Ditron MANUFACTURING

Design for Assembly

Tips and Tricks

1. Communication is Rule #1
2. Considerations During the Planning Phase
3. Printed Circuit Board Marking
4. Part Number Marking
5. Panelization and Fiducials
6. Component Orientation
7. Miscellaneous Information
8. Documentation

Ditron Manufacturing, Inc. Quality Policy

Ditron Manufacturing is committed to understanding its customers' needs and delivering fully-conforming products using efficient, cost-effective, state-of-the-art technologies.

Design For Assembly: Rule #1

To realize cost-effective design for manufacturability and assembly, Rule #1 is "communicate, communicate, communicate." Communicating with your board fabricator and the experts at Ditron before, during and after your design project is a key element for success. The best way to keep abreast of the most recent technologies that facilitate reduced costs is to ask your fabricator and our engineers.

Communicate with your suppliers. New technologies are continually being developed that revolutionize the manufacturing and assembly process.

Dītron
MANUFACTURING

Design For Assembly: Considerations During the Planning Phase

Discussions during the design phase can reduce or eliminate process issues during manufacturing. The following elements of your design should be discussed in detail with the staff at Ditron during the design phase:

Type of Assembly. Designing a board for Surface Mount machine assembly, rather than Thru-Hole or hand assembly will help to ensure accuracy and quality, while reducing assembly costs.

Panelization. Efficient panelization can reduce costs of the raw board, components, and process time.

Component selection. Many factors should influence the selection of components: Fine pitch, size, lead-free versus leaded components, temperature tolerance for the end use, availability in the marketplace, etc.

Solder Requirements. Determine whether or not the assembly will require the use of no-clean solder. In high-speed/high volume assemblies, this will increase efficiency and reduce process time.

Other Considerations. Component placement/location on the PCB, size and number of thru-holes, fiducial locations/types, gold plating requirements, conductor width and spacing.

Note: Production volume is not determined solely on the total quantity of units being produced. A small run of boards containing several hundred components could be considered high volume, while several hundred boards with only a few components per unit may be considered low volume. Simple math: Process + Components + Units = Total Volume.

Design for Assembly: PCB Marking

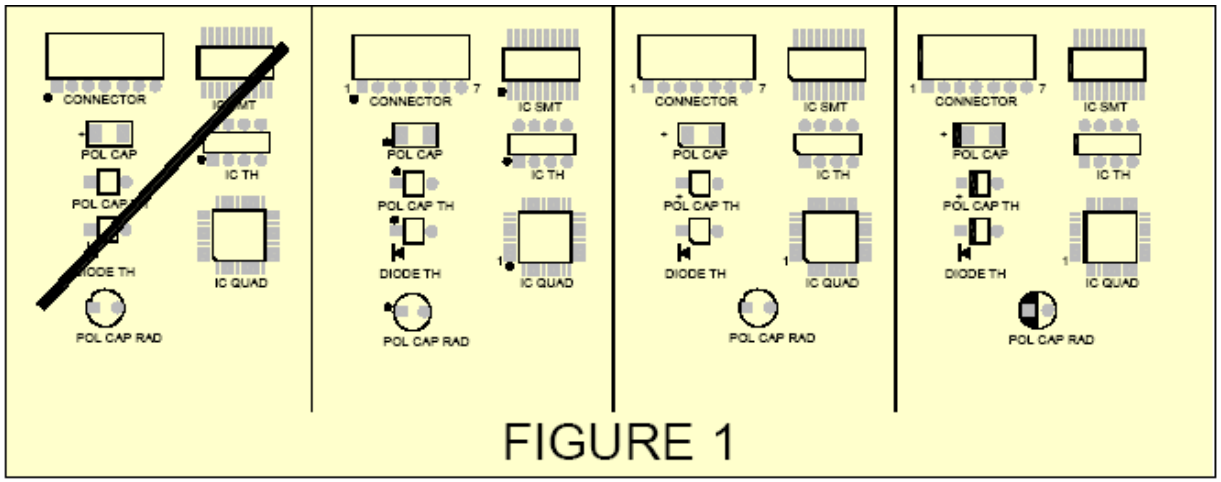
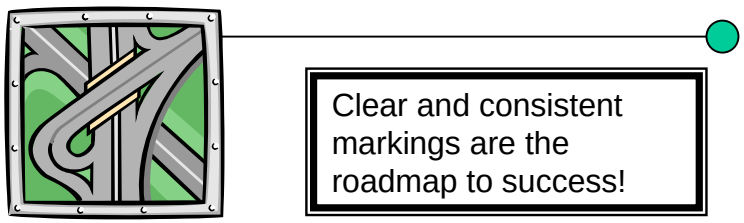


