Technical Note - TN004
PCB Panel Design Guidelines

Introduction

This document provides a set of guidelines that have been developed by Norcott to assist customers in the design of PCB panels that will be suitable for manufacture on automated SMT assembly equipment. If implemented they will minimise problems with the PCB assembly, thereby increasing yield and reducing assembly costs.

Note on Dimensions

All drawing dimensions are specified primarily in inches with millimetres in brackets. e.g. 0.010 (0.234)

Panellisation

Panellisation or step and repeat as it is also known, is a method of grouping PCB's onto one piece of laminate (or panel). The images may be a repeat of a single design or a grouping of various designs. If a group of various designs are used this is referred to as a Family of Boards (FOB).

Figure 1 shows 4 rectangular circuits in a single panel with the required tooling features and using breakout pips. This will be used as an example to describe the required features.

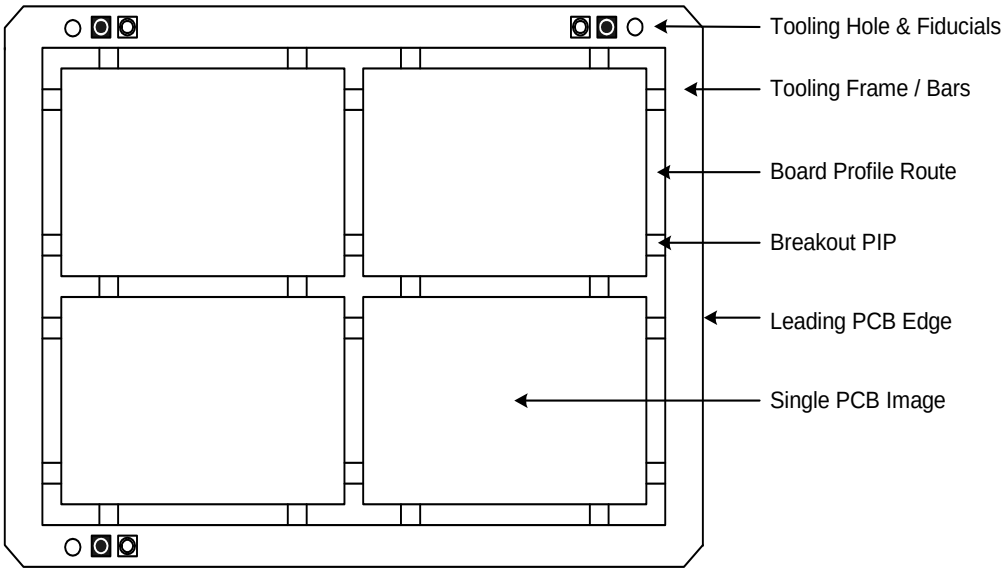

Figure 1 : Stanard Panel Features

Determining Step and Repeat Values

The number of circuits stepped up into the panel is governed by the following factors:

- The size of the individual circuit.
- The weight of components to be placed.
- The overall panel size that can be handled by the assembly equipment.
- Minimum panel size 3.9 (100) x 3.9 (100).
- The most efficient use of the PCB manufacturer standard laminate sub panel.
- Minimum fiducial separation 3.3 (85) .
- Overhanging components and/or spacing of components from the edge of the circuit.

A large panel containing many circuits often has very poor mechanical strength, which requires additional support as there is a tendency for the panel to bend under its own weight during assembly and reflow.

Determining Step and Repeat Values continued..

However, whilst a small panel with relatively few boards may be stronger, it may also be very inefficient in utilising the PCB manufacturers standard fabrication panel sizes. It also will increase the amount of board handling during assembly, thereby increasing the overall cost.

As a 'rule of thumb' the optimum panel is approximately the size of a sheet of A4 paper or 300mm x 220mm.

Tooling Bars

To allow a PCB to be held in assembly equipment a component free zone of 0.200 (5.00) is required on two parallel sides. Where components are closer than this an additional 0.400 (10.00) waste area around the board edges, known as the tooling bar or tooling frame is introduced. Where components overhang the edges of the PCB the width of the bar may need to be increased appropriately. A 0.120 (3.00) chamfer on each external corner of the tooling frame assists with alignment through the assembly equipment and minimises the risk of jamming.

