

[Wiki »](#)

PCB Design Checklist

<!--td {border: 1px solid #ccc;}br {mso-data-placement:same-cell;}-->

Library Elements	Library elements are the combination of schematic symbols and pcb footprint. For every new (not used before in a design) review the following
Verify footprint dimensions with second set of eyes	Have a second colleague review each custom library element to the datasheet specified land pattern and pinout. Verify pin mapping is correct.
Verify mechanical dimensions and features of part	Verify any mechanical features such as board guides, thermal pads, floating "NC" pins, confirm units converted correctly (mm to inch or inch to mm), verify the package choice to the library footprint you are using
Schematic Symbol	Verify all pins, even NC are present in schematic symbol, make sure pin numbers are mapped to correct pin for the package you use, always shown pin numbers and functions in schematic, label schematic elements with a name and value field, make sure the name and value are linked to the footprint, ensure part outline is correct, ensure "keepout" is correct for routing/courtyard maintenance
Footprint	Ensure name field is linked to schematic symbol, ensure pinout and pin labeling match datasheet for all functional pins, ensure non functional pins are labeled in an obvious way, ensure dimensions of pads include adequate soldermask dam (approx 4% increase over copper area), ensure paste layer is present on all pins, ensure all necessary through holes, board guides, thermals are present, avoid placing vias under thermal pads, avoid placing vias in pin pads. Ensure proper courtyard clearances for pick N place assembly
Linked Part	Ensure you link the mechanical pins to the functional pins according to the datasheet, check this at least twice. Ensure the linkage is for the correct package you intend to use, many times different packages have different pinouts while having the same total # of pins.
Silkscreen and Designators	Verify the following for all designators and silkscreen shapes
Polarity marks exist and are visible after component is installed	Polarity markings shall be visible after components are installed. Immediate visibility of these marks aids automated inspection, supports quality and decreases cycle time.
Diode orientation identified clearly	Adopt a feature, such as a 'K' in the silkscreen outside of the component courtyard, or a even a unique pad shape, to denote the Cathode on all diodes locations. Components should also be Cathode marked whenever possible.
Reference Designators use vector font	Vector fonts should always be used during layout to avoid differences from design to final printing of board
Reference designators aligned in same orientation	Preference is to have all Ref Des in same alignment
Reference designators same size, positioned appropriately	Ref Des should use the same font size across the board and be given adequate clearance from the associated parts and other nearby parts so they are easy to read and optically inspect
Silkscreen color defined in assembly	Board assembly document or fab drawing should call out silkscreen color. Color should be chosen so that it contrasts the soldermask color

docs & chosen to contrast soldermask appropriately	sharply and is easy to read
Part number, revision, date, and authorship/copyright	Silkscreen should contain a part number or board name, revision level, date of fabrication, and authorship record/copyright information. Alternatively, this information can also be etched directly in the copper in any unused space on top or bottom layer
Component Placement	It is important to carefully consider the placement of the components on a PCB to ensure that the board will be easy to manufacture, and rework
Components have adequate spacing to board edge or other critical dimensions	Using an automated router to depanel the arrays into individual PCBs can help preserve the edge-mounted components. Generally a minimum component to edge or critical dimension spacing of at least 0.02" should be maintained. Panelizing into arrays with frames would cooperate with existing conveyors and clamping mechanisms.
Lead/hole size relationship	Through hole parts have adequate holes defined to account for constriction of apertures during final plating? Generally lead holes should be oversized by at least 0.005"
Do packages fit the pads?	Print out a 1:1 ratio of the part footprints and place part with tweezers on the paper print out to ensure everything fits properly
Really small components, really fine pitch, leadless devices?	Anything smaller than an 0603 can cause trouble in reflow soldering if the pads and surrounding traces/pours are not properly considered
Courtyard issues, are devices too close?	Avoid courtyard issues by always defining an appropriate courtyard in your library, never place parts within the courtyard of another part
Devices that are difficult or cannot be reworked post assembly?	BGAs notoriously difficult to rework, some DFN/QFN packages as well, thorough library verification will minimize these issues
SMT verses PTH mix?	Many designs need both SMT and PTH parts, just keep in mind this does add an extra assembly step, while you want to design to minimize assembly cost, you should never do so if it puts your reliability at risk. (If doing population yourself, not an issue)
Fiducials inadequate or missing?	Fiducial marks are used by your manufacturer to establish a coordinate reference system for pick N place and automated optical inspection. Best design practices would emphasis that each PCB contain at least three fiducials, two of which are diagonal and as far apart from each other as possible. The third shall reside in the same plane as at least one of the other two fiducials. Each fiducial shall consist of a finished copper landmark measuring 0.025" in diameter with a 0.085" keep-out area that is relieved of copper and free of any silkscreen, attribute, or other material that would jeopardize the contrast required for successful automated registration. (If doing population yourself, not an issue)
Do wire modifications exist?	Avoid wire modifications at all costs on production/flight hardware. Take the time to make the corrections and re print the boards before sending boards with wire modifications out the door. Wire modifications should only be used in debugging/prototyping and testing
Dry parts exist?	Dry parts are parts that can not be exposed to cleaning chemicals associated with removing flux residue, these are often mechanical parts that are at risk of failure if liquids get inside them. Many potentiometers, relays, switches, connectors, etc can be at risk. CHECK THE DATASHEETS