Figure 1. Silkscreen markings should be consistent in your design. Make polarization markers very bold. Use only one method of marking pin orientation and polarization. Pin identification is required when orientation is critical. Historically, components such as diodes, voltage regulators, transformers, etc. do not have standardized pin configuration. Be consistent with line thickness and text height and thickness. Consistency and accuracy in markings helps facilitates the assembly and inspection of your PCBs.



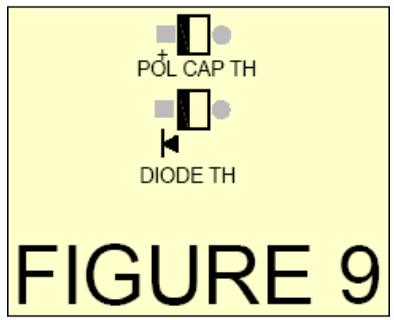


Figure 9. Bold and obvious orientation markers should be used to identify components whose orientation is critical to the performance of the end product. The marking should always be visible, even when the component is installed. This will facilitate inspection and trouble-shooting.

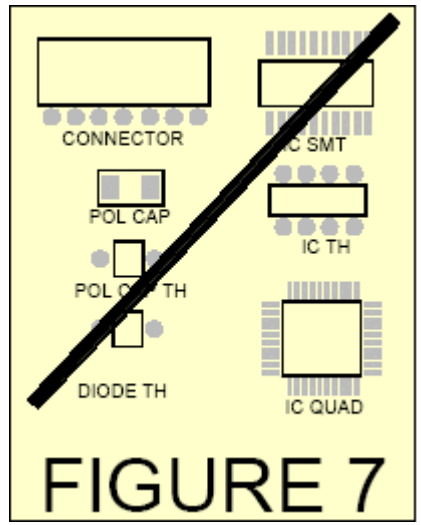


Figure 7. Not identifying component orientation leads to misplacement in the assembly process!

Design for Assembly: PCB Marking

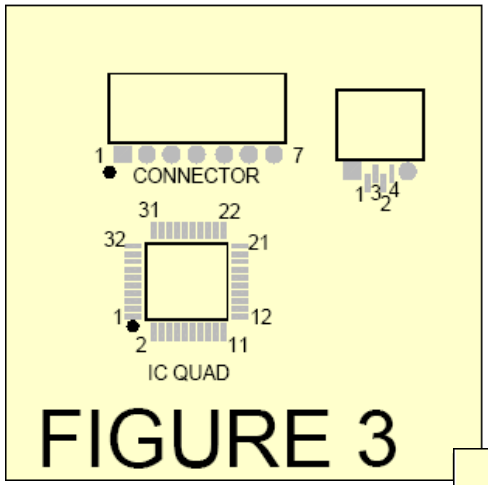


Figure 3. Pin marking will expedite trouble-shooting and assembly inspection. Always identify Pin 1 on every component where orientation is critical. Beyond that, identify the first and last pins of each row on components such as connectors and ICs. For clarity, it may also be necessary to mark each individual pin of the component. When identifying connectors mark not only the pins on the primary side of the PCB, but add the pin numbers, where space permits, in copper on the secondary side of the board.

