DFM Guidelines

These international guidelines developed by the Association Connecting Electronics Industry or IPC have helped standardize the assembly and production of electronic equipment and assemblies. Following these guidelines will allow for a smooth and accurate quoting and manufacturing process.

Bill of Materials (BOM)

BOM Example

- BOM's are used to detail component information, purchase components, and place stated components.
- Preferred format is Microsoft Excel file for easy data transfer.
  - File name should be specific with a revision or date created.
- Select components with machine placeable features and packaging options that can be placed with automatic pick & place equipment.
  - Many manufacturers offer pick caps for SMT connectors/switches.
    - Avoid selecting SMT components that need to be hand placed as this increase's costs and the opportunity for errors.
- BOM's should include the following:
  - Manufacturer's name and component number for each component to be placed including hardware and wires needed.
    - List acceptable substitutions in own column.
    - Check for and remove duplicate component numbers.
  - Reference designators for each component.
    - When complete, sort BOM by reference designators from A to Z.
    - Check for and remove duplicate reference designators.
      - If more than one component goes to the same reference designator, adding an additional letter to the additional component reference location is helpful or stating "For x location".
  - Quantity of each component.
    - Check that the quantity matches the number of reference designators listed.
  - Description of each component.
    - Digi-Key descriptions preferred.
  - Separate tab or section for Box Build or Higher Level Assembly BOM.
    - All specifications above should be included.
    - Custom components should state current manufacture if known.
  - When complete, sort BOM by reference designators from A to Z.

- BOM's can include the following:
    - Assembly number/name and revision.
    - PCB number/name and revision.
    - Non-populated/DNP locations.
        - This can be either an entire list of DNP locations at the bottom of the BOM or component lines with a quantity listed as "0" or "DNP".
    - Component Value.
    - Component Package/Decal.
    - Component parametric information.
    - Indicate if a component is "critical" and no subs are allowed.
    - Indicate if a component has to be purchased from the manufacture directly.
    - Identify moisture-sensitive devices and the MSL Level.
    - Identify components with temperature or process restrictions.
    - Notate components with "extra" manual assembly effort but state all instructions on the Assembly Drawing.
- BOM's should **NOT** include the following:
    - Component placement instructions.
        - This information should be detailed on an Assembly Drawing document.

Assembly Drawing
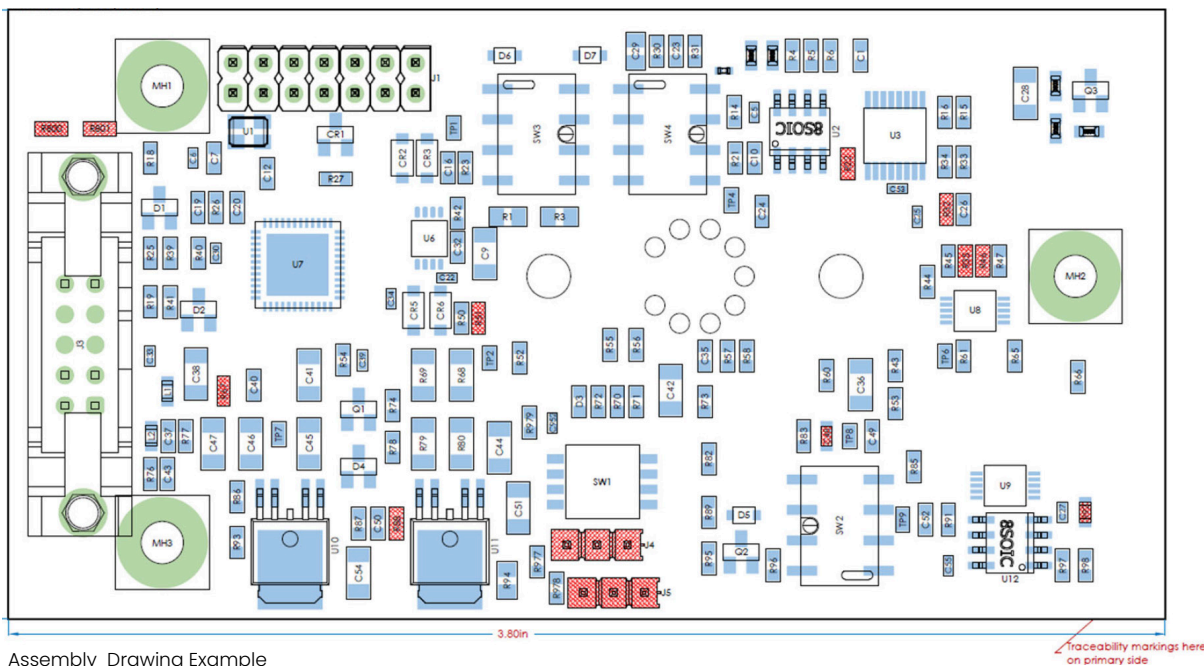
Assembly Drawing Example

