

# Multi-Flex Circuits Aust.

## PCB DESIGN GUIDE

---

### Contents:

- [Base Materials](#)
  - [Laminate](#)
  - [Prepreg](#)
  - [Panel Size \(Utilization\)](#)
  - [Multilayer Layup](#)
  - [N.C. Drilling](#)
  - [Pattern design](#)
  - [Impedance control](#)
  - [Solder mask type](#)
  - [Legend](#)
  - [PCB Finishing](#)
  - [Gold Plating](#)
  - [Profiling](#)
  - [Final testing](#)
  - [Documentation](#)
-

# Multi-Flex Circuits Aust.

## PCB DESIGN GUIDE

---

### DESIGN GUIDES:

#### 1. Base Materials:

**\* Material sources:**

-Nan Ya Plastics Corp.

-Elite Electronics

**\* Material Types:**

-FR-4, Tg135• natural or yellow (UV block ) color, flame class, 94V-0,

MIL-P- 13949-G class II.

-Tetra-functional FR-4, Tg140•

#### 2. Laminate:

-Thin core thickness:

Min.: 0.005" preferred, 0.003" available.

Max.: 0.047"

-Copper foil:

Min.: 0.5 oz, 1/3 oz available.

Max.: 2 oz

-Unbalanced copper clad thin core, i.e. one/two ounce(s), is also available with more expensive material and running cost.

-Weight of the thickest copper for inner layers is 2 ounces and the thinnest is 1/2 ounce. 1/2 ounce copper core is not recommended for high reliability products and 2-ounce core is not recommended for internal signal layers.

-H.T.E.(High Temperature Elongation) and super H.T.E. copper clad thin cores are intended for high layer counts and high reliability products.

# Multi-Flex Circuits Aust.

## PCB DESIGN GUIDE

---

### 3. Prepreg:

unit: mil						
Type	106	108	2116	1506	7628	7630
NanYa Thickness	2.11	2.78	4.59	6.45	7.29	8.3
Elite Thickness	2.40	2.83	4.52	6.44	7.33	8.2

#### \*Application:

- For commercial and industrial grade products, one ply of high resin prepreg is recommended without compromising other quality concerns.
- Min. 3.5mil dielectric thickness shall be met at all times unless agreed by customers.
- Lay up shall be balanced and no more than 3 plies of prepregs placed in any dielectric layer.
- If no specific dielectric thickness is required, it is best to allow TPT to make the material selection. Selected materials will be utilized to meet industry standards, be of lowest cost, and apply to the most effective manufacturing methods.

# Multi-Flex Circuits Aust.

## PCB DESIGN GUIDE

### 4. Panel Size (Utilization):

- SHEET SIZES AVAILABLE:

36"x48 "	40"x48"	42"x48"
----------	---------	---------

- Panel BORDER and MARGINS:

X	Y	A	B (w/o G/F)	B (with G/F)
10.0~20.0"	10.0~24.0"			
0.25 increment	0.25 increment	0.65 "	0.1"	0.2"

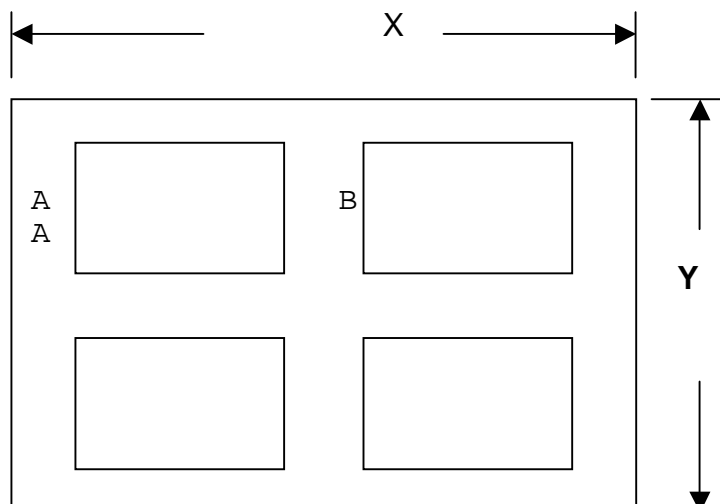
### -PANEL SIZE LIMITATIONS:

Panel width: 10 " ~ 20 " , increasing by the 0.25 inch.

Panel length: 10 " ~ 24 " , increasing by the 0.25 inch.

Minimum panel size: 12 " \* 13 " .

Maximum panel size: 21 " \* 24 " .



# Multi-Flex Circuits Aust.

## PCB DESIGN GUIDE

---

### 3.5. Multilayer Layup:

-Maintaining a balanced lay-up in relation to the Z-axis median of the board will assure minimum bow and twist. This balance includes the following:

- ♦ Dielectric thickness of layer
- ♦ Copper thickness of layers and its distribution
- ♦ Location of circuit and plane layers
- ♦ Outer Layer Circuitry
- ♦ Circuit area and distribution between the front and back of the board should be as balanced as possible.
- ♦ Plating thinning of low pattern density and cross hatching of external plane area should be considered.
- ♦ Thickness Tolerance
  - ♦ As the overall thickness of a multilayer board increases, the thickness tolerance should also increase. A good rule is to specify a tolerance of +/-10% of overall thickness.

### 3.6. N.C. (Numerical Control) Drilling:

Expense associated with drilling can be the second largest cost component of a PCB. Number of drill hits, stack height, and number of different drills selected are critical components of drilling.

#### 3.6.1: Stack height

-One shot drilling (drill size vs. board thickness):

Drill size \ Board Thickness	Board Thickness			
	<0.31 "	<0.68 "	<0.100 "	>0.100 "
0.20 " - 0.30 "	4	2	1	1
>=0.30 "	5	3	2	1

# Multi-Flex Circuits Aust.

## PCB DESIGN GUIDE

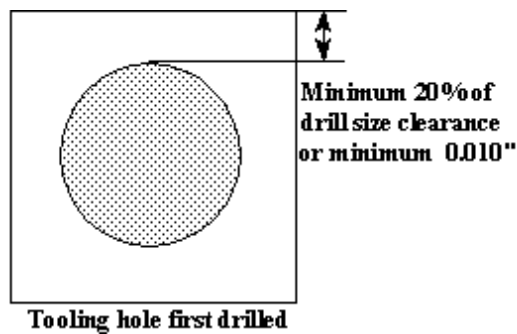
-Multiple step (PECK) drilling

Drill size \ Board Thickness	Board Thickness		
	<0.40 "	<0.68 "	<0.100 "
0.015 – 0.021 "	4	3	2
0.0098–0.0138 "	3	3	1

### 3.6.2: Tooling holes primarily drilled:

-To improve the accuracy of tooling holes and other non-PTH holes primarily drilled and to reduce cost, the following limitations shall be considered:

- Hole size shall be greater than 0.021 " and less than 0.200 " .
- Min. 20% drill size clearance of hole or min. 0.010 " around holes required.
- For elliptical holes, max. length shall be less than 0.200 " and width less than 0.090 " . Length shall not be greater than 2 times the width.



# Multi-Flex Circuits Aust.

## PCB DESIGN GUIDE

---

### 3.6.3: Hole size:

-The tolerance of a finished hole size is

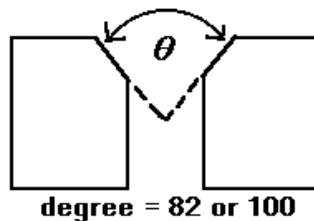
PTH:  $\pm 0.003$  "

NPTH:  $\pm 0.002$  "

Smaller tolerances are achievable by non-HAL finish and tighter process control.

-Min. drill sizes available are 0.0118 " for boards 0.062 " thick; 0.0098 " for boards less than 0.050 " thick.

-Countersunk tooling holes are available for 2 angles, 90 or 120 degrees, but are not preferred due to time consumed.



### 3.7 Pattern design:

#### 3.7.1: UL limitation:

-UL file no. E88441.

-Min. line width: 0.004 " .

-Min. edge width: 0.005 " .

-Min. board thickness: 0.015 " .

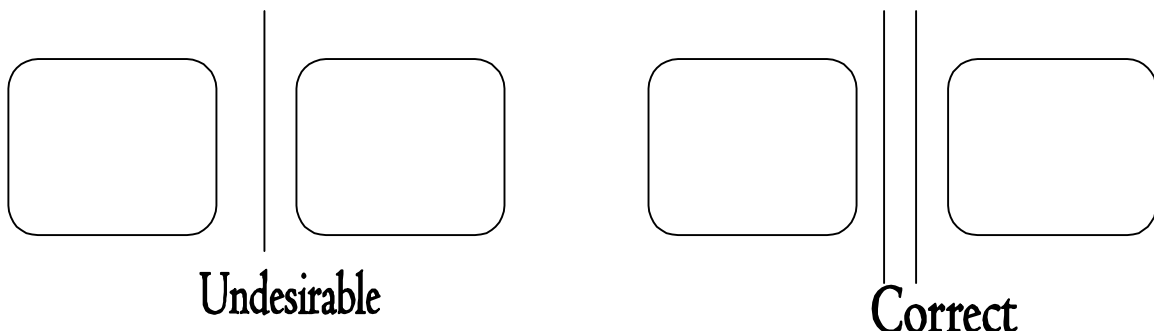
-Max. operating temperature: 105•.

-Flammability: 94V-0.

#### 3.7.2: Inner layer:

-Min. line width and spacing is 0.004 " for normal production and for limited production.

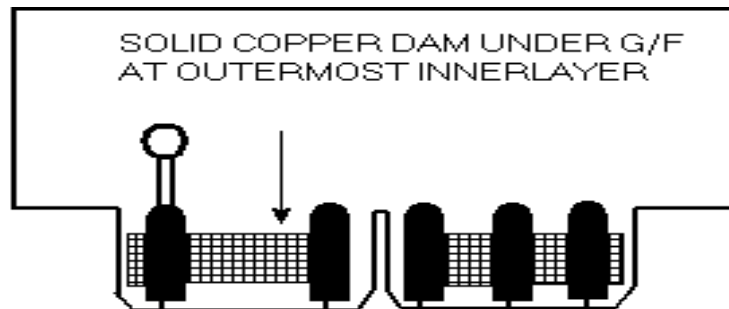
-Pattern lay-out shall follow design rule of equal space.(see diagram below)



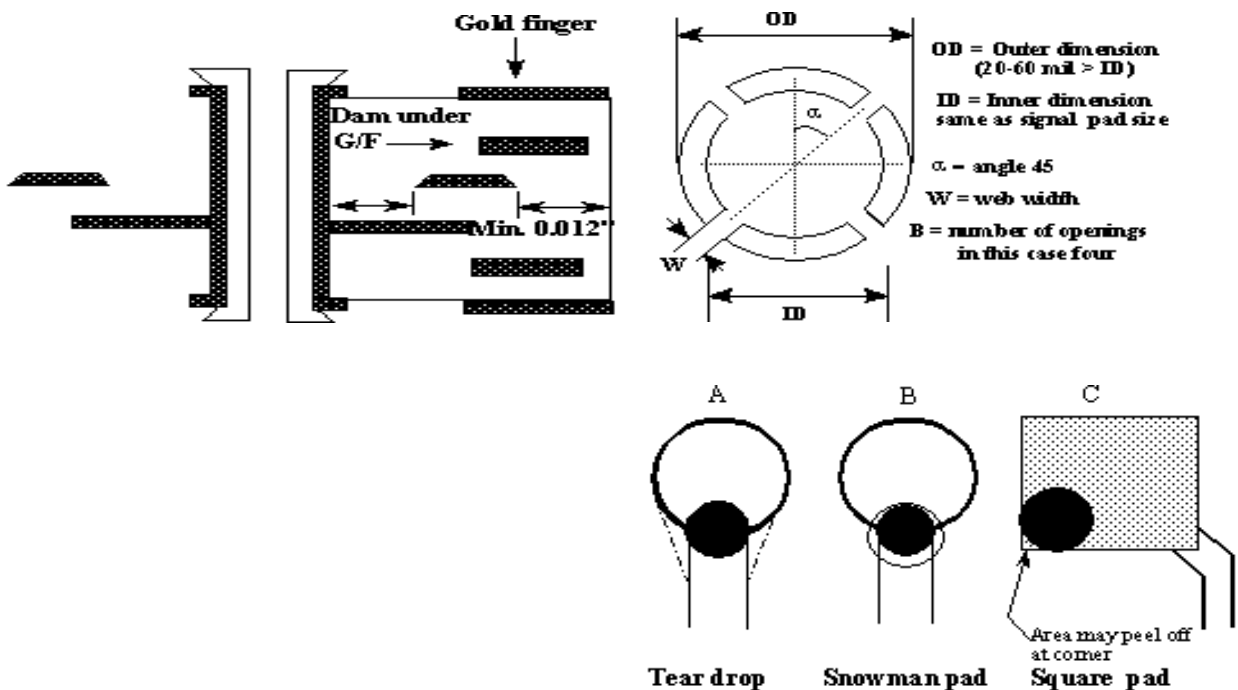
# Multi-Flex Circuits Aust.

