Design Reuse in Altium Designer Q & A

<table>
<tr><th>Question/Comment</th><th>Response</th></tr>
<tr><td>If we paste a schematic snippet, change the reference designators, AND then we paste the corresponding PCB snippet that has the old reference designators - will the two snippets lose sync?</td><td> Yes, they will! This has to do with the way Altium handles unique identifiers (aka Unique IDs). Every component symbol is given a unique ID upon placement. Even during a paste of a schematic or footprint component, Altium Designer will create a new unique ID for the pasted component. There is one exception. When Altium is adding footprints to the PCB layout during the ECO process, the corresponding footprint is given the same unique ID as its symbol. Altium does this so that the user does not have to worry about losing the symbol-to-footprint relationship if reannotation is performed in either the schematic or the PCB layout. The issue with snippets is that it is a copy. When Altium places the copied snippet, it changes the unique IDs. This is true for both the schematic and PCB snippets. As a result, the user must relink the schematic symbols to their corresponding footprints after the snippets have been pasted. The best way to handle this is by making sure that the designators are the same for the schematic and footprint snippets prior to running the Project >> Component Links (or performing the ECO process if the designators are truly unique to the rest of the project). Altium recognizes this issue and has provided a video that walks the user through the steps: http://www.altium.com/video-how-employ-snippets-design-reuse </td></tr>
</table>

For the PCB snippets, what happens if your snippet has layers that do not match the current design? What about PCB stack up differences between the original PCB snippet and the target PCB? Are there issues there?	For signal layers, Altium will add the signal or plane layers to your design. Be aware of this, especially if the layer stack up was carefully prepared. For the mechanical layers, Altium will enable the mechanical layer if it is not turned on. If the name of the mechanical layers are different (but correspond to each other based on mechanical layer number) the name in the PCB layout will NOT be superseded by the layer name from which the snippet was copied. For example, if my mechanical layer 5 on one project was Called 'Board Layout' and the snippet primitives came from layout that call mechanical 5 'Board Edge', the snippet would be placed on mechanical 5. The name 'Board Layout' would be retained by my layout file.
Is there any way to link the snippets in Schematic and PCB with the reference designators reset before using them? If you were adding a snippet to an existing design with annotation already completed, dropping in duplicate designators will cause issues. This may be the case if a PCB is already made, and you are adding to the design.	A good practice when creating snippets is to copy both the proposed schematic and PCB snippets to a temporary project. Link these together through the ECO process. After this has been cleaned up, reset the designators and provide them designators that would not be normally found in a project schematic design. The annotation capabilities in Altium allow you to start the annotations at any number you want. After this is performed, you will want to perform the ECO process again to push these new designators to the PCB. Take a look at the following video for more information: http://www.altium.com/video-how-employ-snippets-design-reuse

Does it make sense to store the reuse items in vaults? Is it possible to put device sheets or snippets into the vault?	Snippets are not saved in the vault; however, device sheet can be saved. The two biggest advantages of saving the device sheets in the vault: 1) The originals are protected from overwriting, which can easily happen if they were stored in a network shared directory. 2) If changes need to be made to the device sheet, one need not worry about impacting prior designs. All of the sheets are under version control in the vault.
How do you account for identical designators between multiple snippets and/or device sheets?	For device sheets, it's not a big issue because there is no corresponding PCB layout device sheet (unless you are trying to associate a PCB snippet to a device sheet… see below). We add the device sheet to the project and simply re-annotate the schematics. In situations in which we are trying to save a schematic snippet that matches a PCB snippet, we have to be strategic about providing designators that will not duplicate the existing designators in the project. This leads to the commentary in the prior questions about creating unique designators prior to making them snippets. Take a look at the following video for more information: http://www.altium.com/video-how-employ-snippets-design-reuse

What if you want to use two snippets in a single project that have the same designators, won't that mess up the linking?	Unfortunately, one should not just drop these snippets onto the schematic in rapid succession. Not only will the unique IDs not match, the snippets will have duplicate designators. Therefore, one will need to place the schematic snippet, followed by the PCB snippet. Relinking the schematic and PCB components would be achieved through the ECO process or Project >> Component Links. Then these designators can be changed to match the annotations of the project. Once this is done, the next snippet can be placed. Take a look at the following video for more information: http://www.altium.com/video-how-employ-snippets-design-reuse
When working with a "standard" multi-connector arrangement (ala the Arduino in your example), other than a snippet is there any way to create something akin to a part (aka library component multimodule) that contains the schematic connections, footprint, and 3D model?	From snippet point of view, the answer is no. The snippet is simply a copy of the primitives as they are. Unfortunately, there are no additional parameters that can be declared so that multiple snippets are associated to each other.
Is there a method of sequencing reference designators by their particular component type say 1K resistors are sequenced as R1-R36, the 2K resistors as R37-R49, the 3Ks as R50-R62.....?	Sequencing of this nature could only be done if one was selecting/filtering these components in the annotation dialog. The automated features of the annotation dialog are usually based upon the sheet which the component resides, whether it was selected for change, and its location on a particular sheet. If it was a multi-part device, parameters could be used to order those submodules. Otherwise, there is no method provided for ordering based on parameter value.

Is there a way to attach a layout snippet to a device sheet, or are they schematic only?	There isn't a specific feature to associate a PCB snippet to a device sheet; however, one could still have a device sheet and a PCB snippet that goes along with the device sheet. Caution would have to be taken to handle the unique IDs and designators. For example, the moment that the device sheet is used, the PCB snippet would be added to the PCB, followed by the process to reconnect to associate the symbols with their corresponding footprints.
Is there a "refresh" function for device sheets in your project if the original/saved version does need edits?	One can either: 1) Pull the device sheet into a project, refactor it into a schematic sheet, make the changes, and then refactor it into a device sheet with the same name. It will overwrite the original device sheet. 2) Simply navigate to the .schdoc in the device sheet folder, open it in Altium Designer, make changes and then save. Note that if other projects used this sheet, they may be impacted.