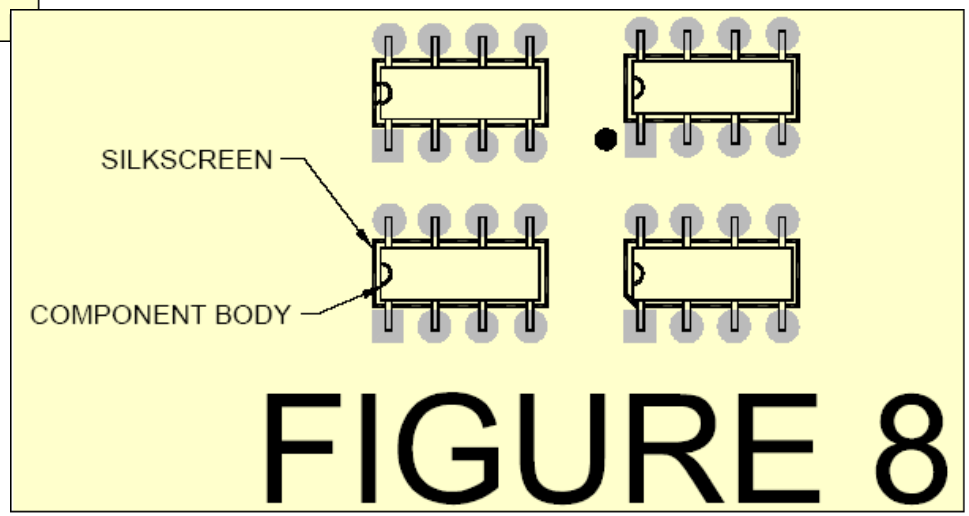


Figure 8. The silkscreen outline should always be larger than the actual component; a good rule of thumb is 10% larger. This ensures that during inspection the correct placement can be visualized and misplaced components would be more obvious.

This identification process doesn't require a passport, but pin marking and identification are the passport to success for your assembly!

Dītron
MANUFACTURING

Design for Assembly: Part Number Marking

PCB Assembly Name

PCB Assembly P/N: 123456 Rev: [redacted]

Bare Board P/N: 123456 Rev: A

Artwork for your PCB should include the part number, revision, company name and a proprietary note on each sheet of the artwork. The part numbers and company name allow for easy identification of your artwork layers.

On the silkscreen: The Assembly Name and Part Number should be clearly marked on the primary side. A large area should be reserved for marking, stamping or labeling the revision. Leaving an area for marking the assembly revision, instead of having it pre-marked, allows for easy revision of the assembly. If your numbering system integrates the revision as part of the part number, leave an area large enough to incorporate the Part/Rev number.

On the primary side in copper: The bare board part number and revision should be marked in copper. By doing so, it then differentiates it from the assembly part number and prevents modification of the part number and revision. When a bare board needs to be revised, it should be assumed that the schematic and artwork would be updated.

Ditron MANUFACTURING

Design for Assembly: Panelization and Fiducials

Figure 5. Fiducials are used by automated assembly machinery to check the positioning of the PCB under the gantry of the machine. Skew and offset can also be determined and adjustments can be made automatically when placing components. Fiducial sizes and requirements vary between assemblers. The fabricator must also allow solder mask clearance to prevent covering the fiducials, thus reducing placement accuracy. Fiducials should always be in pairs and in a minimum of three locations: on the lower left and upper right of the assembly panel; on the lower left and upper right of the PCB; on the lower left and upper right of fine pitch components.

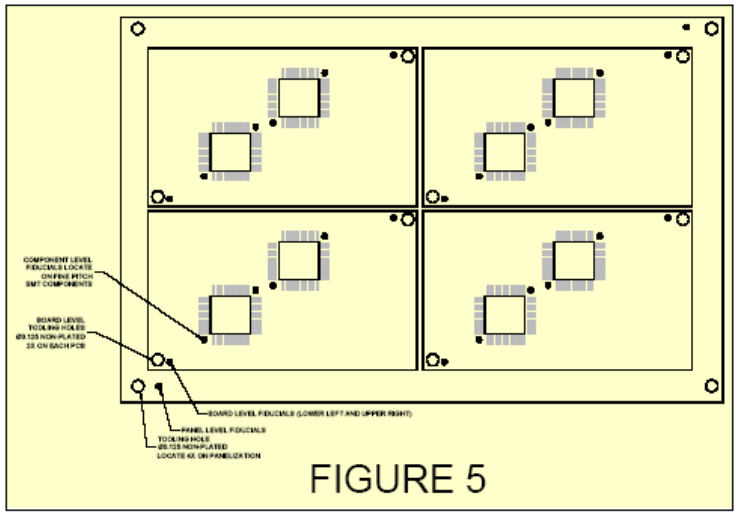


FIGURE 5

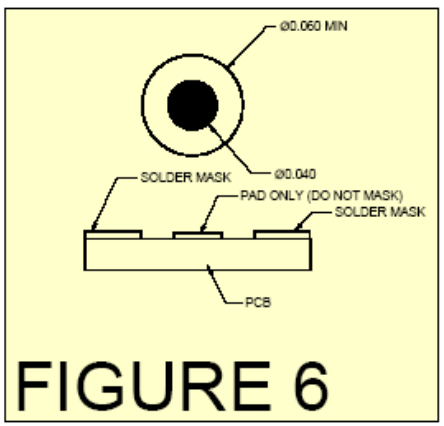


FIGURE 6

Figure 6. One of the easiest ways to increase through put is to panelize your PCBs. While fabrication panelization is designed to ensure the best possible use of a fabrication panel --- critical for low-cost bare boards --- panelization for assembly is no less critical for high production runs. Special design criteria must be taken into consideration and discussed with the process engineers at Ditron: tooling holes on both the raw PCB and the panels; use fiducials when the PCB contains surface mount components intended for machine placement; break-away mechanisms, tabs or “V” groove (see Figures 2 & 15); orientation of boards within the panel; and panel size.

