

“Navigating Altium Designer” Webinar

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

Comment / Question	Response
<p>What’s the shortcut for viewing zero rotation and orthogonal rotation?</p>	<p>As a side note – this question is in reference to the 3D viewing capability in Altium.</p> <p>For the zero degree rotation, press the zero ‘0’ hot key.</p> <p>For the 90 degree rotation, press the ‘9’ hot key.</p> <p>Both the zero and 90 degree rotations are with respect to the X Y plane.</p> <p>For the orthogonal rotation, press the ‘8’ key (note that the orthogonal view was not shown during the presentation)</p> <p>The orthogonal rotation is with respect to the Z axis.</p> <p>Note that these viewing commands are also listed in the View menu when in the PCB 3D view.</p>
<p>From a component in a schematic, what is the fastest way to edit that component in whatever library it is in?</p>	<p>To answer the immediate question, you will need to open the properties of the component to determine library name and component name. From there, you will have to open the library for editing using File » Open.</p> <p>Altium does not have a feature that allows you to “jump” from a selected component in the schematic to its representation in the library. This is for two reasons:</p> <ol style="list-style-type: none"> <li>1. Altium uses a static library methodology. When a component is placed, it is copied to the schematic file. It retains information about its origin; however, there is no active link.</li> <li>2. Altium has a number of different library types and some of which would make it extremely difficult to perform this operation. For example, if a company was using a third party database, Altium would not have the ability to open the database and jump to the record (i.e., the component) of interest. Scripts could be written to do it; however, it’s not a trivial task.</li> </ol>
<p>What is the difference between compile and recompile in the project menu?</p>	<p>When using the Compile command, Altium completely rebuilds the netlist from scratch. Recompile simply updates those items that have been changed. In past,</p>

	<p>given the processing speed, a recompile would be quicker than a compile; however, with the speeds of the processors in the last 15 years, it is very difficult to see the difference between recompile or compile unless you are dealing with a huge design of 50 or more pages with a complex hierarchy.</p>
<p>How would I control visibility of different text types. For instance, top and bottom reference designators?</p>	<p>Recently, Altium added a transparency tab to the View Configuration dialog in the PCB editor. This dialog can be accessed using the 'L' hot key. This allows the user to control the transparency based on primitives (such as the text) on any particular layer that exists in the PCB.</p>
<p>Can you explain or expound on having power ports and global power symbols, and how they interact in hierarchical designs?</p>	<p>Power ports are special primitives in the schematic editor. They are global in nature, regardless of whether one uses flat or hierarchical design. The name given to the power port is the most important aspect. It is the name (not the symbol picture) that the netlist contains.</p> <p>Where there may be points of confusion lie in hierarchical designs. By default, one does not have to show the power ports in sheet symbols; however, for better readability, it is advised that the power be added.</p> <p>If one is going to show power ports going to and from their sheet symbols in the hierarchical designer, they should be aware of a feature added to Altium that allows for localized power ports. This is found in the project » project options dialog, options tab. The following comes from Altium tech docs:</p> <p><b>Power Port Names Take Priority</b> - The software has the ability to localize a global power net, by wiring a power port to a normal port. This would force all pins on that sheet, connected to that power port, to be in a separate net. Turning on this option would force it to use the net name on the Power Port.</p>
<p>How do you measure distance on a PCB between two spots?</p>	<p>In both the schematic and PCB editors, you can use Ctrl-M. This will add a crosshair to your cursor. Click once for the first point, then drag your cursor to the second, and click on the second point. A dialog box will provide the total distance, along with the X and Y distances if your 2 points are not perfectly horizontal or vertical.</p>
<p>Is it Ok for each engineer in an organization to have their own repository for SVN?</p>	<p>It depends on where the final source files will be located. If each designer does their own thing and maintains their own files including the final manufacturing files, this will be</p>

	<p>problematic, given that no one else can see what they are doing. If they leave the company, good luck trying to find the files.</p> <p>Therefore, there needs to be some master repository for all of the final art work. Whether this is done in SVN or another version control tool is up to your company. The key thing is to be able to find the artwork.</p> <p>If your company does have a procedure for final artwork being stored and assuming that no other individuals need access to the work in progress, then it would be a good thing to have a personal repository. By the way, there is nothing that prevents a designer from creating their own repositories on their computer or user space on the network.</p>
What buttons go left and right like page and page down?	<p>PageUp and PageDown are used to zoom in and out. To pan, you have several options:</p> <ol style="list-style-type: none"><li>1. Right mouse click and drag the mouse</li><li>2. Use the scroll bars on the bottom and right side of the editor window</li><li>3. Use the arrow keys on your keyboard. In addition, by holding the SHIFT key, the arrow keys will allow for faster panning.</li></ol>