When assembly is complete the individual circuits are removed and the tooling bars discarded.

In some cases it may not be necessary for a full frame around the circuits, only tooling bars along top and bottom edges, see Figure 2. This can be done if there are no overhanging components along the leading edge. The body of the nearest component to the leading edge should be placed no closer than 0.100 (2.54).

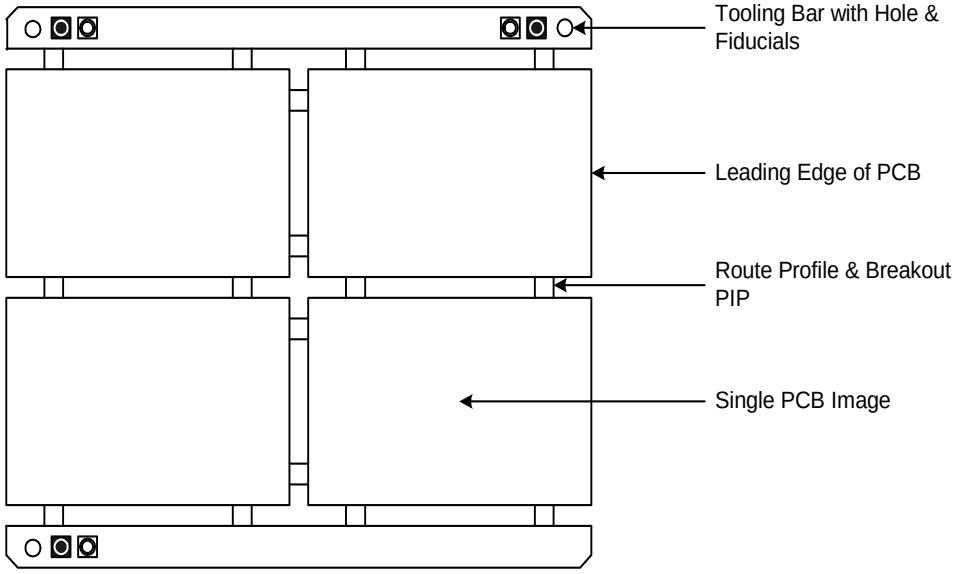

Figure 2 : Panel with Tooling Bars

Tooling Holes and Fiducials

Tooling holes are added to the tooling bars to accurately align the circuit board during the solder paste screen printing process. Panel fiducial marks provide overall alignment information for the SMT placement machines.

As standard three fiducials are added per SMT side plus three 0.120 (3.00) diameter non-plated tooling holes are located in three corners. Figure 3 shows the position of a tooling hole and a top and bottom fiducial with respect to the corner of a panel. Three fiducials are required to correct for scaling and rotational offsets.

Fiducial Design

*It is important that only three fiducials are used. This ensures that if a panel is accidently inserted into the placement equipment rotated through 180°, the equipment can detect it and halt assembly. *Fiducials must be at least 85mm apart .*

A panel fiducial consists of a copper pad with a clear solder mask window around it. To increase contrast and aid recognition a square backing pad is placed directly underneath on the adjacent layer.

The position of the fiducials must be present in the x, y placement data for the SMT placement programs or provided separately from the CAD system. The accuracy of this information will determine the quality of the final component placement.