- State Assembly Number/Name and Revision.
    - Changes should be detailed with each revision change.
- Define workmanship standard of either IPC 610 Class 2 or Class 3.
    - Pro-Active Engineering Default: Class 2.
- State RoHS requirement if needed.
    - Pro-Active Engineering Default: RoHS Compliant.
- Define any solder/flux and cleaning/ionic testing requirements.
    - Pro-Active Engineering Default: "No Clean" solder/flux and standard wash process.
    - Indicate if there are "non-hermetically" sealed components that cannot be washed.

- Provide tolerances for any stated dimensions.
- State Programming and Testing Requirements.
  - Programming Files and Programming/Testing Documents should be separate if detailed.
- Specify any of the following, if required.
  - Labeling for PCB assembly and programmed components and labeling for box builds.
    - Visual Depiction Recommend.
  - Components that need special forming, preparation or have restrictions should be specified. (e.g. Components required to be held to a certain height tolerance).
  - Components that need special adhesive or thermal paste should be specified.
    - Pro-Active Engineering Defaults: Devcon 14270 Adhesive and Aavid 100300F00000G Thermal Paste.
  - Conformal coating.
    - Pro-Active Engineering Defaults: Dow Corning 1-2620 (Silicon), Humiseal 1A33 (Acrylic).
      - Typical Types: Acrylic, Polyurethane, or Silicon.
    - Specify any "Keep Out" areas for coating with tolerance, if required.
      - e.g. headers, connectors, switches, LED's, holes/via's under said parts, etc.
    - Specify coating thickness with tolerance, if required.
  - Torque requirements for hardware.
  - Potting material, location of material and thickness with tolerance.
  - Gluing, epoxy, or RTV locations and materials.
  - Tamper seals for box builds.
  - UL, CSA, ATEX or other Regulatory Agency requirements.
  - Any other items critical to build.

Bare PCB Design
- File name should be specific with a revision or date created.
- Minimum single PCB/array size for equipment use = 3.00" x 3.00" which can include rails.
- Maximum single PCB/array size for equipment use = 17.00" x 20.00" which includes rails.
- Minimum of two fiducials on opposite sides of the PCB are required for accurate solder and component alignment and placement. Use 0.040" round dots with some soldermask clearance.
  - Fiducials should be at least 0.3" away from PCB edges if possible.
- Layers of the PCB should be balanced in both layer count and across the PCB to avoid warpage.
- Soldermask clearance from copper pad edges is typically 0.003" with 0.002" minimum.

