

DFM Webinar (Part 3) June 2015

Questions and Comments from DFM Series – Library Considerations

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

Comment / Question	Response
<p>Is the Library available? Is the recording available? Are the slides available?</p>	<p>Yes! Request them via http://ninedotconnects.com/webinar-request-dfm-part3.</p>
<p>I setup a project template a few years ago and it was one of the best things I've done for standardizing our source documents</p>	<p>You are a rarity! The fact of the matter is that it is hard to justify the effort to a management, especially when projects need to be finished. In the end, the efficiency of standardization is far better than everyone recreating the wheel.</p>
<p>Regarding Metacharacters</p>	<p>To summarize: use them sparingly and judiciously.</p>
<p>What about footprints for different assembly technologies (like wave vs reflow)?</p>	<p>There are various processes which will require different footprints based on the physics of the process. Just decide which ones you need and take care building them. The goal needs to be to make <u>all</u> processes optimized by the footprints. Not just assembly.</p>
<p>When setting the components on PCB, what does the green and red color meaning when moving the component to a spot?</p>	<p>Those are automated placement guides based on shortest connection lengths. The official name is the "Placement vector." During our Altium support days (this being Paul Taubman and Jeff Condit), we jokingly called them the "Altium gummies." In the forums, they are referred to as "jellyfish."</p> <p>According to a forum post by the legendary Harry Selfridge (RIP), it is a holdover from Protel 99SE. At this point, there does not seem to be any way of setting the distance or disabling it.</p>

<p>How do you put fiducials on PCB?</p>	<p>Fiducials can be placed by: 1) placing pads directly on the PCB, 2) Placing Fiducial footprints on the PCB, or 3) Placing No-BOM fiducial symbols on the schematic which automatically bring fiducial footprints onto the PCB. The latter is recommended for reusability and simplicity. Fiducials have rules about location, size, clearance on the same layer, clearance of other layers, lack of solder mask, sides, proximity to components, proximity to corners, placement on panels if paneling, etc. Though IPC has specific requirements and recommendations, know that, at a minimum, 2 fiducials, placed on opposite corners of the board help the assembly house tremendously.</p>
<p>When placing special items on the schematic (like RTV glue); is there a recommendation on how to deal with quantities?</p>	<p>Are you referring to the amount in each location on the board, or the quantity in ounces for a BOM, etc.? Regarding the BOM, the quantity column is tabulated by the number of 'components' of the same type, so generally 'no'. However, you can post edit the BOM to either set an amount of change it to A/R. The major benefit is that it is not forgotten.</p>
<p>Can you post the IPC standard for footprints?</p>	<p>The link: http://landpatterns.ipc.org/IPC-7351BNamingConvention.pdf Please note - This is only a naming convention document. If you are seeking other requirements, please contact the IPC for more information.</p>
<p>Unlike the mechanical 3D layer, our mechanical courtyard layer does not switch between top and bottom with the component. Consequently we can't discern overlap between components on the Altium PCB Document. How can you flip the courtyard layer with the component?</p>	<p>Merely set up the layer pairing between two component courtyard layers in the PCB, the same as you would with assembly layers. For example the Top Courtyards could be on layer Mech-15 while the Bottom Courtyards are on layer Mech-16. If layers Mech-15 and Mech-16 are layer-paired, then when you flip the component to the bottom, all the paired layers get flipped automatically.</p>

In the webinar, a recommendation was made to have the comments and the description be the same value. Please explain.

The reason why we opted to use the same information for both the description and the comment has to do with the fact that the comment parameter is the ONLY parameter of the component that is passed to the PCB in Altium Designer. Therefore, what we wish to do is provide the most detail to the PCB footprint. A well-formed description has a good deal of information in it.

Aside from the comment parameter, the description parameter is absolutely critical due to the fact that it summarizes the component in one cell of data. Consider the following - If a bill of materials is uploaded into a purchasing or accounting database, the only way one could quickly obtain the component's "essence" (i.e., voltage, tolerance, value, footprint, component type, etc.) would be through this description. Any other data in the record (like mfg. name and part number) would require us to search or piecemeal information together. Even if there was a link to a datasheet, one would still have to open the datasheet to obtain the "essence" of component.

(The word "essence" is being used for the lack of a better word; "characteristics" is a bit too deep. What we are trying to achieve is a description that immediately brings to mind the component. For example, if the description is "RES0402,10K,1%,1/4W,-55Cto85C", most designers can envision this part very quickly.)

Going back to our concept of "comment equals description", if the description can summarize the "essence" of the component, we want that summary to be passed along to footprint; otherwise, we will have to constantly flip back to the schematic for its details.

<p>We did not create a separate Assembly and Courtyard layer, what is the difference?</p>	<p>One of the advantages of separating the assembly from the courtyard is the ability to disable layers for visibility purposes in the layout. During placement, I could simply enable just the copper layer and the courtyard. That would be adequate for ensuring that my placements are reasonable from the get-go. It is true that Altium has a component clearance checker; however, courtyards provide a wonderful visual that goes beyond a minimal clearance rule. In addition, it gives some flexibility to enable/disable layers when creating PDFs of the PCB.</p> <p>Second, we tend to draw the assembly layer with an outline of the pads rather than a courtyard. This is to assist in component orientation. In addition, we will add dot to mark pin 1 on the assembly layer.</p>
<p>Does the assembly layer require a designator?</p>	<p>There is supposed to be a designator on the assembly layer, per IPC standard 7351B. The designator on the assembly layer is can be implemented by adding the text string ".designator" in Altium to that layer in the library. You do not have to use .designator for the silk; Altium will automatically do this upon ECO from the schematic to the PCB.</p>
<p>Should we create a separate Courtyard layer?</p>	<p>If you have a setup that works, then the answer is "no"; however, if the folks in fab, assembly, or test are asking for it or if you believe that it will in any way remove ambiguity or improve your process, then you should consider it. It is more important that you have it; as for which layer, that's a technicality.</p> <p>Keep in mind that the IPC specifications are there to assist in making the handoff between you and the manufacturer as smooth as possible. Not adhering to the IPC standards does mean that the board can't be manufactured; it's there to minimize the turn time, cost, hassles, and possibly respins.</p>