Ditron
MANUFACTURING

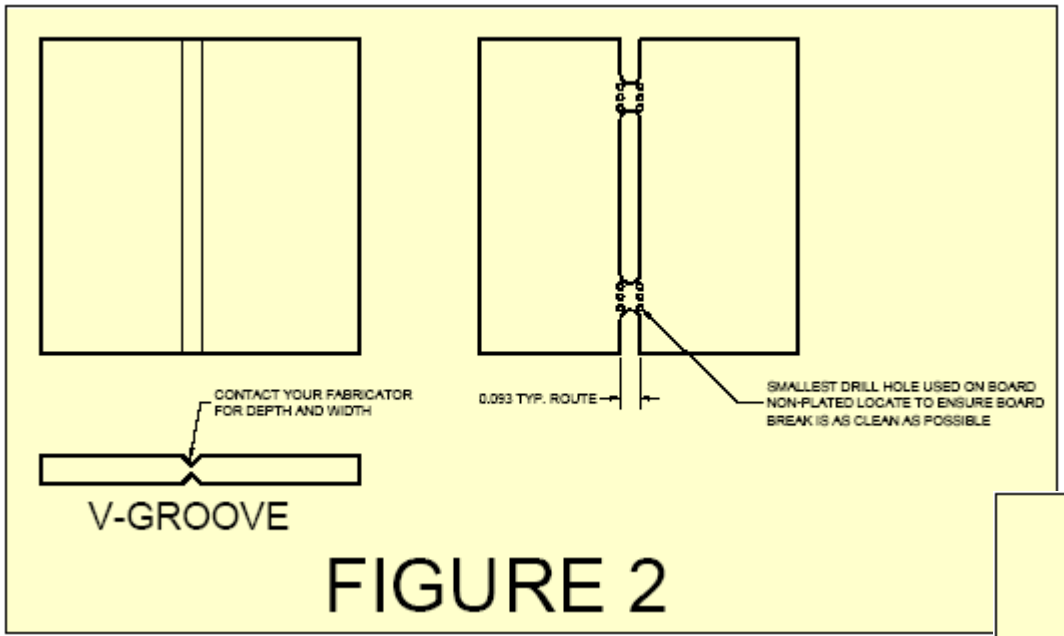
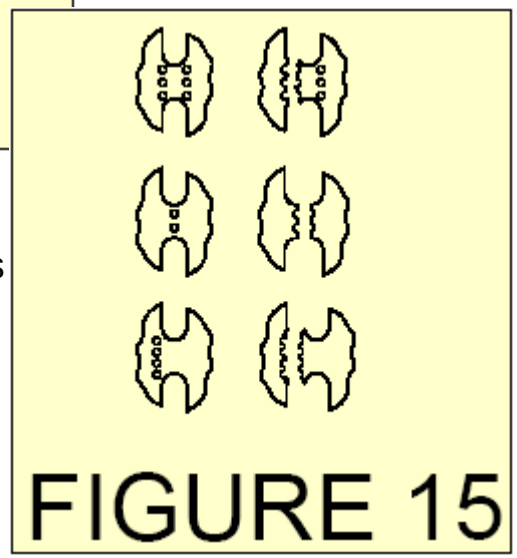


Figure 2. When assembly panels are used, determine in advance the method for holding the PCBs in the panel, as well as separating the PCBs from the panel. There are two popular methods: the "V" groove and the route out. Each method has its own application; Ditron can assist in the selection of a method most appropriate for your assembly.

Figure 15. Break away tabs are used to connect PCB to PCB within a panel, with the placement of the tabs being determined by the PCB designer. Tabs should be spread out to properly support the board throughout the panel. The designer should avoid shadowing the tabs with components that are at right angles, as that increases the difficulty of breaking the tabs.



Examples of break-away borders and their expected break pattern.