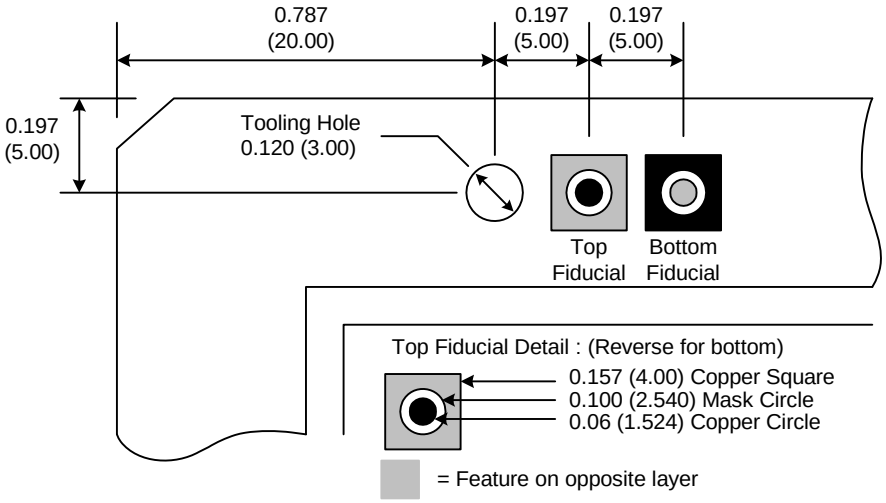

Figure 3 : Tooling Bar Features

Breakout Pips

To support the individual circuits during assembly, features known as breakout pips need to be added around the perimeter of the PCB profile. The spacing of these is dependent on the type of components being used; a board using a lot of large, heavy electrolytic capacitors may need more closely spaced pips. Typically pips spaced on 2.0-3.1 (50.0-80.0) centres will be adequate, but there must be at least two pips on each edge of each circuit for stability.

Pips must not be located beneath overhanging components at the edge of the PCB e.g. connectors, as this would obscure the pip thereby preventing access for the breakout tool.

Ideally any board features should be kept at least 0.200 (5.08) away from the edge of the board directly adjacent to a breakout pip. This includes components, traces, holes, vias and inner layer planes.

Where possible, any SMT components (0402, 0603, 0805 etc) located near to a pip should be positioned at 90° to the edge of the board. This minimises the possibility of damage to the devices due to the stresses imposed on the laminate during breakout.

When using breakout pips the profile of the board is routed out by the PCB manufacturer, omitting small sections to form the pips. The route tool diameter can be 0.063 (1.60) minimum with 0.079 (2.00) and 0.095 (2.40) as preferred sizes.

There are many types of breakouts used, but we have found the following types to give good results.

Solid Breakout Pips

Figure 4 shows an example of a solid breakout pip. It requires the use of a nibbler to remove the PCB from the panel. The advantage of using this type is that it uses slightly less PCB area than a perforated breakout and is stronger see later examples.

However, when broken out this type of pip leaves a small pimple of waste material on the edge of the board. This may require remedial filing to remove if the board edge is to be level, e.g. when the PCB is used with a card guide.

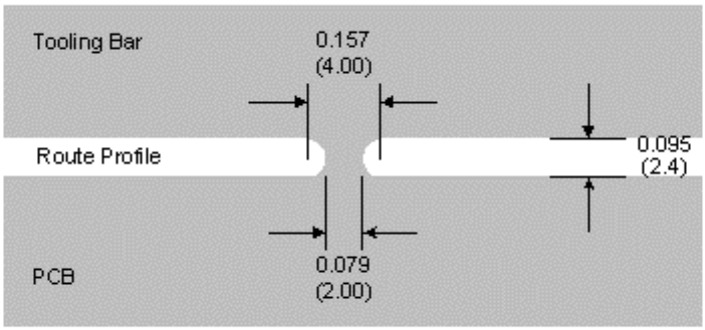

Figure 4 : Solid Breakout Pip

Perforated Breakout Pips

A perforated breakout, also known as a postage stamp breakout, shown in figure 5 has the addition of 4 x 0.040 (1.00) non-plated holes on a 0.060 (1.5) pitch centred along the PCB edge. For clarity only the edge that meets the tooling bar has been shown.

The advantage of this breakout is that it is easier to break than a solid pip and takes up minimal space on the PCB but it can leave a rough edge to the board.

