Quick Notes for Altium6. 30 October, 2007

First View:

1. When you open the program, it will open in the same configuration as was used during your last session. This means that if you were editing a schematic when you last used the program, the same schematic will reappear.
2. The left pane in your screen will have the "*Projects*" tab that allows you to see the current schematics that you are working with.

Starting a new or Opening and existing Schematic:

1. To start a new schematic, go under the "*File*" menu and select "*New*", "*Schematic*". A default name schematic will appear on the screen and also in the "*Projects*" browser. You should rename and save this schematic immediately by doing a "*Save As*" under the "*File*" menu.
2. To open an existing schematic, go under the "*File*" menu then "*Open*" and shop for your file.

Accessing Libraries:

1. On the right margin, you should see the vertical label "*Libraries*". Clicking on this will temporarily open the library browser window. The tiny down arrow, underneath the "*Place*" button, will allow you to select a library that has been previously loaded.
2. You should keep your libraries docked on the left, but if, for some reason, the "*Libraries*" button is no longer visible, you can reopen the libraries browser window by selecting it under the "*System*" tab in the bottom margin of the screen.
3. In the Libraries window, there is a button "*Libraries…*" that allows you to load or unload new libraries.

Working With Templates:

1. To change the current template, go under the "*Design*", "*Template*", "Set Template File Name …" and shop for the desired template. Usually, the smaller the template size, the more readable the final schematic will be.
2. To change the parameters on a template, such as the engineer name, title, revision etc., left-click in the schematic in a blank spot, then right-click to access the "Options", "Document Parameters …" menu. From this menu, you will be able to change all the fancy text on your template.

Creating and Modifying Libraries

1. The first thing you are going to want to do is create your own library. Pick a name that makes sense to you like "myparts" or something. Go under the "*File*" menu and select "*New*", "*Library*", "*Schematic Library*". It's important to recognize that you only do this one time, don't create a library for each part you want to use; the library will be a collection of parts that you create. Immediately, save your new library in your library folder using the name you have selected.
2. Notice that once you have created your library, a tab has been created in the lower left portion of the screen named "*SCH Library*". Click on this tab and the left

pane of your screen will reflect details of your new library. Remember, you can go back to the familiar "Projects" view by clicking that tab in the lower left of your screen. Try it to cement the concept.

3. To add a component to your new library, select "*Tools*", "*New Component*" on the top menu bar. When prompted, give a name for your component like "cool thing" or something. It's important to understand the difference between a component and a part. A component could be a single IC chip for instance, whereas a part is a section of a component. Examples of parts would be: One section of a quad op-amp, or one portion of a dual stacked connector. Parts are a more advanced concept and won't be necessary initially.
4. Now, let's build the part. First, we'll put down a pin. From the top menu bar, select "*Place*", "*Pin*", remember to hit the "Tab" key so you can see the details of the pin. You can also access the pin details by double-clicking on the pin itself after you place it.
5. Place a pin and then double-click on it. The *Pin Properties* window will open. Ensure, for now, that the *Electrical Type* is set to *Passive*. Set the Display Name to some text like, "my input" or such. Set the Designator to the number "1". Choose a length of 20 or so (it's important that the length be some even multiple of 10). Hit "OK" to close the dialog box and see what you have.
6. Next we are going to put a box around the pin so make what looks like a nice component. Place a rectangle by selecting "Place", "Rectangle" and cover the text "my input" leaving the pin sticking out the side. You will see that the text has been covered by the box. To fix this type "m" and select "*Send to Back*", then click on the rectangle.
7. Finally, you will set a few default parameters on your new part by going to "*Tools*", "*Component Properties*". To set the Default Designator, use a capital letter followed by the question mark i.e. "C?" for a capacitor. This allows the program to automatically set the designator later on. Put in a comment to aid in identifying the function of your component. Hit OK, and save the library.
8. Now, when you load this library in the schematic editor, you should be able to place the part on your sheet.

Here are some useful things to type and play with. Use the *right-click* mouse button often when you get confused.

1. Lines versus Wires: There is a difference between a Line and a Wire in Altium. A Wire conveys some electrical connection meaning, whereas a Line is just a pretty picture. The bottom line here is: Use Wires!
2. Moving and Dragging – The difference between moving and dragging is that dragging rubber-bands the attached wires, whereas moving will detach connections. Moving a single object can be done by clicking and holding the left mouse button on the object you want to move. To move multiple objects, or to drag an object, you should first select it (by clicking on it, or dragging a box around it), then under the "*Edit*" menu, select "*Move*", and chose "*Move Selection*" or "*Drag Selection*" as you desire. Another way to quickly access the

move or drag menu is to select an object, then type “m”. This will open the same menu for you. Many commands have this type of shortcut.

3. Deleting – Selecting objects by dragging or clicking will allow you to use the “Delete” key on your keyboard to remove items from your schematic.
4. Copy/Paste – These functions adhere to the windows standard methods of ctrl-c, ctrl-v, etc.
5. While placing new parts, or just about any function that has something attached to the end of the cursor, you can hit the “Tab” key on the keyboard to modify the entity. Try it while placing a part, for example.
6. Double-clicking on many things will open up option dialog boxes that allow you to modify parameters.
7. To place text on the sheet, choose “Place”, “Text String” or type “pt” on the keyboard. With the text stuck on the end of the cursor, hit the “Tab” key and modify the text string.
8. Zooming is easy if you remember to hit the “z” key then choose from the menu options that pop up. Remember that holding the right-mouse button down on the schematic allows you to pan around nicely.