Ditron
MANUFACTURING

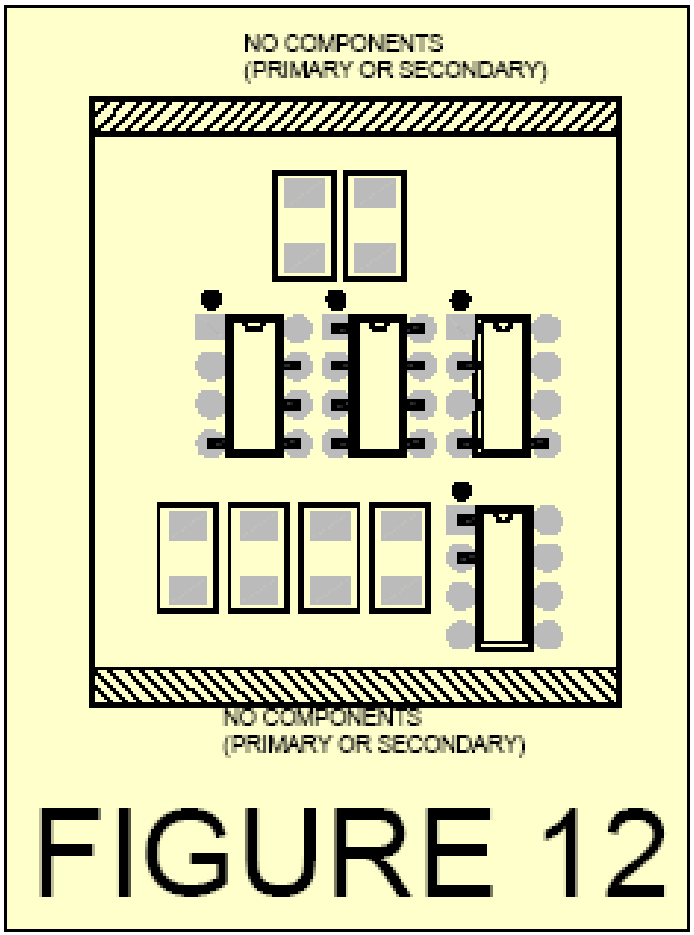
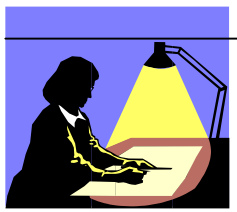


FIGURE 12

Figure 12. Panelization is not always a requirement. In some cases, i.e., when the PCB is very large, it's impossible to panelize. However, to aid the automated assembly process, the design should include rail edges. Components should be kept a minimum of .0250 away from the edge of the PCB. Some rails need to be designed specific to the automated machinery that will be used to assemble the board. Tooling holes and fiducials may still be required, but some PCBs should be designed without break-aways. Regardless of tooling railings, all components should be kept a minimum of .100 away from the board edge to allow for slide rail positioning during the assembly process. Consult with the engineers at Ditron regarding the panelization and railings before you release your board design..



Appropriate fiducial markings, panelization and tabs are critical to the assembly process. Don't be in the dark when you design your PCB's; consult Ditron for the most effective methods for your assembly!

Ditron MANUFACTURING

Design for Assembly: Component Orientation

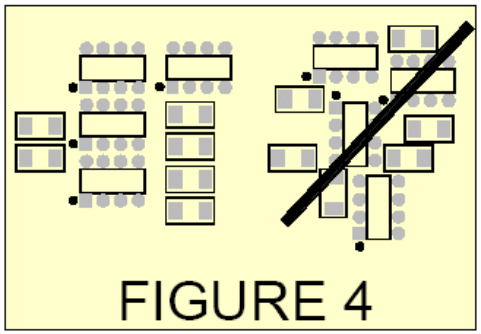
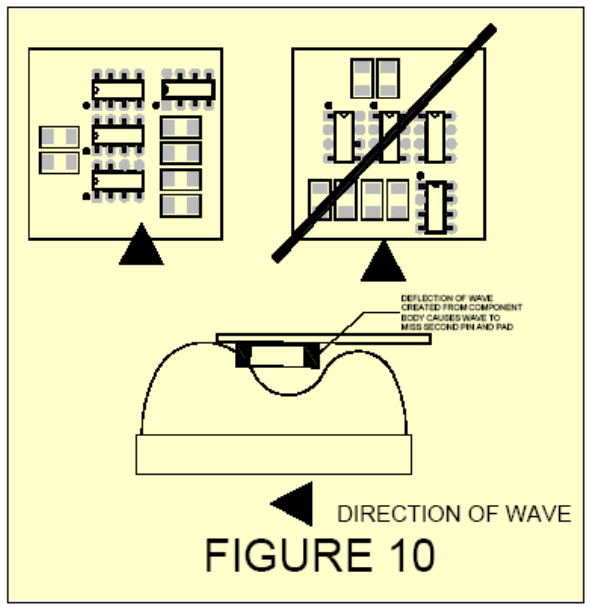


