PCB Foundations Quiz
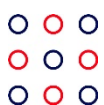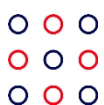
1. For non-single layer boards, PCBs should be fabricated with _____ layers.

   a. An Odd Number of
   b. An Even Number of
   c. Any Number of

2. Which of the following is not taken into consideration when calculating the characteristic impedance for each layer?

   a. Trace width
   b. Trace height
   c. Trace length
   d. Distance of the trace to a plane

3. The declaration of vias-in-pads is determined in the

   a. Assembly Notes
   b. Fabrication Notes
   c. Fabrication Files (ODB++, Gerber, IPC 2582, NC drill)
   d. Via stack up
   e. Layer Stack up

4. Select the answer that is true: Mechanical layers…

   a. Are unique to each project
   b. Are only declared in the PCB editor during layout
   c. Follow an IPC standard
   d. Should be consistently used in the footprint libraries for courtyards and assembly files

5. Silk Screens

   a. Are not required for every component
   b. Are highly recommended for prototype boards
   c. Should be kept to a minimal for production boards
   d. All of the above

6. The use of 3D component models in the PCB layout can assist with the

   a. Electrical clearance rule
   b. Component clearance rule
   c. Silk clearance rule
   d. All the above
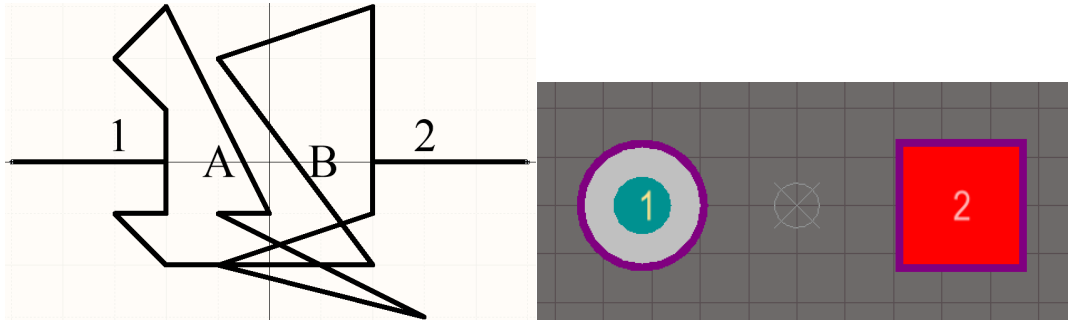
Leader in EDA and PCB training and service
info@ninedotconnects.com   (214) 699-6966
ninedotconnects.com/training

7. True or False – The solder mask has an impact on the trace impedance.

    a. True
    b. False

8. When writing a requirement, the following word should be used to denote a requirement:

    a. Should
    b. Would
    c. Can
    d. Shall
    e. May

9. When writing a requirement

    a. The use of the word "and" is okay
    b. The use of the word "or" is okay
    c. Both of the above
    d. None of the above

10. A well written requirement

    a. Has a complex sentence structure
    b. Uses technical verbiage
    c. Is testable by its nature
    d. Needs to be vague
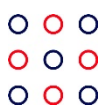
Leader in EDA and PCB training and service
info@ninedotconnects.com   (214) 699-6966
ninedotconnects.com/training

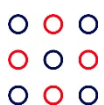11. Consider the following symbol and corresponding footprint:

The following symbol and corresponding footprint

    a. Are recognized by the EDA tool as a valid component
    b. Would be flagged by the Electrical Rule checker in the schematic tool
    c. Would be flagged by the Design Rule checker in the PCB editor
    d. Both B and C

12. When is the earliest time one should communicate the bill of materials?

    a. After fabrication has been started
    b. After the boards have been fabricated
    c. When the schematics are completed
    d. After the layout is complete

13. What is the best method to provide a unique identifier for each component in the library?

    a. Use the manufacturer's part number
    b. Use the vendor's part number
    c. Use a part number that is assign by your company
    d. Use a part number that you make up as you go

14. During post assembly testing, it was found that a latch on one of the connectors could not be fully extended due to other components located nearby. What would have prevented this situation from occurring?

    a. The use of a 3D component
    b. The use of a 3D component and component clearance rule
    c. The use of a courtyard
    d. The use of a courtyard and a component clearance rule

Leader in EDA and PCB training and service
info@ninedotconnects.com   (214) 699-6966
ninedotconnects.com/training

15. Though schematic tools do have to use a grid for pin connectivity, in theory, the grid used in a schematic is

    a. In metric (mm)
    b. In imperial (mils)
    c. Both A and B interchangeably
    d. Dimensionless

16. The biggest factor when it comes to characteristic impedance is

    a. The clock speed of the board crystal
    b. The integrity of the signal from the output buffer of the sending component
    c. The voltage level
    d. The physical aspects of the board

17. True or False: Microvias are the same as a one-layer blind or buried vias

    a. True
    b. False

18. Which of the following drill document(s) must be provided to the fabricator to program their drilling equipment?

    a. Drill Plot
    b. Drill Drawing
    c. NC Drill file
    d. All of the above

19. True or False: ODB++ and Gerbers are proprietary formats

    a. True
    b. False

20. The solder paste files are generally produced

    a. In a fabrication file format
    b. Through a manual drawing
    c. As part of the bill of materials

Leader in EDA and PCB training and service
info@ninedotconnects.com   (214) 699-6966
ninedotconnects.com/training

PCB Foundations Quiz

Do you want to know how you did on the quiz? Click below for the answers:

Click Me

Leader in EDA and PCB training and service
info@ninedotconnects.com   (214) 699-6966
ninedotconnects.com/training