CircuitStudio FAQs

1. **Is there an offline installer available for CircuitStudio?**

   No, we only support the online installer.

2. **Firewall is blocking CircuitStudio from installing. Can you please advise what port is the software uploading from?**

   It uses SOAP over HTTP (port 80). If the firewall or proxy is blocking it, it's most likely because SOAP is blocked as a protocol. Try Downloading the software from a different computer that is not blocked by firewall.

3. **I am having difficulties with library management: I can get parts from the vault or from another library on the schematic, but I could not find a way to save this component to another library, nor how to create a local schlib from the schematic. Can you help?**

   There's an easy way to do this - place a part from the vault in a schematic, select it and copy (CTRL+C) then in the schematic library, right-click in the library panel's list of components and choose PASTE (or select that panel and use CTRL+V) it will copy the part into your own library.
   You can follow the same method for acquiring footprints from any design - including those imported from other tools.

4. **Could you please tell me if it is possible to create a circular PCB in Circuit Studio and how to do this?**

   It is possible to make circular PCB outlines, using SHIFT + SPACE to change corner mode while re-defining board shape.

5. **Are delay Rules/ Line Matching Supported in CircuitStudio?**

   Length tuning can be done while routing. However there are no design rules for controlling the length - that is done by the user with the routed lengths displayed in the PCB Panel

6. **Can I use blind/buried vias in CS? Where can I find this feature in CS?**
   Blind/ Buried vias in CS are supported.

   In order to use blind, buried or build-up type vias, the Drill pairs must be configured, with a drill pair for each layer-pair that a via spans.. This can be done by using the Drill Pair manager which can be found under the Layer stack manager. See some screen shots below of where the drill pair manager is found. Go to home tab then click on layer stack manager→Once in Layer stack click on Drill pairs → This will open Drill manager where you can configure

7. **How do you convert special strings in Circuit Studio? All the PcbDoc's I need to convert all have special strings.**

   The option to convert special strings is found in the View Options tab of the View Configurations dialog. The View Configurations dialog is accessed from View>>Switch To 3D>>View Configurations, the L key on the keyboard or the Current Layer button to the left of the layer tabs in the PCB editor.

   http://techdocs.altium.com/display/ADRR/PCB_Dlg-ViewConfigurations_ViewOptions((View+Options))_AD

8. **Does CircuitStudio support embedding embedding STEP models into the PCB design?**

   Yes, you can embed the .STEP file model into a new footprint and place it into the PCB design.

9. **What is the best way to do line matching in CircuitStudio?**

   Display the nets in the PCB panel and route them to the same length. The length calculation is displayed in real time.

10. **How can I move an area of components and traces?**

    Go to preferences in PCB and set "Comp Drag" from "none" to "Selected Tracks", then use the Tools ribbon and Move -> Drag.

    If that doesn’t work you can also try the command Tools | Move » Move Selection.

    http://documentation.circuitstudio.com/sites/default/files/wiki_attachments/240261/PCB_Cmd-Move_Composite_MoveMenu.png

11. **Does CS support a database library .DbLib format used in Altium designer?**

    CircuitStudio uses integrated libraries. Dblibs can be converted to integrated library's in Altium designer so they can be used in CircuitStudio

12. **In the 3D PCB mode is there any way to see other views of the board other than the 0deg, 90deg and orthogonal?**

    To move the board to any angle hold down the right shift key and a rotational icon will appear with the right mouse button rotate the pcb to the desired view.

13. **How do you make a surface mount footprint? All I see is an option for through-hole pads. Is there an option for defining surface mount pads?**

    The SMD pad are created in the same way as through hole pads - just on the layer desired either top or bottom but they do not have a hole size. Place pad – hit the tab key and edit the hole size to 0 and layer from multi layer to top layer

14. **Is it possible to make a 3-wire junction? Currently when I try to connect more than 2 wires, the junction disappears and the new wire jumps over the other one. I tried using the manual junction, but it is unclear how this actually works.**

    It is possible to create a 3-wire junction by clicking directly on an already placed wire(not the end of a segment) when placing the start or end of another wire segment. Here are some links to Altium Designer techdocs about auto and manual junctions.

    http://techdocs.altium.com/display/ADRR/Sch_Obj-CompilerGeneratedJunction%28%28Compiler+Generated+Junction%29%29_AD

    http://techdocs.altium.com/display/ADRR/Sch_Obj-ManualJunction%28%28Manual+Junction%29%29_AD

15. **Where can I edit the properties of multiple pins?**

    The properties of multiple pins can be edited in a schlib using the SCHLIB Inspector, View | Schematic | Inspector.

