

CUSTOMER'S CHECKLIST

Printed circuit boards, or PCBs, are used to mechanically support and electrically connect electronic components using conductive pathways, or *traces*, etched from copper sheets laminated onto a non-conductive *substrate*. Alternative names are printed wiring board (PWB), and etched wiring board.

SCHEMATIC DETAILS (Schematic is a circuit diagram which contains all the details regarding to PCB layout design like components, their connections etc.):

- 1) Schematic Diagram Availability **(It can be a .pdf file or ready to use circuit diagram made in any of PCB Design software):**
- 2) Component details (like designator, value, package, datasheet etc.):
- 3) Bill of material/BOM **(It contains all the details about components, its footprints, quantity etc. which Requires for PCB layout):**

Notes (if any):

PCB DETAILS:

- 1) No. of layers **(No. of layers the pcb must be designed with if the customer has already decided otherwise it should be decided by the designer considering complexity of the circuit):**
- 2) Board Size **(Total pcb board size in MM,MIL or INCHES):**
- 3) PCB CARD name(if any more details on Pcb)**(The name which the pcb must have after fabrication has been given by the customer):**
- 4) Board clock frequency **(It means the value of frequency that the board runs which is very important for the higher than KiloHertz of range):**
- 5) Technology - Through hole/SMD/Mixed**(Information about whether the components are Surface mount, through hole or mixed of both):**
- 6) Hole & Pad size information(like PTH or NPTH)**(That all holes are plated through, non plated through or both) :**
- 7) Component placement guidelines(if any) **(Any special consideration while placement of components in pcb design):**
- 8) High current & high voltage tracks are **(If the circuit has any higher voltages or currents then any special care to be taken while placement or routing of that particular portion):**
- 9) High frequency signals tracks are:
- 10) PCB mounting hole details:
- 11) Legend details (if any) **(If any special consideration has to be taken in legends or text which is Printed on the PCB like width, placement, height etc.):**
- 12) Test points requirement:
- 13) Differential pair routing requirement:
- 14) Equal data length requirement:

DESIGN RULE SPECIFICATIONS:

- 1) Minimum Conductance clearance **(Minimum distance to be maintained between to tracks or pads):**
- 2) Internal plane clearance**(Minimum distance to be maintained between to internal plane in multilayered pcb):**
- 3) Minimum Conductance width**(Minimum width of the traces in all layers):**
- 4) Minimum Conductance width for power signals**(Minimum width of the traces in all layers):**

PCB Express, Inc.
600 E. Higgins Rd.
Elk Grove Village, IL 60007

Phone : (847) 952-8896
Fax : (847) 952-8893
E-Mail: pcb express@aol.com

- 5) Board outline to route area clearance:
- 6) Board outline to component area clearance:
- 7) PCB mounting hole clearance:
- 8) Minimum via for signals(**Via means plated through hole**):
- 9) Minimum via for power signals(**If any limitation is there in no. of vias for power signal**):
- 10) Solder mask clearance:
- 11) Heat releasing area (if any):

SPECIAL ARTWORK INFORMATION (if any):

- 1) Orientation of components:
- 2) Placement of any particular components:
- 3) Ground pattern (if any):
- 4) Grounding of mounting holes:

SPECIAL POINTS TO BE CONSIDERED: