

# ENIGMA INTERCONNECT INC.

## CERTIFICATION

- ISO 9001-2008
- UL approved File E64339
- CGP (Military)

**LAYERS** 1 to 16

## MATERIALS

- FR-4 (1/2oz – 6 oz Cu)
- FR-4 170 (RoHS/Pb-Free)
- Polyimide
- Teflon
- Arlon
- Rogers
- Panasonic R1755 (RoHS High Td)
- T-Clad Metal Core
- Others (please inquire)

## DRILLING & BOARD PROFILING

- Blind & buried via
- Controlled depth drilling
- Countersinking
- Routing
- V-Groove scoring
- Milling
- Beveling

## SOLDER MASK / VIA PLUGGING

- Liquid Photoimagable (LPI)
- Green, matte green, blue, black, red, white, clear, other
- Thermal Solder Mask
- Peelable Mask
- Via plugging

## IDENT / SILKSCREEN

- White, yellow, black, red, other
- LPI or Thermal

## BOARD FINISHES

- Hot Air Solder Level (HASL)
- Immersion Gold on Electroless Ni
- Electrolytic Gold (Hard Gold)
- Gold Edge Fingers (board edge)
- Selective Hard Gold (not on edge)
- Wire-bondable Gold
- Immersion Silver
- Carbon Paste
- Immersion (White) Tin
- Board Edge Plating

## ELECTRICAL TESTING

- Grid Test (dedicated fixture) or Moving Probe (fixtureless)
- Up to 250V electrical test

## SPECIAL TECHNOLOGIES

- Controlled Impedance

## MINIMUM DESIGN LIMITS (mil)

|                                   | Proto-type | Standard | Recommended |
|-----------------------------------|------------|----------|-------------|
| Finished Hole Size                | 6          | 10       | 13          |
| Annular Ring IPC Class 2          | 5          | 6        | 7.5         |
| IPC Class 3                       | 7          | 8        | 9           |
| Via copper pad (hole + ann. ring) | 16         | 20.5     | 26          |
| Trace / Space ½ oz base Cu        | 4 / 4      | 5 / 5    | 6 / 6       |
| 1 oz base Cu                      | 5 / 5      | 7 / 6    | 8 / 8       |
| 2 oz base Cu                      | 8 / 8      | 10 / 9   | 12 / 12     |
| Inner layer copper to hole edge   | 10         | 12       | 15          |
| Soldermask clearance              | 2.0        | 2.5      | 5           |
| Thermal Ident line (std process)  | 6          | 6        | 6           |
| LPI Ident line (opt process)      | 5          | 5        | 5           |
| Scoring board edge to copper      | 20         | 25       | 25          |
| Routing board edge to copper      | 0          | 5        | 10          |
|                                   |            |          |             |
|                                   |            |          |             |

### Note:

- 1 All boards manufactured to IPC Class 2 unless otherwise specified.
- 2 Use "RECOMMENDED" values or greater if design density permits to avoid cost penalties.
- 3 Designs which do not meet minimum requirements must be reviewed by our manufacturing engineers.

## DATA FILE FORMAT

Gerber Data - RS274X (Extended Gerber – preferred); ODB++; RS274D  
Drill Data – Excellon, ASCII

We are capable of converting or regenerating data (design dependant) from:

- AutoCAD (DWG, DXF)
- Acrobat (PDF)
- Picture (JPG, TIFF, other)
- Transparency Film artwork
- and more

## DESIGN VERIFICATION

Each design is routinely checked to identify many design issues such as:

- Broken connections
- Unterminated traces
- Missing solder mask clearances
- Missing pads or holes on through hole padstacks
- and more

...however, we cannot be held liable for failing to identify design issues.

8070 Winston Street  
Burnaby, British Columbia, V5A 2H5, Canada  
Tel: 604.420.3313 Fax: 604.420.7525

<http://www.enigmacorp.com> e-mail: [boards@enigmacorp.com](mailto:boards@enigmacorp.com)

## **ENIGMA PCB DESIGN RECOMMENDATIONS**

### **README / MANUFACTURING INSTRUCTIONS / FABRICATION DRAWING / SLOTS & HOLES**

Manufacturing instructions may be supplied in a separate Readme file or drawing, or part of the drill drawing Gerber file. Enigma requires the following information:

- Stackup information, identifying each layer name/function (identify the Gerber file name and position of each layer in the stackup).
- Holes summary, clearly identifying all slots and non-plated holes as well as their locations and sizes (the standard hole position tolerance of Enigma is  $\pm 0.003$ "; if tolerances are specified by the customer, they are recommended to be symmetric, i.e. plus and minus values are the same).
- Board size and routing/scoring profile.