Are there vias in pads?	Vias in pads is not advisable as it can lead to an uneven surface for reflow assembly, if you need a via in/near a pad, it is recommended to place them at the edges/ends of the pads, rather than in the center. The final plating finish you specify also have different flatness impacts, uneven pads lead to poor reflow assembly and possibility of parts missing connections
Are un-masked vias far enough away from the solder site?	An adequate solder mask dam, about 0.004" minimum, is required to prevent solder from scavenging from the joint to nearby via, thus leaving insufficient defects. Masked (tented) vias are 'best practice' except when they reside in a thermal lug. Mask vias whenever possible.
BGA Specific Issues	BGA or Ball Grid Array parts have many design considerations. If another package option is possible, it is preferable to use non-BGA so that it can be hand soldered/reworked. For each BGA device in your design, look at the following
Un-masked vias or inadequate solder dams under BGA devices?	Do not place unmasked vias under BGA devices, they can lead to shorts or solder reflowing from pads to vias and either shifting chips, or shorting pins, solder dams(mask) should be designed precisely to datasheet recommendations
Solder mask at fine pitch locations?	If you specify soldermask features below 0.004", they may print poorly and in some cases have bad adhesion, this can lead to unintended shorts that can only be seen by X-ray inspection
Different pad sizes for BGAs?	Ensure you adhere firmly to the datasheet for any BGA parts/pads
Mixed SMD and NSMD approaches to BGAs?	Soldermask Defined Pads and Non Soldermask Defined Pads - Generally advised to use one or the other, a mix can lead to thermal gradients and poor reflow quality
Lead-free solder balls on BGAs in a leaded design?	Doublecheck the paperwork, if you are running leaded reflow, the lead free balls on BGAs will not properly reflow. Make sure you order the right parts for your design!
Large deltas in thermal mass across assembly?	Thermal mass should always be balanced across pads, avoid introducing thermal mass gradients, use thermal spokes to balance wherever pads attach to larger pours
Thermal relief?	Thermal reliefs/thermal spokes should be used anytime a small pad attaches to a larger copper pour, unless other concerns like RF performance, high current paths, etc. connecting a thermal pad to a large thermal pour can lead to bad reflow, twisting, turning or tombstoning of parts, it can also lead to parts being incompletely soldered
Thermal balancing of the supporting traces on smaller packages like those 0603 and smaller,	It is critical to balance the thermal gradients across smaller parts, the smaller the part the more important this is. Worst case scenario is having copper going in two different directions off the end of pads. Avoid coming off the corners of pads as well. Route traces off same side, or both ends. If you have one side attached to a copper pour, thermal reliefs are critical and the other trace should leave perpendicular to the copper pour
Traces that promote natural bridging?	Do not route traces directly between two side by side pads on chip, this can lead to natural solder bridging and false alerts to automated inspections. Better to come off pads away from chip, then connect traces
Oversized geometry on thermal lugs?	Need to be careful with mask definitions and thermal pad sizes, too much solder paste on thermal pad can lead to uneven reflow or bridging, or worse shorts to vias underneath chips
Can all of the ingredients survive the higher	Paperwork exercise, if you are spec'ing a lead free design need to painstakingly check datasheet for every single part to verify compliance with required process temperatures