Bare PCB design

  - Use Soldermask dams between pads to minimize solder bridging – 0.005" is preferred but 0.003" is the industry minimum.
  - Connect common pad sizes on IC's outside the pad area with a trace or dam with mask.
- All component footprints should be in accordance to the device component specifications from the manufacturer and IPC-7351.
  - Extending the pad size beyond the shape of the component is recommended for inspection purposes.
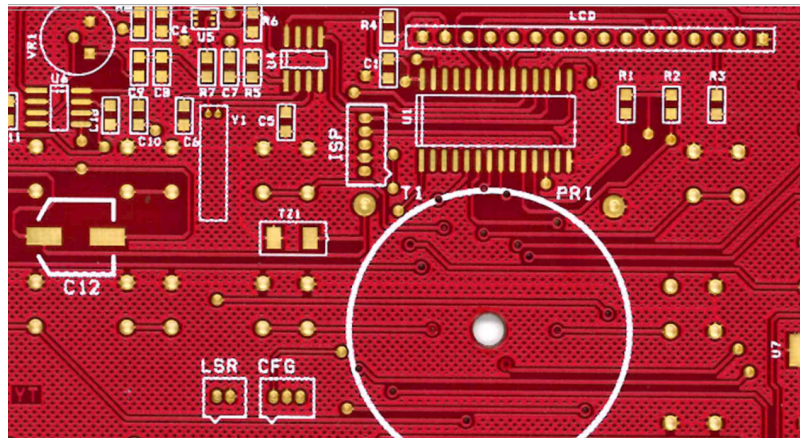
- For Quad Fine Pitch No Lead (QFN) components: keep the pad sizes consistent with each other to prevent tilting which can lead to an 'open' position.
- Avoid hand inserted "hairpin" axial device requirements.
- Maintain component spacing of 0.060" clearance around BGAs for inspection and rework.
- Maintain 0.125" "Keep Out" area between the plated through hole (PTH) lead and the surface mount.
- Avoid vias within the land pattern, underneath components or closely connected to pads. If vias are close to a pad, plug the via or provide a soldermask dam between the pad and via to prevent solder from wicking to via and robbing solder from the intended joint.
  - Cover via to prevent solder from being lost down into the via and starving the joint.
  - Via size will drive the cost of the PCB 0.010" is standard dimension.
- Locate all PTH on primary side of the PCB to avoid manual soldering.
- Place all large mass components on the Primary side of the PCB and items of lower mass on the Secondary side.
  - Typically, Secondary side should contain mostly passive components and smaller / low mass devices.
  - Large / high mass components do not go on the Secondary side. Large components placed on the Secondary side could overcome solder paste surface tension and fall off.
- Minimize SMT component height on secondary side of the PCB to a maximum of 0.25".
- Do not mirror image Leadless (BGA/QFN) components on opposing sides.
  - This makes inspection & rework very difficult.
- Clearly identify polarities on the silkscreen and on assembly drawings.
  - Place polarity consistently in one direction.
- Design for testability: 0.030" – 0.040" test pads on 0.100" centers.
  - Test point for each node.
  - Test pads to be 0.030" from other components.
  - All test points on the Secondary side of the PCB.
- Add at least 0.250" rail on all sides of PCB to accommodate machine handling if possible or where it makes sense, as sometimes only putting rails on two sides would be better due to components overhanging.
  - Minimum of two fiducials on opposite corners on the rails (if used) are required for accurate solder and component alignment and placement. Use 0.040" round dots with some soldermask clearance. These fiducials should be in addition to the PCB fiducials.
    - Fiducials must be at least of 0.250" from rail edges to prevent obstruction from machine clamps.
    - Fiducials should not be on all four corners of the PCB as this can lead to misplaced parts if the PCB is placed into the machine incorrectly.
      - Typically, the other two open corners may have mounting holes added but is not required.
  - If PCB is over 0.093" thick, putting rails on the PCB might not be advisable. Please make sure if your PCB does not have rails that there are fiducials on the PCB.
  - Rails with V-Scoring
    - V-Scores are applied on both top and bottom sides of the PCB along the PCB edge between the PCB itself and the added rails. It is a "v" shaped groove leaving approximately 1/3 of the PCB thickness with a web of the remaining material to support the PCB and stay attached.
    - V-Scores are made with a V shaped bit that typically has either a 300 or 450 angle. The smaller the angle the smaller the cut into the actual PCB will be.
    - Depending on PCB thickness, the webbing left after v-scoring is typically 0.016" to 0.012".
    - Components or other features should not be too close to the PCB edge wherever possible or damage may occur during the depaneling process.

- Rails with Tabs/Mousebites and Routing.
  - Tabs or mousebites with routing might be necessary when v-scoring is not an option but Pro-Active would prefer to have v-scores instead of tabs/mousebites.
    - Typically, this is needed for circular or Flex PCB's or very thin PCB's where v-scoring leaves the PCB rails too flimsy to stay attached.
  - The location of the tabs or mousebites should be carefully located if they might interfere with assembly or placement into higher level assembly enclosures where size and shape may be critical.
- Add at least 0.250" spacing rails between PCB's in array if components overhang.
  - Depending on the type of component and how the component overhangs, routing into the rails may be necessary especially with SMT USB connectors that have a lip that hangs over the edge of the PCB itself once placed.