16. **Track to pad/via connectivity is lost when moving components, I have to manually edit each track after a move**

    There is a preference (File » System Preferences » PCB Editor » General) in the "Other" region, set "Comp Drag" to "Connected Tracks." Then when editing, use the command, Tools | Arrange | Move » Drag.

17. **Please provide Keyboard shortcuts for regularly used features such as toggling via size or track width.**

    Min, Max, preferred via size is toggled with the 4 key or a favorite is set using SHIFT+V. Min, Max, preferred track width is toggled with the 3 key or a favorite is set using SHIFT+W. Press the Tilde key ~ while routing to see a list of available shortcut keys.

18. **Provide a way to make a mass edit of text size/style on the silkscreen or assembly layer. Currently they have to be edited one by one and I could not find any place where to define it globally.**

    Text on an assembly layer can quickly be selected using the PCB Filter(View | PCB | Filter), multiple selected items can be edited using the Object Inspector(View | PCB | Inspector).

19. **There is a step model I downloaded from a supplier. I can connect it to the PCB file, but how do you align it to the pads?**

    The step model can be dragged and positioned in the X and Y axis in the 2D view of the PCB Library editor, the Z axis is set in the properties dialog as Standoff Height

20. **Where can I add snap points?**

    Snap points can be inputted manually in the 3D body properties dialog.

21. **Is it possible to make the unrouted connections more visible or change their color?**

    It is possible to change the color of the rats nests unrouted connection lines for individual nets or a selection of nets using the Nets display of the PCB panel, View | PCB | PCB » Nets. Right-Click a net or a selection of nets in the PCB panel and choose 'Change Net Color'. Hovering over a unrouted connection line will make it more visible with live highlighting.

22. **Is it possible to refresh the connections and eliminate the zig-zags after moving the components?**

    Use the command Tools | Netlist | Netlist » Clean All Nets to refresh the connections and eliminate the zig-zags. If that doesn't work try:

    Select all the nets in the pcb panel then find the primitive PAD free1 in the primitive panel these are the items that need to be deleted. Highlight all the pads and then select them and then delete the object by clicking in the pcb window and pressing the delete key. Re run the design rule test and re sync the board with the schematic just in case you deleted any other objects in the pcb.

23. **In Circuit Studio the Document Parameter dialog lets you fill in text strings such as Title, Revision, etc. How do you get these to show up on the schematic title block?**

    Special strings are used to show the parameters. To create a special string, place a text string on the schematic or on the schematic template with a equals sign(=) before the parameter to to be displayed.

    Example special text strings for title and revision.

    =Title

    =Revision

24. **Is there any way to use CircuitStudio without a numpad? I have a desktop that I use, but my laptop doesn't have a dedicated numpad and I haven't been able to figure out how to switch layers during interactive routing without it**

    SHIFT + Cntrl + Mouse Wheel to change layers while interactive routing...

25. **After making a pcb library component and using it in a project how do you get the project to update the component footprint if the footprint in the library is changed?**

    Directly from the updated pcb library or from the project.If it is from the library just update the pcb using right click on the item. If from the project just push the updates after a compile to the pcb This should change the footprint

26. **I don’t find a way to re-annotate resdef from PCB positional. I wish to start top component resdes numbering from lower left side to higher right side. And bottom side numbering begin at 500 ex: R500, C500 It’s it possible to do that?”**

    In CS there is no tool do automate a ref des strategy. However, you can very much name the reference designators to whatever you like them to be on the PCB and push those changes back into the sch without the risk of losing the synchronization of the Sch and PCB. This is simply done by clicking on the update PCB action while you are in the PCB mode. This will take you through the standard ECO dialogue boxes to execute and document the changes. Once you’ve done that, everything will match up

27. After making a pcb library component and using it in a project how do you get the project to update the component footprint if the footprint in the library is changed?

This is only achieved as a "push" from the library editor vs a "pull" from the design. From the library editor open the PCB List panel. navigate to comp, edit it, and then right click on it from the list panel. From there you can update the design and it' should be very clear from there. ☺

28. Is there a way to add a clearance rule to a plane ? The rules apply properly to polygons (copper pour), but it does not seem to apply to planes.

The best way to go about it is to convert the plane layer to a normal layer and pour polygon's . You can apply rules to each polygon to get the desired clearance. It will be quicker and easier to control this way – Just allow for board edge pull back by setting up a keep out on the board edge.

29. In an open PCBdoc, use Search to locate a component. The component highlights and is selected, so I can move it to where I need it. But...then cannot be unselected. How do I unselect the component?

You will need to clear your mask. You can do this from the view menu and click on clear at the far right or simply pressing SHIFT + C. There is a video about this on the e14 pages under the tutorial videos. That particular video is called "Cross Probing with Circuit Studio" Here is a link http://www.element14.com/community/videos/view-video-lightbox.jspa?videoID=16331&lightbox[iframe]=true&lightbox[width]=520&lightbox[height]=400

30. Is there a way to automatically set down a via and change layers while manually routing?

Use CTRL + SHIFT + Mouse wheel will change layers. Also the * key works but it's on the numerical keypad (not SHIFT+8). If you are not getting a via it's because there's something in the way or their via size is too big.

31. **I'm trying to import a DXF file into CircuitStudio and it does not seem to be working properly. I do File->Import and I get a dialog that says "Done" and a new PCB window but I don't see any outline.**

    There are normally two main reasons why the DXF file doesn't import correctly.

    1- The import may have completed, but at a VERY small scale. Try Zooming in to see if the image then shows up. If it does you may have to readjust your scale settings. 2- You may need to uninstall/reinstall the installation of TeighaX, the installer is located at C:\Program Files (x86)\Altium\CS\System\Installation\TeighaX_Setup_3.9.0.msi. TeighaX is a 3rd party utility used during import of dxf/dwg. Setting the correct scale and installing/repairing TeighaX should get things working

32. **It is possible to create a panlized board array in CircuitStudio?**

    It is possible to create a panlized board array in CircuitStudio but this is completely manual process involving copying and pasting the schematics and PCB layout multiple times then re-annotating. The board fabricator may be able to create the panel in their CAM tools.

33. **Is there a way to print the schematics with the variant showing ?**

    The schematics can be printed with the not fitted graphics by selecting the correct variant for the project or schematic prints in Projects | Generate Outputs or by making the compiled tab the active document and going to Outputs | Print.

    http://techdocs.altium.com/display/ADRR/The+Editor+View+and+the+Compiled+Documents+View

34. **Wanted to get rid of "Net has no driving source" warnings. First clicked on Electrical Rule Check in the menu and changed the setting in the Error Reporting section. That did nothing. Then right-clicked on the project, clicked on Project Options, then changed the setting in the Error Reporting section. That worked. I would think both should be the same, but maybe not. If not what is the difference?**

    The Electrical Rule Check Setup dialog found under the Outputs menu allows the user to configure the Electrical Rules Validation Report. These settings are used during batch output generation but not compilation.

    The Project Options dialog allows the user to setup Project level Electrical Rule Checking during compilation. Although the Error Reporting and connection tabs are similar in both dialogs they do serve different purposes.

    http://documentation.circuitstudio.com/display/CSTU/WorkspaceManager_Dlg-ElectricalRulesCheckSetup((Electrical%20Rules%20Check%20Setup))_CS/?help-data=WorkspaceManager.Dlg.ERCReportSetup_Form.Button_OK

35. How do I Create a new component based on an existing library component? The schematic symbol I want is in the vault, and so is the PCB footprint, but I need to combine them from two different existing components to create a new component, then modify the pin names and save it in my library. I have tried to find a way to do this and can't. Does it have something to do with the Clone operation?

Yes, you will need to perform a clone on the symbol and the footprint then copy them to your library. Navigate to the component that contains the symbol or footprint in the Content Vault Set the bottom right panel to preview Double click the symbol or footprint in the listbox to jump to the models folder in the upper right pane Right-click the model in the upper right pane and select Operations»Clone <model name> A new library containing the symbol or footprint will appear under Free Documents in the projects panel Right-click the component in the SCH or PCB library panel and select copy Open the PCB or SCH library to add the component to Right-Click in the SCH or PCB library and paste the component Repeat for both the symbol and footprint Link the footprint to the symbol in the symbols properties and make any modifications