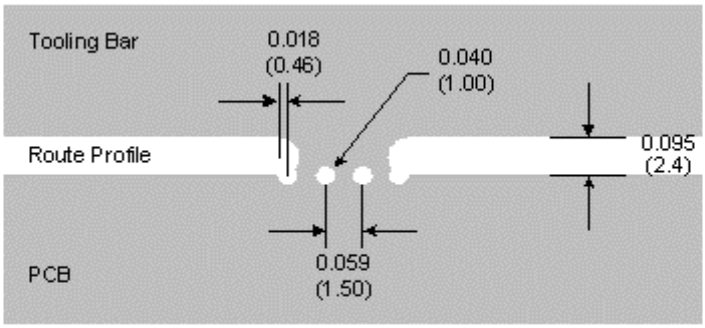

Figure 5 : Perforated Breakout Pip

To make the perforated pip breakout cleanly, the 4 non-plated holes can be offset into the PCB by 0.010 (0.25).

The holes can be further offset into the PCB so that there is no material left beyond the PCB edge. This is useful if the final assembly of the PCB has tight tolerances to the mechanical housing and no excess material outside the board edge can be tolerated. It may be necessary to ‘return’ the rout into the PCB to achieve satisfactory breakout.

For perforated breakout pips, components, tracks, vias and inner layer planes should be placed no closer than 0.100 (2.54) to the breakout holes.

Pip Keep-out Area

The pips are most effectively removed using a mechanical nibbler. To allow access for the nibbler there needs to be sufficient clearance around the pip that is free of components. The keep-out area is shown in Figure 6 below.

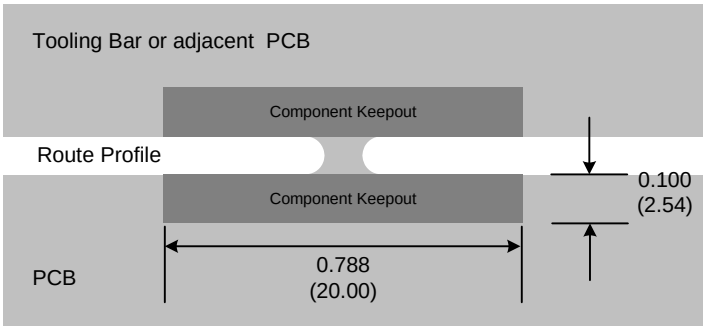

Figure 6 : Pip Component Keep-out

‘V’ Scoring

‘V’ scoring may be used as an alternative to breakouts for volume production and for boards with no overhanging components, e.g. connectors. This method is cheaper to implement on the PCB and is ideal for multiple rectangular boards on the same panel, see Figure 6.

It is very important to keep SMT components away from the edges of the board when considering the use of ‘V’ scoring as the force used in breaking the circuits out may place stress in the area alongside the score. The absolute minimum distance from a pad to the edge of the board is 0.079 (2.00). SMT components should also be aligned along the axis of the score.

The boards are scored by the PCB manufacturer and then broken out using a separating machine after assembly. This produces a rough unfinished edge to the circuit board, which is unsuitable for card guides, but is adequate for most applications.

The depth of the score varies depending on board thickness but the PCB manufacturer will normally advise on their preferred set-up for these.