Figure 4. When possible, orient your like components in the same direction and space them out evenly to the horizontal and diagonal. Inconsistent component placement makes he automated placement process more complex and is more difficult to inspect. It is understood that certain situations require that design criteria take precedence over component placement. However, high volume processing benefits the most from cohesive component placement. It facilitates component identification, troubleshooting and inspection.

Figure 10. Component orientation all needs to be considered in conjunction with the assembly *processes* that will be used to assembly your PCBs. Processes such as wave solder are particularly sensitive to component orientation. Proximity of components, even distribution, and height ratios also affect the type of process and the total time a board stays on the assembly line.



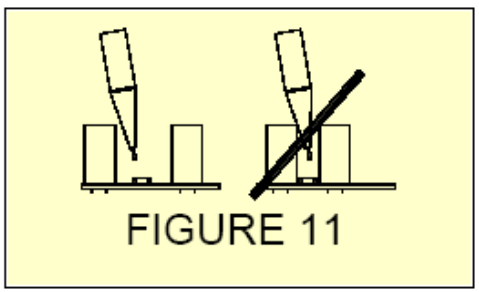


Figure 11. Pay close attention to height relationships when placing components. A chip capacitor placed between two high connectors is hard to rework, troubleshoot and inspect.

Ditron
MANUFACTURING

When considering a design that will require populating both sides of the PCB, there is one important factor to consider: multiple assembly processes will increase the per unit cost of the assembly. Whenever possible, design a single-sided PCB. When it is necessary to have a double-sided PCB, follow these suggestions:

- Place all thru-hole components, ICs and fine pitch devices on the primary side.
- Place chip capacitors and resistors on the secondary side.
- Place large-height components on the primary side and low-height components on the secondary side. Keep in mind height aspect ratios on double-sided PCBs.
- Keep heavy and sensitive components on the primary side.
- Keep thru-hole components or components that require special assembly considerations or hand placement on the primary side.

Careful review of parts placement with the engineers at Ditron can help to ensure efficient and accurate parts placement.

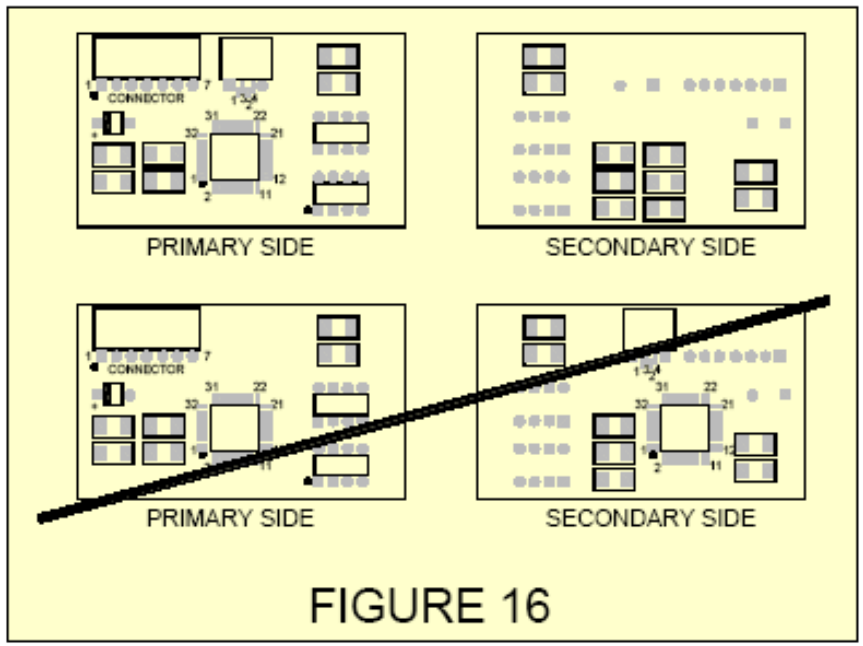


FIGURE 16

Let the engineers at Ditron give you a hand up in your PCB design. Just like components need to go in the same direction, Ditron can point you in the right direction in component placement criteria!