Bare PCB Specs

- Define PCB Fabrication Specifications (Often all added on a separate layer in the gerbers themselves or can be a separate PDF document or text file added to the gerber file folder).
  - PCB Size (Often added on the Drill Chart layer or PDF).
  - PCB Layers.
  - PCB Material (Be Specific, if needed).
    - Pro-Active Engineering Default: FR4.
  - TG Rating.
    - Pro-Active Engineering Default: 150° C or Higher.
  - Thickness.
    - Pro-Active Engineering Default: 0.062".
    - Other Typical Thicknesses: 0.031" and 0.093".
  - Finished Outer Layers Copper (Includes plating required).
    - Pro-Active Engineering Default: 1oz.
    - Other Typical Thicknesses: 2oz and 3oz.
  - Finished Inner Layers Copper (If needed, no plating required).
    - Pro-Active Engineering Default: 1oz.
    - Other Typical Thicknesses: 0.5oz and 2oz.
  - Solder Mask Color (Be Specific, if needed).
    - Pro-Active Engineering Default: LPI Green.
    - Other Typical Colors: Blue, Red, and White.
    - If needed, Matte or Glossy finish can be added.
  - Silk Screen Color (Be Specific, if needed).
    - Pro-Active Engineering Default: White Epoxy.
    - Other Typical Colors: Black.
  - Finish/Plating (Be Specific, if needed).
    - Pro-Active Engineering Default: ENIG (Highly recommended for BGA's/QFN's).
    - Other Typical Finishes: HASL (Non-RoHS), LF-HASL and Immersion Silver.
  - Specific Stack-Up Design.
    - Visual Depiction Recommend.
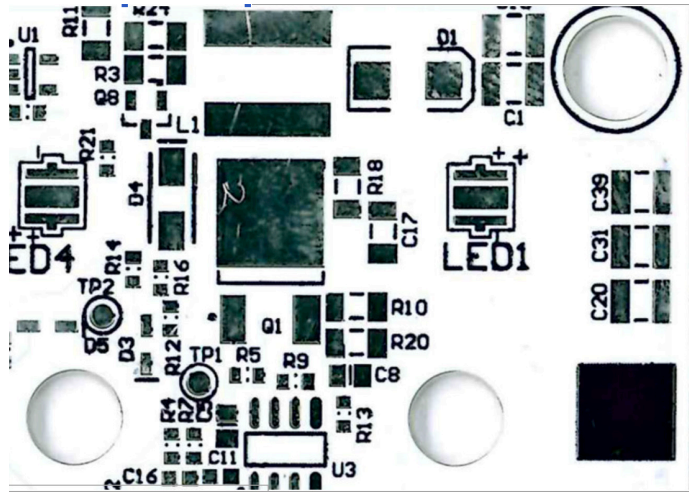
Bare PCB Example

- RoHS Compliance Requirements.
    - Pro-Active Engineering Default: RoHS Compliant.
- Class 2 or Class 3 Manufacturing.
    - Pro-Active Engineering Default: Class 2.
- Drill Chart with Symbols, Sizes, and Plated/Non-plated Holes noted.
- Note following when needed or when outside of Class 2 or Class 3 stated requirements.
    - UL Requirements.
    - Minimum Trace/Spacing Requirements.
    - Via Requirements.
        - Via-in-pad fill Requirements.
        - Vias in general Requirements.
            - Covered, Flooded, Plugged or Tented.
                - Note if needed for certain size vias only.
    - Blind/Buried Vias Requirements.
        - Visual Depiction Recommend.
    - Controlled Impedance/Dielectric Requirements.
        - Clearly specify the correct size traces and make sure to have the thickness of the material match the requirements.
    - Plating Requirements.
    - Hole/ Artwork/Soldermask Tolerance Requirements.
    - Electrical Testing Requirements.
    - Warp and Twist Requirements.
    - PCB House Logo Requirements.
    - Gold Tab/Hard Gold Requirements.
    - Edge Plating/Castellation Requirements.
    - Certificates/Solder Samples/Coupons.
    - Any other items critical to build.
- The customer is responsible for providing RoHS compliant Bill of Materials if the assembly requires it. If required, Pro-Active Engineering will build assemblies in accordance to RoHS compliant processes.