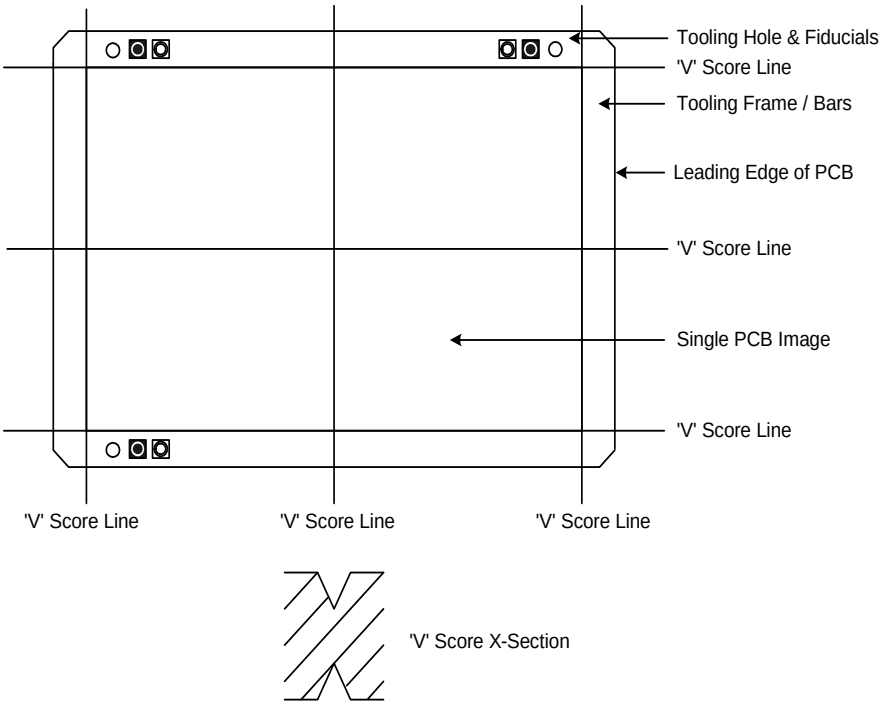

Figure 7 : ‘V’ Scored Panel

Non-Panellised PCBs

If a regular shaped PCB is not being panellised, provision needs to be made along the longest edges of the PCB for it to be held in the SMT assembly equipment. No components, tracking or other mechanical feature must come within 0.200 (5.00) of the edge of the PCB.

In this case 0.120 (3.00) tooling holes must be added within the PCB outline in two opposite corners to allow for screen printing alignment. There must also be a component free circular area around the tooling holes of 0.236 (6.00) diameter centred on the tooling hole. For SMT boards, fiducials in three corners on each side of the PCB need to be present for component alignment purposes.

No overhanging components should be present on the leading edge of the PCB.

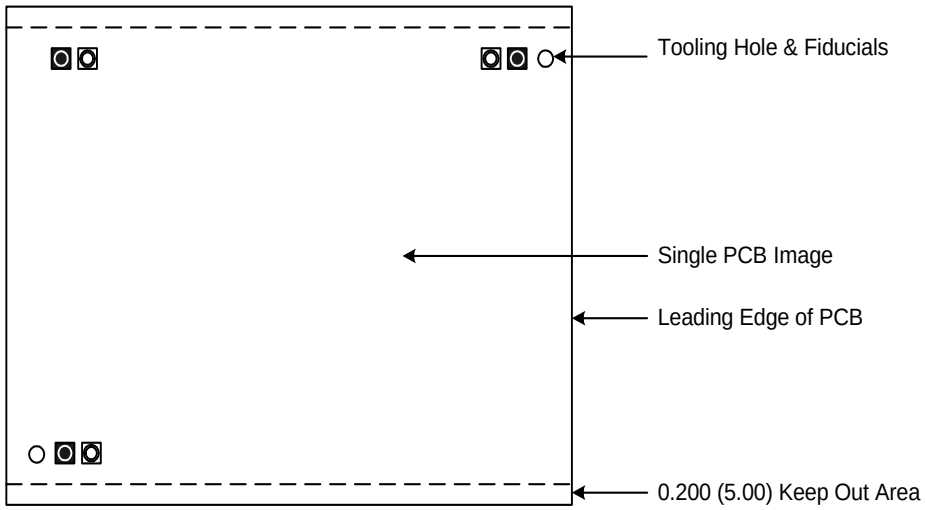

Figure 8 : Non Panellised PCB with Tooling Strips

Irregular Shaped PCBs

Irregular shaped PCBs can be made 'equipment friendly' by panellising. Rectangular boards with irregular cut outs, for example PCI boards, should have waste pieces added to fill the gaps.