Ditron MANUFACTURING

Design for Assembly: Miscellaneous Facts

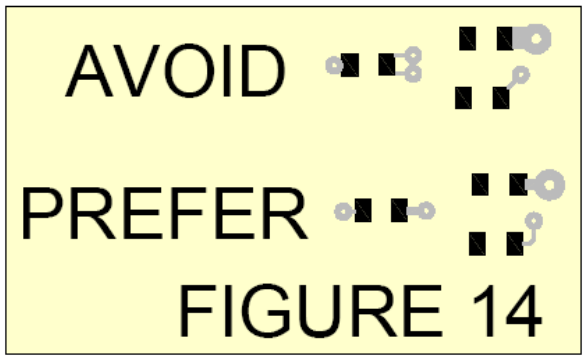


Figure 14. Design for Manufacturability guidelines and Design for Assembly guidelines have shared commonalities. For instance, trace routing to prevent acid pockets during fabrication also applies to the assembly house, just in a different way. In general, good design prevents bad solder joints, solder wicking, etc. When connecting a pad to a plane, make sure thermal relief is provided, as this aids in soldering components to the PCB.

When designing PCBs with leaded components, pay particular attention to the diameter of the hole and the relationship to the diameter of the component pins. Too large of a hole allows voids to form in the barrel of the hole; too small of a hole prohibits solder from wicking up the hole. IPC guidelines concerning this issue are helpful if proper classification of performance is predetermined.

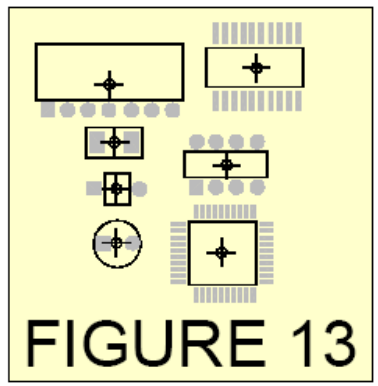


Figure 13. The “centroid” is the theoretical center of a component and can be created automatically by most layout software. In some cases, the centroid point may not be the exact center of the part, but instead the point of physical horizontal balance. Origin points of parts --- usually Pin 1 of the component --- differs from the centroidal point. Centroidal points for thru-hole components can be the origin part of Pin 1, but the centroidal point of surface mount devices should be in the center of the part. Providing a centroid data report in an ASCII format can speed up the time it takes to pre-engineer your PCB for assembly.

Collaboration is a key component in your Design for Manufacturability. Turn to the experts at Ditron during the design phase to meld the product design with DFM.

Design for Assembly:Documentation

When it comes to documentation, there is never such a thing as too much! Provide everything your company will allow. Ditron treats every piece of documentation as proprietary data. Specific security measures are taken to ensure document security and non-disclosure. Secondly, the distribution of customer-provided documentation is controlled. Processes have been developed to ensure document originality can be identified and distribution of documents controlled.

To facilitate your assembly, the following must be provided:

Bill of Materials. The BOM should be in electronic format, with field columns delimited, when possible. Microsoft Excel, Adobe PDF, ASCII with delimited fields are most preferred.

Assembly drawings. The drawings should describe in detail the assembly of every component to be placed on the PCB. Include any special assembly requirements. Drawings should be provided in electronic format: AutoCAD or DXF is acceptable; Gerber with encapsulated Aperture or Adobe PDF is ideal.

Special Instructions. Any special instructions you require, such as markings or conformance requirements should be communicated in writing or in electronic format: ASCII text, MS Word or Adobe PDF is preferred.

Automatic Placement Data. If your CAD Design software provides centroid data, this should be provided to Ditron. ASCII output is preferred.

Gerber Data. Any information available in Gerber format is beneficial. Most importantly would be the Paste Mask, if there are surface mount components on the PCB. Gerber 274X is preferred.

Programming Information. If the assembly has programming requirements of electronic devices on the PCB, include program files, to include the checksum. Test the program before providing it to Ditron to ensure functionality; provide detailed instructions on how the device should be programmed. ASCII output of the instructions is preferred, but MS Word or Adobe PDF is acceptable.

Final Tip. Same as the first ---- communicate, communicate, communicate!