## PCB DESIGN GUIDE

- Remove all non-functional pads to minimize drill bit damage.
- Prevent isolated space areas on outermost dielectric layers, copper dam under gold finger is preferred.
- Place solid copper dam under gold fingers to prevent lamination from wrinkling.
- Min. pattern tolerance is 0.001 " for both line width and spacing, the standard tolerance is +/-0.0015 " .



- 1 ounce copper is highly recommended, max. copper weight is 2 oz.
- For power/ground layers, min. web width is shown as below. Standard thermal relief design is preferred. Direct drill on plane is not recommended.
- Edge space (conductor to board edge or to hole edge) shall be at least 0.012 " .



### 3.7.3: Outer layer:

- Min. line width and spacing available is 0.004 " and 0.004 " is available.
- Pattern density should be as even as possible. Use "plating thieves " to balance the pattern density.



# Multi-Flex Circuits Aust.

## PCB DESIGN GUIDE

---

-Min. tolerance is  $+0/-0.001$  " for line width, and  $+0/-0.001$  " for spacing. Tighter tolerance is achievable by tighter process control.

-For big copper planes on outer layers, the clearance between hole and plane shall be  $0.008$  " min. Any gap between two copper planes shall be  $0.010$  " min.

### 3.7.4: SMT:

-Min. pitch available is  $0.016$  " with minimum  $0.060$  " in length. Finer pitches are also available but limited to non-HAL finish.

-Pattern layout is recommended to follow IPC-SM-782 design rules.

-Fiducial marks shall be solid and continuous to get uniform solder thickness. It is recommended to put copper underneath fiducials on inner layer to prevent solder mask ghost images (UV light transmission through laminate)

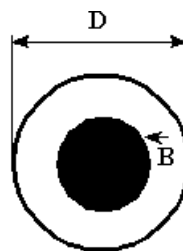
-Fiducial marks may also be used to show that panelized boards are good. In the case of a defective board, the fiducial mark can be crossed out, damaged or otherwise marked to show that the board is defective

-A solder mask dam should be added between SMT solder pads having a pitch of  $0.025$  " or more. For pitches below  $0.020$  ", solder mask dams will be more expensive.

-Solder leveling is not recommended when the SMT pad pitch is less than  $0.025$  ". It is recommended to use immersion nickel/gold, Preflux or Entek

### 3.7.5: Pad size:

-To keep a min.  $0.001$  " annular ring, pad size shall be drill size plus  $0.014$  " for innerlayer and  $0.012$  " for outerlayer. Smaller pads are also available with more expensive cost



D: pad size  
A: drill size  
 $B = (D - A) / 2$   
 $B > 0.005$  " for  
interlayer  
 $B > 0.005$  " for

-Drill size = max. finished hole size plus  $0.002$  " or  $0.003$  " for solder leveling finish. For non-HAL finish, drill size is equal to max. finished hole size  $+0.001$  ".

-Outerlayer copper thickness is based on 1/2 ounce. When copper thickness is 1 ounce, then pad size shall be  $0.002$  " larger than that of 1/2 ounce.

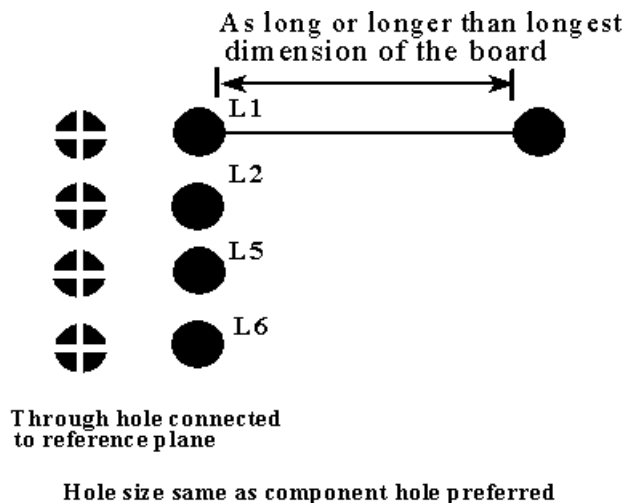
# Multi-Flex Circuits Aust.

