Boot Camp-Special Ops Challenge Quiz

1. What's the critical difference between a panel and dialog window?

    a. There is none
    b. Panels must be closed to continue editing, whereas dialogs can be left open
    c. Dialogs must be closed to continue editing, whereas panels can be left open

    Reason: An open dialog box will prevent the user from performing any other edits within the tool until the dialog box is dismissed by pressing OK or Cancel.

2. What is the recommended grid size for the schematic symbols?

    a. 10 Millimeters
    b. 10 Mils
    c. 10 Inches
    d. 100 Mils

    Reason: Although schematics are, by nature, dimensionless, Altium Designer uses imperial and metric 'dimensions' for the schematic and schematic library editors. They can be thought of as relative distances so that grids can be consistent in both editors. Furthermore, most of the Altium and third-party provided footprints are created with a 100 mil grid; therefore, it is best to use the 100mil grid size for any new symbols that need to be created.

3. Which of the following provides settings for ALL projects opened within the tool?

    a. Tools>>Preferences
    b. .PrjPCB file
    c. Project >> Options menu command
    d. Tools >> Document Options (This now only exists in the Schematic Library editor; otherwise it's covered in the Properties panel. Not sure how to word that.)

    Reason: Tools >> Preferences impact all of the files that are opened. They can be thought of as global settings.

4. The query language is only used for design rules in the PCB editor.

    a. True
    b. False

    Reason: Though commonly used in the PCB editor for design rules, the query language is also available in the schematic library, schematic editor, and PCB library editor. It can be used for the global selection of primitives.

5. The only electrical primitive provided in the schematic symbol library editor that is not provided in the schematic editor is the:

   a. Component
   b. Line
   c. Pin
   d. Pad

   Reason: Altium does not want schematic symbols to be made outside of the schematic library editor. Not providing the pin primitive in the schematic editor, makes it impossible to create components in the schematic editor.

6. In the schematic editor, pins can be moved or deleted:

   a. True
   b. False

   Reason: Altium recognizes that pins need to be moved around in the schematic editor to better group them based on function, especially on large IC components. These pins can be deleted (though not recommended), but pins cannot be added.

7. An Integrated Library is required for a project.

   a. True
   b. False

   Reason: Altium can place a component on a schematic from a symbol library, which will link to the PCB footprint. This link can be used by Altium to retrieve the necessary footprint directly from the PCB footprint library. The advantage of an integrated library is to keep both libraries together in one file. In addition, the integrated library is in binary format, which at one time could be read faster than the ASCII-based symbol and footprint files. However, with the speed of today's computers, this is not a major sell point.

8. A project must have a dedicated library.

   a. True
   b. False

   Reason: Though a project library should be made at the end of the project to preserve a copy of the components of the project, the libraries can come from any location that Altium has permission to view.

The optimal solution is to use Altium's installed library option, which always looks for specific libraries, regardless of which project is opened.

9. One can create a schematic and PCB library from a project's schematics and PCB layout, respectively.

   a. True
   b. False

   Reason: Since Altium copies the symbol and footprint into their respective files, Altium can copy these symbols and footprints into their respective libraries.

10. The user can add a property to a primitive.

    a. True
    b. False

    Reason: Though the user might be able to add a parameter to a primitive, Altium does not allow a user to add or remove any properties of the primitive.

11. A component parameter consists of:

    a. A name and a numerical value
    b. 2 numerical values
    c. Name and Value

    Reason: That's all a parameter is in Altium – the name of the parameter and the value of the parameter.

12. The place menu items of the PCB editor are the same as the schematic editor.

    a. True
    b. False

    Reason: In the schematic editor, the place menu consists of logical connection primitives such as wires, ports, and net labels. The PCB editor place menu consists of physical connection primitives such as copper traces, polygon pours, and vias.

13. When a filter is applied in any of the editors, which of the following can be used to remove the filter every time?

    a. Simply clicking on the editor window
    b. Using the Shift+C shortcut
    c. Hitting the Escape key

    Reason: When a select or highlight is done in which the user requests Altium to mask out items that are not selected, the user generally has to "clear" the mask. Thus, using the Shift+C shortcut or the Clear Current Filter icon  *informs Altium to remove the masking.*

14. If one links a 3D model directly to the schematic symbol, it can be viewed in the PCB editor by pressing the 3 key.

    a. True
    b. False

    Reason: Associating a 3D model to the symbol was the method Altium employed prior to AD6.8 in 2007. In AD6.8 and thereafter, 3D models are associated with the PCB footprint to ensure that their orientation matches the drawn footprint.

15. One can create a polygon pour in the PCB library editor.

    a. True
    b. False

    Reason: Polygon pours require clearance rules which cannot be defined in the library. If a polygon pour is copied from the PCB Editor to the PCB Library, Altium will convert it to a fixed region.

