**1. In Cadence Allegro Schematic, what is the function of the "Place" menu?**

A) To define electrical constraints  
B) To add components to the schematic  
C) To define the netlist  
D) To generate the bill of materials (BOM)

**Answer:** B) To add components to the schematic

### ****2. How do you assign a footprint to a component in Cadence Allegro Schematic?****

A) By modifying the component properties in the schematic  
B) By linking it to a physical footprint library  
C) By clicking on the "Assign Footprint" button  
D) Footprints are automatically assigned in Allegro

**Answer:** B) By linking it to a physical footprint library

**3. What does the "Update" command do in Cadence Allegro?**

A) Updates the schematic design rules  
B) Updates the layout with schematic changes  
C) Updates the netlist with new components  
D) Updates the design report

**Answer:** B) Updates the layout with schematic changes

**4. In Cadence Allegro, which of the following is used to check for electrical errors in the schematic?**

A) ERC (Electrical Rule Check)  
B) DRC (Design Rule Check)  
C) LVS (Layout Versus Schematic)  
D) BOM (Bill of Materials)

**Answer:** A) ERC (Electrical Rule Check)

**5. Which command is used to create a netlist in Cadence Allegro?**

A) Generate Netlist  
B) Export Netlist  
C) Create Netlist  
D) Generate Report

**Answer:** A) Generate Netlist

**6. How do you edit a component's properties in Cadence Allegro Schematic?**

A) By right-clicking and selecting "Properties"  
B) By double-clicking the component and modifying the attributes  
C) By selecting the component and pressing "F2"  
D) By selecting "Edit" from the toolbar

**Answer:** B) By double-clicking the component and modifying the attributes

**7. Which file format is commonly used for exporting the netlist in Cadence Allegro?**

A) .brd  
B) .net  
C) .xml  
D) .txt

**Answer:** B) .net

**8. In Cadence Allegro, what is the purpose of the "Symbol" editor?**

A) To define the schematic layout  
B) To edit the schematic symbols used in the design  
C) To generate the bill of materials  
D) To create component footprints

**Answer:** B) To edit the schematic symbols used in the design

**9. What is the purpose of the "Pin" tool in Cadence Allegro?**

A) To add components to the schematic  
B) To define electrical pin connections  
C) To assign a footprint  
D) To place via connections

**Answer:** B) To define electrical pin connections

**10. In Cadence Allegro, which of the following is used to check for design rule violations in a schematic?**

A) ERC  
B) DRC  
C) LVS  
D) DRU (Design Rule Update)

**Answer:** A) ERC

**11. Which tool would you use to perform Layout Versus Schematic (LVS) checking in Cadence?**

A) Cadence Layout  
B) Cadence Allegro PCB Designer  
C) Cadence Assura  
D) Cadence PSpice

**Answer:** C) Cadence Assura

**12. How can you place a power symbol in Cadence Allegro Schematic?**

A) Using the "Place" menu and selecting "Power"  
B) By drawing a rectangle and labeling it as power  
C) Power symbols are automatically included with the components  
D) By using the "Add Part" tool

**Answer:** A) Using the "Place" menu and selecting "Power"

**13. In Cadence Allegro, what is the function of the "Generate Report" tool?**

A) To generate a report for the bill of materials  
B) To generate a list of errors in the schematic  
C) To generate the design rule check report  
D) To generate the layout for the PCB

**Answer:** A) To generate a report for the bill of materials

**14. In Cadence Allegro, which tool is used to define net classes and electrical constraints?**

A) Constraint Manager  
B) Design Rule Checker  
C) Layout Editor  
D) Component Manager

**Answer:** A) Constraint Manager

**15. Which type of symbol is used to represent a ground in a schematic?**

A) GND  
B) GRD  
C) VSS  
D) All of the above

**Answer:** D) All of the above

**16. What is the "Part Search" tool used for in Cadence Allegro?**

A) To find specific component pins  
B) To locate parts in the library  
C) To check for unconnected parts  
D) To search for footprints

**Answer:** B) To locate parts in the library

**17. How do you create a custom component in Cadence Allegro Schematic?**

A) By importing an external library  
B) By using the "Symbol Editor" to create the symbol  
C) By editing the netlist file manually  
D) By using the "Create Part" command

**Answer:** B) By using the "Symbol Editor" to create the symbol

**18. Which of the following would you use to define the physical properties of components in Cadence Allegro?**

A) Component Footprint  
B) Pin Assignment  
C) Component Properties  
D) Schematic Symbols

**Answer:** A) Component Footprint

**19. In Cadence Allegro, which tool is used for schematic annotation?**

A) Annotator  
B) Netlist Generator  
C) Design Rule Checker  
D) Component Manager

**Answer:** A) Annotator

**20. In Cadence Allegro, what is a "Design Rule Check (DRC)" used for?**

A) To ensure components are placed in the correct locations  
B) To verify that the schematic meets specified design constraints  
C) To generate the component footprint  
D) To verify the netlist against the schematic

**Answer:** B) To verify that the schematic meets specified design constraints

**21. How do you connect two components together in Cadence Allegro Schematic?**

A) Use the "Wire" tool to draw a wire between the pins  
B) Use the "Net" tool to connect components  
C) Components are connected automatically when placed  
D) By adding a net name manually

**Answer:** A) Use the "Wire" tool to draw a wire between the pins

**22. In Cadence Allegro, what is the "Pin Swap" feature used for?**

A) To swap the positions of pins on a component  
B) To exchange pin functions between components  
C) To assign the same footprint to different pins  
D) To automatically place pins in the layout

**Answer:** B) To exchange pin functions between components

**23. How can you perform a quick check for unconnected pins in Cadence Allegro?**

A) Use the "Check Pins" tool  
B) Perform an ERC (Electrical Rule Check)  
C) Manually inspect the schematic  
D) Use the "DRC" tool

**Answer:** B) Perform an ERC (Electrical Rule Check)

**24. In Cadence Allegro, what is the typical extension for a schematic file?**

A) .brd  
B) .pcb  
C) .sch  
D) .lib

**Answer:** C) .sch

**25. How do you assign a part number to a component in Cadence Allegro?**

A) By editing the component properties and adding the part number  
B) By naming the component when creating the symbol  
C) By linking the component to a database  
D) Part numbers are automatically assigned

**Answer:** A) By editing the component properties and adding the part number