## PCB DESIGN GUIDE

### 3.8 Impedance control:

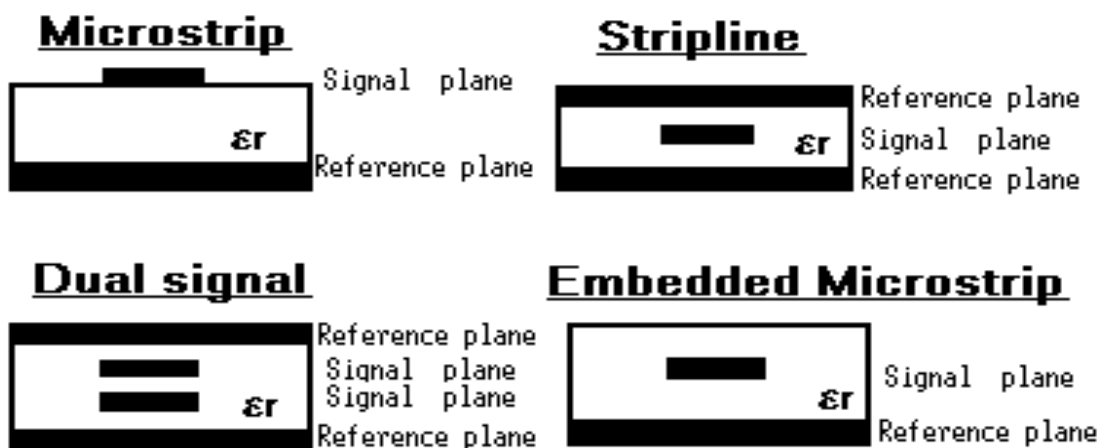
-Tolerance shall be  $\pm 8$  ohms (radius shall be  $\pm 3$  ohms). Tighter tolerance is achievable at higher cost.

-A test coupon should be placed within free area of finished board (if available). A separate coupon will increase the price.



-Signal length of the test coupon should be longer than 6 " to get a more accurate reading.

-We have 4 different constructions available.



# Multi-Flex Circuits Aust.

## PCB DESIGN GUIDE

---

### 3.9 Solder mask type:

#### 3.9.1: Available types:

-PC401: Screen-print Epoxy base.

-PSR4000(yellow & green): Liquid photoimagible solder mask.

#### 3.9.2: Solder mask thickness

Unit: mil

	Over trace	Over laminate	Over corner
PC 401	0.2-0.7	0.8-1.8	0.2-0.4
PSR4000	0.2-0.8	1.0-2.0	0.2-0.6
Ciba Geigy	0.2-0.7	1.0-1.8	0.2-0.5

#### 3.9.3: Via hole plugging capability:

-Max. drill size:

PC401	PSR4000	Ciba
0.030 "	0.025 "	N/A

# Multi-Flex Circuits Aust.

## PCB DESIGN GUIDE

---

### 3.9.4: Solder Mask Clearance and adhesion:

	Adhesion	Clearance
PC401	0.006 "	0.006 "
CIBA	0.004 "	0.006 "
PSR4000	0.004 "	0.006 "

Adhesion: Min. S/M strip width.

Clearance: Min. air gap to prevent S/M on pad.

### 3.9.5: Min. clearance between SMT and via:

PSR4000	PC401	Ciba Geigy
12 mil	20 mil	14

### 3.10 Legend:

-Color should be white, yellow or black. White is preferred.

-Font is optional. Min. character width shall be 0.005 " and min. character spacing shall be 0.008 " to keep a clear legible legend.

-Ink should be EPOXY or UV curing base.

### 3.11 PCB Finishing:

-Types available:

-Horizontal solder leveling.

-Entek.

-High temperature preflux.





-Immersion nickel/gold.

-Reflow solder.

