,+Model+and+Library+Concepts Look under the topic titled “Multiple Linked Documents – Right-click Access”
In order to get R and C values to be sorted properly in the BOM do you always have to put them in the "comment" field? How do you add new column to the SCH Library panel (by default Components and Description column is enabled)?	In the Library panel, one can add, arrange, and sort parameter columns; however, this capability is not given to the SCH Library panel. The only 2 columns in the SCH Library column are the Component name and the Description. Therefore, one must be savvy about how they either configure the name or the description for the purposes of sorting.
Is it possible to have library interaction with SAP? Or SAP with library?	It is possible; however, not directly with the schematic symbol library. In order to use a database library like SAP, one has to interface Altium to the database itself using a.DbLib file and an ODBC driver. The database itself has to have location references to the footprint and symbol files since the graphics are stored outside of the database. For more information on databases: Webinar: Tour of Altium Libraries http://ninedotconnects.com/video-request-altium-provided-libraries

What are common use cases for alternate symbol modes? How can you support SoCs (System on a Chip) with alternative pin functions? I would like avoid placing all pin functions into to same part. Eg. Pin 1 = GPIO1, RXD, SDA, SCLK. I rather have one part that list the pin function that I actually use in the design. This pin function might be different in another design. Right now I am just creating another part for this.	The main uses: 1. When one wants to represent the component both as a discrete and as an IC 2. When there are 2 or more ways to represent the component. For example, the method used to represent the resistor in North America is different from the way it is represented in Europe. 3. System on a Chip (SoC) in which a pin has more than one capability. Commonly found on PICs and microprocessors.
Is it possible to create a "generic" schematic symbol, say for a resistor, and link it to many different manufactured parts? Or is it preferred to have separate schematic symbols made for different component part numbers? Does Altium ever see putting a method of being able to create a single symbol and reference it in multiple parts? It's a real headache to have to copy paste the same symbol time and again. It's especially heart wrenching to have to go back and change all those copied symbols when changes are made.	In a database library, the row of information represents the component. Within that row, there are links to the models, including the symbol. Though there may be dozens of resistors in the database, the database simply points to a generic resistor graphic for the symbol and a generic footprint that matches the package for the resistor. As for the second question, if one is using a symbol-centric library (as described above), then one should have a separate schematic symbol for each part. Webinar: Tour of Altium Libraries http://ninedotconnects.com/video-request-altium-provided-libraries
How about adding pin delay to big IC's in the symbol at once?	This could be done using the IBIS model; however, it would only be of value if one was planning on using the signal integrity tool within Altium. If the netlist was being exported to another tool for timing analysis, Altium does not have a capability to add in pin delay information. This would have to be done by a script.
I had my default units grid set to Mils in lieu of DXP Defaults. The difference is 10,50,100 as opposed to 1,5,10. P-CAD parts was set to 25 which doesn't connect up on 10 mil grid.	The DXP units were created to keep with the notion that schematics are dimensionless. That is why there is no unit posted after the value. However, Altium Designer is a computer program and it has to have a coordinate system for its editors. Therefore, Altium based the DXP unit off of mils, in which 1 DXP unit = 10 mils.

Why would one use Altium Vault… or why not use it?	The vault's strong point is its revision control and life cycle management of projects and their documents; however, when it comes to libraries, the benefits can be debated. In general, the value added for putting a component into any version control system does not really make up for the time and effort. Parts are generally static. They are not going to change. If the manufacturer changes a part, they are going to give it a new part number. Therefore, why the effort in tracking something that is rather static? One of the benefits of the vault is that if the library exists in the vault along with the project documents, then one can perform a "where used" function. This can be useful if a part in the library has been declared obsolete and one wishes to see which projects are impacted by it. However, the vault does not have the ability to swap the part. Each project that is impacted will have to opened, modified, and then saved as a new revision. Here's the kicker – if your company has a good PLM system, one can already do a "where used" search because the part and the project(s) it was used on are related!
What kinds of things does the "Add As Rule" button offer?	Not sure where this was seen in the library, however, it is a feature in the schematic editor that allows a user to add a directive to a component with design rule information that will be propagated to the PCB layout upon executing an ECO. For more details: https://techdocs.altium.com/display/ADOH/Adding+Design+Rule+Directives+to+a+Schematic+Document
Is there a way to place the reference designator or comment such that if you update the symbol in the future (even when flipped or rotated) that the text stays in the same place instead of moving	In the parameter properties, there is a check button called "Autoposition" that can disabled. For more details: https://techdocs.altium.com/display/ADRR/Sch_Obj-Parameter((Parameter))_AD
Do you prefer the DXP component parameter or load component parameters from the Supplier Search?	It is my preference to manually load the static information into the symbol, as shown in the webinar. The reason is simply due to the fact that different components have different pertinent values. When one uses the supplier data, it can provide more information

	than what may be useful. In addition, the names of the parameters can vary based on the supplier chosen. Therefore, it adds a lot of columns to the parameter manager, making it difficult read and requires some clean up. It also provides (by default) dynamic data such as pricing and availability. This should NOT be in a library because the data will become stale quickly. Throughout my experiences, I have found the following 4 things are needed to help identify a component: Company part number Manufacturer Name Manufacturer Part Number Datasheet link As for the supplier data, I would encourage you to add the supplier data to the component in the schematic. At that point, you have the key static data from the library and adding the supplier data will not muck things up since you can easily choose the desired columns of interest in the Bill of Materials.