temperatures associated with lead-free solder processing?	
Panelization	Panels of PCBs can reduce manufacturing costs by placing multiple design on the same PCBs. If used, care needs to be taken around the borders of the panels
Tabs/mousebites?	Using an automated router to depanel arrays into individual PCBs would eliminate the risk of damaging the edge-mounted components that reside throughout and, ultimately, serve to increase the available real estate in support of greater overall layout densities. The router requires that each individual PCB within the array has three holes @ 0.128" diameter to secure them individually during the routing operation.
Panel breakaways interfere with overhanging components or features such as connectors?	Ensure that your panel connection points are not in line with any parts that need to stick paste board edge
Finishing & Documentation	These steps help ensure that your design is manufacturable by the PCB fab house. Not all of the below are important for all designs (especially for student projects), but implementing good design practices is a good habit.
Is no-clean flux residue acceptable?	Paperwork exercise/operating environment study. Generally it is always better to remove flux residue, but some fringe cases can allow skipping this step
Are trace cuts required?	Similar to wired reworks, take the time to fix the design rather than go to production with traces that require cutting. If you are unsure about a trace, design a bridge feature that is easy and obvious on where to cut
Is 'trace & space' and smallest hole diameter identified?	Must specify your minimum trace size, spacing / clearance size, and small drill hole in your documentation. Generally 0.006" tracing/spacing and 0.008" drill holes are non cost adders for most manufacturers, smaller sizes can be achieved but may add cost. Design for what you need to have a reliable functioning circuit, only focus on driving down cost if your design is losing money :)
Is the product controlled by RoHS?	Paperwork exercise to verify BOM parts are all compliant with RoHS if you wish to claim RoHS compliance
Is the PCB finish clearly identified?	Documentation/fab notes should clearly specify final pad plating, soldermask color, silkscreen color, overall thickness tolerance
Is the laminate stack clearly defined?	Documentation/fab notes should clearly specify copper thicknesses, substrate type/thicknesses, prepreg type/thicknesses
Are build standards identified?	Any relevant IPC or other standards that need to be adhered to in printing/assembly process. Any controlled impedances should also be called out
Maximum potential voltage identified?	Documentation should specify a maximum operating voltage for the PCB, will not always align with the available electrical test equipment
PCB base-material laminate specs	Always remember to spec your material FR4, Rogers, etc and thickness, don't leave it up to chance

missing or inadequate?	
Does a location exist for a label?	Include a 0.25" x 1.0" box in the silkscreen of each board, if the product is to be labeled for product id & traceability.
Does a legend exist for Gerber layer files?	Put a gerber legend in your fab notes, clearly calling out file names, extensions, and proper order in the layer stackup for the mechanical layers
Does the product have Regulatory Requirements?	RoHS, FCC, CE, CPSIA, FAA, Other? Master your regulatory environment
Multi source Approved Vendor List (AVL) / Approved Materials List (AML)	Generally you should try to have at least two sources vetted/approved for all of your printing, assembly, and components. Many ICs do not have an acceptable substitute, these are known as supply chain blockers and should be planned more carefully for worst case potential lead times
Torque specifications identified?	Fab notes should specify maximum torque to be used while assembly PCB into mechanical enclosure. If you are screwing into plastic, or metal, or some other material, the specs will be very different. They will also be differentiated based on the screw head type, material, and finish
Clearly defined fabrication notes exist?	Fabrication notes must be delivered with every design, reduce your risks as much as possible by leaving nothing to chance. Build a template and copy it/edit for each design. Include the fab notes in your design review
Electronic Data?	All products require Gerber data, clearly defined fabrication notes, CAD data in ASCII format, BOM, X-Y Data, schematic and any other info that might support overall quality.