16. What workspace panel is used to place components onto the schematics?

    a. Projects Panel
    b. Navigator Panel
    c. Components Panel
    d. Storage Manager Panel

    Reason: The Projects panel has to do with the opening and association of files to their projects. The Navigator panel is simply used to navigate the schematic. The Storage Manager panel is another way of looking at the files of a project. In short, none of these other panels have anything to do with component placement.

17. The electrical grid is …

    a. The same as the snap grid
    b. The distance before the cursor snaps to a wire or pin hot spot
    c. The grid that is seen when one zooms in

    Reason: This "electrical grid" is a bit of a misnomer. It is a "vortex." When the cursor comes within the boundaries of the electrical grid, the cursor is immediately pulled to the closest hot spot or center of the symbol or footprint, depending on the DXP preferences.

18. Which of the following statements about the schematic template is true?

    a. The schematic template must use the features in Tools >> Document Options
    b. The schematic template is saved in Project >> Project Options
    c. The schematic template can be graphically edited at any time in the project schematic
    d. The schematic template has the extension .schdot

    Reason: The schematic template acts as an overlay to the schematic. It cannot be edited as an overlay. The template can be created with non-electrical primitives found in the schematic editor. As for default templates, this is established in the Tools >> Preferences.

19. One can have different designators in the schematic and PCB for the same component.

    a. True
    b. False

    Reason: Altium allows this primarily due to multi-channel design. The .annotation file is created by Altium to track the different designators being used.

20. The workspace panel used in the schematic, PCB, schematic library, and PCB library to perform global editing is/are:

    a. Properties Panel
    b. Filter Panel
    c. PCB Panel
    d. List Panel
    e. Properties Panel and List Panel

    Reason: Both the Properties and List panels can be used for the purposes of global editing. The PCB and Filter panels are only used to search. The PCB panel is only available in the PCB editor.

21. The workspace panel used in the schematic, PCB, schematic library and PCB library to globally select components is/are:

    a. Properties Panel
    b. Filter Panel
    c. List Panel
    d. Filter Panel and List Panel

    Reason: Both the Filter and List panels can be used to globally select primitives.

22. Which workspace panel allows for smart grid paste and smart grid insert?

    a. Properties Panel
    b. Filter Panel
    c. List Panel

    Reason: The smart grid paste and insert are based on importing tables from a PDF or spreadsheet, which is a similar structure to the list panel.

23. In the PCB editor, which default hotkey is used to open the View Configuration panel?

    a. L
    b. M
    c. C
    d. V

    Reason: The L key is heavily used by Altium users to turn on and turn off various layers in the PCB layout

24. To add a plane or signal layer, which tool is used in the PCB editor?

    a. View Configuration
    b. Layer Stack Manager
    c. PCB Panel

    Reason: The Layer Stack Manager is used to add a layer to an existing PCB. Note that a footprint can also add a layer by declaring it in the library; however, this is generally a bad practice since it forces this layer on any PCB board that wishes to use the component.

25. What file type can be generated and provided to the fabricator?

    a. Gerber
    b. Gerber 2X
    c. ODB++
    d. IPC-2581
    e. Any of the above

    Reason: Over time, each of the formats has been made available. Please discuss with your fabricator their preferred format.

26. A union of devices can only be created in the PCB Editor.

    a. True
    b. False

    Reason: A union can be created in either editor.

27. If a mechanical layer is used in a footprint, that mechanical layer will be enabled in the PCB editor if that footprint has been placed.

    a. True
    b. False

    Reason: When a footprint is placed into the PCB Editor, Altium Designer places all layers that are associated with a component.

28. If a design rule is syntactically incorrect, what should you NOT do?

    a. Attempt to fix it
    b. Disable it
    c. Fix it when you can get back to it

    Reason: The online DRC checked goes into action every time there is an edit. If there is an issue with one of the rules, the DRC check will constantly generate an error message. Therefore, if the PCB Rules and Constraints Editor dialog box is warning of an error, it is best to either disable it or try to correct it.

29. Suppose there are several rules established for the electrical clearance, each with different scopes. If a net name shows up in two or more scopes for a particular rule, Altium Designer will

    a. Test each rule listed against the net
    b. Only test the net with the highest priority rule
    c. Only test the net with the rule containing the scope and has the highest priority of all other rules that may have the same scope.

    Reason: For each rule, only one rule can be applied for each instance in the PCB editor. If there are three rules pertaining to the same net, the rule with the highest priority will be the rule used to evaluate the instance for a pass/fail status.

30. Suppose there are several rules established for the electrical clearance, each with different scopes. If a scope that covers all primitives (ALL-ALL rule) is given priority 1, all the other rules will be ignored.

    a. True
    b. False

    Reason: Each instance where a rule can be applied is only tested by one rule. Therefore, a rule that covers all instances will always be applied. The other rules will never be used.