

## In Training

M. Simon



I'm starting to get a feel for [Altium Designer](#) [1]. After a few hours with ignorance painted all over my face I have started to find my way. What I found most helpful was their [tutorial page](#) [2] that has their manual sectioned in 20 pdfs plus an index pdf.

I have opened a schematic page and things are starting to get very slick. I hit the help button, typed in a topic - "title block" in this case and a page opened right in the editor and my schematic is right next to it. So I can go back and forth between the working page and the help page in one application. Slick. Very slick.

So with your schematic open you go to the **Design Button** on the main page. Click on **Document Options** and then **Preferences**. Now just fill out the form and you have a title block. Unfortunately I had a problem. I couldn't reach the "Title" bit for the document. No worries. I have plenty of other stuff to learn. I did send in a bug report to Altium and they got back to me Monday. Unfortunately I was not looking at my e-mail when it came in and was too late to get any help from them Monday night - I do keep programmers hours so the mismatch was kind of inevitable. Their rep (Dave - just to give him a name) got back to me Tuesday morning and we went through the drill. He took over my computer (with permission) and we started to replicate the problem. And we did. Then he does some checking with my Windows set up and told me that my "hard of seeing small print" adjustment to Windows was messing things up. Given that I'm on a strict budget (the monitor I use was donated to me) buying a bigger monitor was not an option. So on his advice I reset the "hard of seeing small print" gadget and what do you know? That fixed it.

He then asked me if I had any other questions and I said yes. How do you move a component or a block of components and keep the wires intact? Just hold down the [ctrl] key while doing the move with your trackball he said. Easy. I then told him that I had some other questions and would find the answers on my own unless I got really stuck. He did suggest the **Help** button at the bottom of the page which gives a list of Commands and how to command them. And he also said (because I asked about it) that for certain commands (like **Place Wire**) there was a continuously scrolling list of available commands at the bottom of the page like "X" for x axis

## In Training

Published on Electronic Component News (<http://www.ecnmag.com>)

---

mirroring a component. "Y" for mirroring the y axis. "Space Bar" for rotation. etc.

This is getting good. Commands like those available for [TinyCAD](#) [3] and [ExpressPCB](#) [4] and [FreePCB](#) [5] are all available for [Altium](#) [1]. They are just done differently.

I did tell Dave that as usual the [Altium](#) [1] customer service was excellent and that I was a fan. I did hasten to add that because of the integrated environment that I wouldn't consider using the hobby packages mentioned above for commercial development even if Altium didn't have all the features I wanted. Well it has the features I wanted and it keeps track of all the data required for production including supplier part numbers. Way cool.

Let me add that using the arrow keys and page up/down keys to move around and zoom in and out of the schematic is a real pleasure. I do wish that when moving a block the block didn't go invisible. But it does become visible when you stop moving it for a second, so that is a very minor quibble.

Well, the schematic is done (including some parts I had to make which were not in the libraries provided) and I will soon be off doing a PCB layout. Which will include more makings (footprints). I look forward to it now that I know my way around a little better. Plus I know I have the gang at **Altium** covering my six. Very comforting that.

All that took from late Thursday evening to Tuesday morning - start to finish. Not too bad for a brand new and command intensive application. My next report will be on their PCB layout section.

M. Simon's e-mail can be found on the sidebar at [Space-Time Productions](#) [6]

**Source URL (retrieved on 10/24/2014 - 9:51pm):**

<http://www.ecnmag.com/blogs/2011/12/training>

### Links:

[1] <http://www.altium.com/>

[2] <http://www.altium.com/training/en/manuals-and-downloads.cfm>

[3] <http://sourceforge.net/apps/mediawiki/tinycad/index.php?title=TinyCAD>

[4] <http://www.expresspcb.com/>

[5] <http://www.freepcb.com/>

[6] <http://spacetimepro.blogspot.com/>