-Selective solder

# Multi-Flex Circuits Aust.

## PCB DESIGN GUIDE

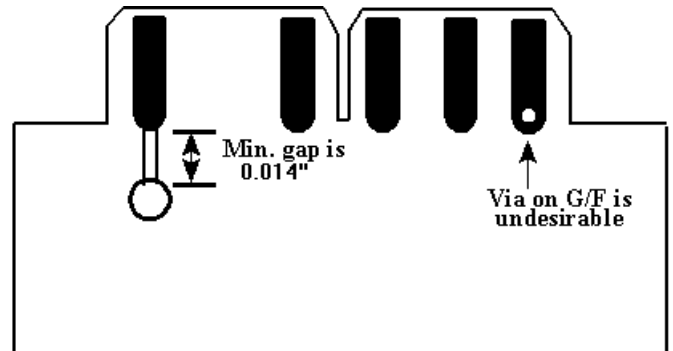
	SMOBC	Preflux/Entek	Reflow solder	Selective solder	Immersion gold
Coating Thickness	~300• "	~12• "	100~1000• "	Average:300~500• " Sigma:50~80• "	Nickel:120• " Gold: 4• "
Co-planarity					
Assembly Process	Wave soldering IR reflow	Wave Soldering IR reflow	Wave soldering IR reflow	Wave, hot bar soldering IR reflow	Wave soldering IR reflow COB.
Concern Points	Uneven solder thickness. Excess thermal stress Higher ionic Contamination Flux residue	Less solder wicking action for through hole. Print solder paste  Over all bare copper area.	Solder mask may peel off after soldering	Limited capacity. Longer lead-time. Complicated process	More expensive. Sensitive to baking
Recommend Application	Greater than 25milpitch SMT	Fine pitch SMT. Competitive product.	Low density product.	TAB	COB Fine pitch SMT
Cost	100%	-US \$0.4 – 0.7/SQ.Ft for Preflux and 0.5-0.8 US\$ for Entek	100%	+\$5 USD/SQft.	+US \$2/SQft

# Multi-Flex Circuits Aust.

## PCB DESIGN GUIDE

### 3.12 Gold Plating:

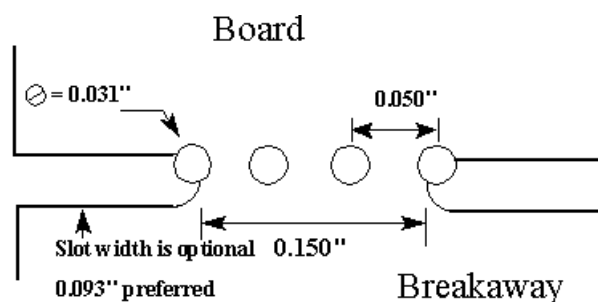
- Types available:
- Soft or hard dip tank gold plating.
- Gold tab plating.
- Panel immersion nickel / gold.
- Pattern design consideration of gold tab plating:



- Gold tab face outward as always.
- Min. board width of two end gold finger is 4.5" , if not, more expensive dip gold plating is required.
- Tip of gold finger to adjacent via shall be min. 0.014 " .
- Spacing of gold tab shall be no less than 0.010 " .
- Do not place via on top of gold finger.

### 3.13 Profiling:

- NC routing and blanking are both available.
- V-cut is also available but limited to only horizontal and vertical cutting.
- Standard router bit is 0.0984 " ,0.0945" ,0.0394" ,0.100"and 0.0787 " (Radius shall be at least half of router bit diameter plus 0.005 " , min internal radius 0.020").
- Unify slot width. Prevent different slot widths on same piece to increase productivity.
- Prevent different radius of corners, because of the need of different router bits to meet nominal radius.
- Other router bits available are 0.062 " . Router bits smaller than 0.094 " will cost higher.
- Breakaway tabs may be designed as below to get smooth edge after break.



# Multi-Flex Circuits Aust.

## PCB DESIGN GUIDE

---

### 3.14 Final testing:

- Test fixture is generated by internal CIM computer working files.
- Test program is net list testing generated in house.
- IPC-365 test format is highly recommended to prevent common defects and to minimize the set up time.
- Will mark edge of boards to indicate that this board has been tested unless otherwise instructed.
- Required min. 3 first drilled N-PTH holes with diameters greater than 0.100 " for O/S testing.

### 3.15 Documentation:

- Documents shall be complete, clear, updated and easy to understand. Basic element of documents shall include:
- Type, shape and size of the board and tolerance conditions thereof.
- Dielectric separation between layers .
- Overall board thickness and tolerance.
- Marking locations, type, size and content.
- Hole sizes, tolerances, plating or not and NC drill files.
- Layer sequence.
- Color of legend and type of solder mask.
- Plating thickness and type.
- Reference acceptance criteria and specification.
- CAD File:
- Both floppy diskette and tape are acceptable.
- Files may be transferred to Multi-Flex by Email at [sales@mfca.com.au](mailto:sales@mfca.com.au)
- Aperture list shall be clearly defined. Special patterns shall be illustrated in clear sketch.
- CAD format, code (ASCII, EBCDIC, EIA...), coordinates (incremental, absolute), unit (inch or metric), zero omitted (leading or trailing) and number format shall also be clearly defined.
- Reference paper hard copy or mylar film copy of each layer artwork is highly preferred to prevent any potential data error.