### **BOARD OUTLINE / ROUTING / SCORING PROFILE**

Enigma requires a board outline drawn as a closed polyline using 0.001" or thicker line, where the middle of the line represents the finished board outline. However, in the case where PCB design software cannot produce a precisely drawn outline, the customer is recommended to supply an additional drawing with dimensions, or to show dimensions in the Gerber file. The drawing, if provided, will take precedence over the drawn outline.

- It is suggested to provide a reference dimension from a non-plated hole to a board edge.
- Crop marks or layer registration marks are not required on each data layer however care should be taken to output all layers with a common offset.
- Board outline is required on only one layer (usually the fabrication drawing) instead of every layer.

### **SLOTS & NON-PLATED HOLES**

The customer is recommended to:

- Clearly identify all slots.
- Specify whether they are plated or non-plated.
- Identify all non-plated holes.

Slots or inner cutouts may be represented on the fabrication drawing, drawn with 0.001" or wider lines. The middle of the line will be used to rout out these features.

Note: Inner corner radius should be specified on fabrication drawing. Min. radius must be equal to or larger than the router bit radius (r0.047" preferred). Boards requiring smaller router bits can be manufactured upon request.

### **VIA**

The customer is recommended to use a unique drill size and pad size for via holes. For example, if the finished hole size for the via is 0.011" and there is a similar padstack in the design, the size of the via hole should be changed to 0.012". This will allow Enigma to identify all via holes for DFM checks and optimization during the tooling process. Provision of solder mask clearances on via pads is not recommended unless it is required by the design or testing process.

### **HOLES**

The customer is recommended to reduce the number of hole sizes (drill tool changes) by matching sizes of holes (except via holes which should be of a unique drill size) in the design wherever possible.

### **ANNULAR RING**

Annular rings should be 0.0075" and larger if the design density permits. While Enigma can manufacture smaller annular rings, teardrops will be required. They may be added by the customer or during the tooling process by Enigma.

## **TRACE / SPACE / COPPER WEIGHT**

Trace width and clearances are dependent on the copper weight. The customer is recommended to distinguish between base copper and finished copper. Please note that usually 1 oz copper is added to the outer layers and plated holes during the manufacturing process of double sided and multilayer boards. For example, if 1/2 oz base copper is specified for the outer layers, the finished copper will be 1.5 oz. Inner layers are not electroplated, and therefore their copper weight remains unchanged.

### **PLANE LAYERS**

- Wherever possible, flash (negative) planes instead of drawn (copper pour) should be created.
- If hatched fill is used orthogonal hatch (not diagonal) is preferred.
- Voids in hatched planes should not be smaller than 0.015"x0.015".
- The customer is recommended to ensure sufficient clearance from plane layers to edges of holes. The recommended clearance from the edge of a plated hole to any inner layer copper feature is 0.015" or greater (0.010" minimum).
- 0.010" plane clearance from copper features is preferred.

### **SOLDER MASK OPENING / SOLDER DAMS**

- When creating solder mask openings, consistency is important. If possible, different solder mask annular ring clearances should be avoided. We recommend that a solder mask pad should be equal in size to the copper pad (except fiducial). During the tooling process, Enigma will increase the solder mask clearance to the required size.
- If solder dams (distance between edges of solder mask openings for adjacent pads) are required, please note that the minimum solder dam width is 0.003". At least 0.008" should be allowed between the edges of adjacent copper pads to account for solder mask clearances when solder dams are required.

Note: Solder dams cannot be achieved on SMT devices of  $\leq 0.0157$ " pitch.

### **IDENT / COMPONENT LEGEND/SCREENPRINT**

The customer is recommended to:

- Use 0.006" line for all screenprinted text and graphics.
- Keep all screenprint features at least 0.005" away from solder pads.

Note: All features within 0.005" from solder pads will be clipped away to achieve IPC specifications.

### **GERBER & DRILL OUTPUT**

- The highest possible precision for file output should be used. At least four decimal places are recommended.
- Use the same output precision for Gerber and drill outputs.
- Create library footprints with high precision if the software, Allegro for example, has a separate setting for the library.
- All layers should be output as viewed from the top (component) side.
- Data output should NOT be set to mirror any layers.
- Supply all Gerber files registered to each other with the datum (working zero) at the lower left corner of the board. Do not 'center on page'.
- It is recommended to review all files in a Gerber viewer prior to sending to a manufacturer. If the design software does not include a Gerber viewer, GC-Prevue (a free Gerber data viewer) can be downloaded from <http://www.graphiccode.com>.

### **CONTROLLED IMPEDANCE**

If the customer would like Enigma to perform or verify the stackup calculation for impedance-controlled traces, the following should be supplied:

- The desired impedance value with tolerance.
- Trace width of impedance controlled traces.
- Board stackup, specifying the layer carrying the impedance controlled traces and reference layers.

### **GOLD FINGERS & SELECTIVE HARD GOLD**

- A plating bar connecting gold fingers together is not required.
- Minimum 0.030" clearance must be maintained between the edge of a gold finger or selective hard gold feature and any plated through hole.