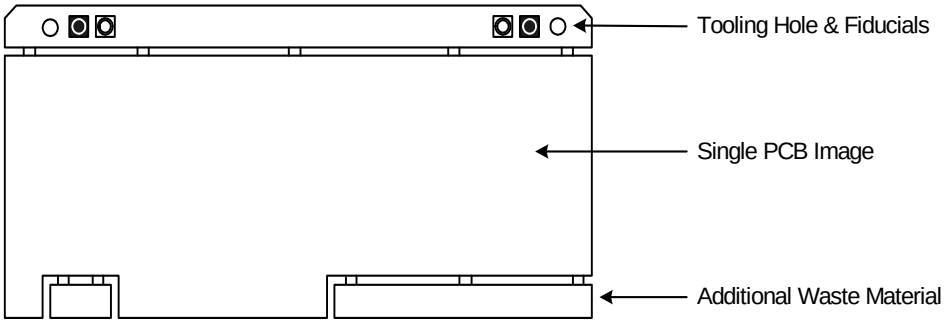

Figure 9 : Example of PCI Card with Infill Panels

Combined Rout and 'V' Score

Where the shape is even more irregular, boards can also be panellised using a combination of rout and 'V' score. By careful interleaving of the PCB's, the laminate usage can be optimised. In Figure 10, alternate circuits have been rotated by 180°. Additional waste material in the centre of the panel is necessary to allow the rout tool to over run to achieve the correct board shape without damaging adjacent boards. Typically this would be .300 (7.62) to .400 (10.16) wide.

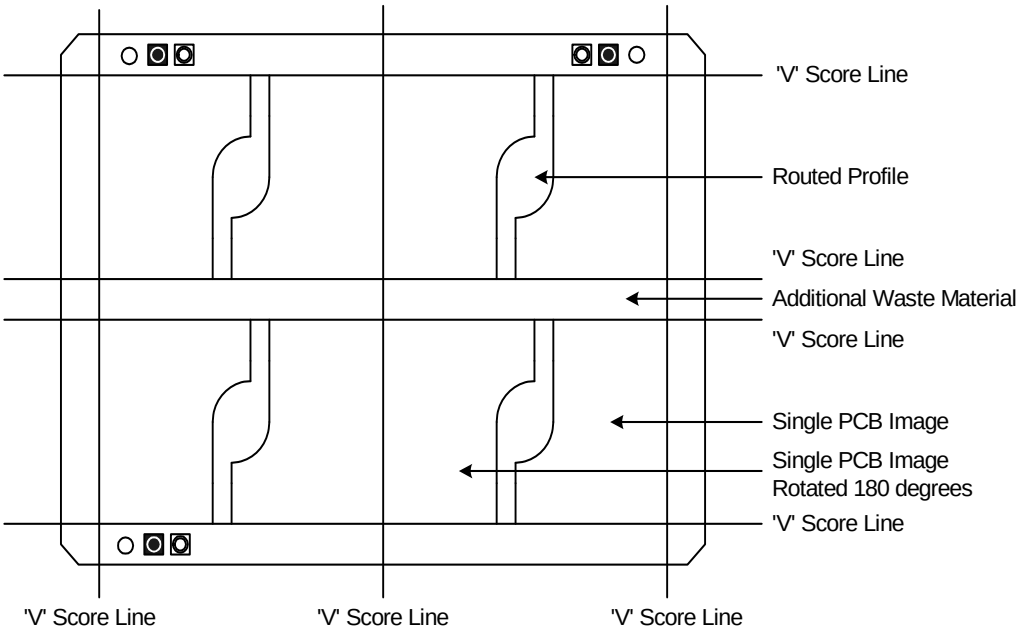

Figure 10 : Combined 'V' Score and Routing

Revision History

Date	Revision/ Issue	Author	Description of Revision
09-Oct-03	1	Martin Baker	First Issue
10-Oct-05	2	Pete Lomas	General Update
26-Oct-05	3	Pete Lomas	Added Pip Keep-out detail
16-Jan-08	4	Ian Wilson	Style Update
28-Jun-09	5	Rupert Lindley	General Update
12-Jul-13	6	Ian Wilson	Style Update

Disclaimer and Copyright

This technical note is supplied free of charge on an 'as is' basis. No warranty, expressed or implied is given. Norcott Technologies Limited will not accept any claim for loss, consequential or otherwise, arising out of the use of this document.

We are constantly striving to improve the quality of our technical notes. If you find an error or omission please let us know. Email us at: technical@norcott.co.uk