Modular Components National, Inc.

Design Guide

TABLE OF CONTENTS

105 E. Jarrettsville Road, Forest Hill, MD 21050 Phone: 410-879-6553 Fax: 410-638-7356
Contact: sales@modularcomp.com

I. SUBSTRATE MATERIALS

High frequency designs (1 GHz and up) require materials with closely controlled dielectric constants and dissipation factors. The materials normally used in the printed circuit industry, epoxy or polyimide based, do not have the desired controlled characteristics.

The substrate materials used for high frequency applications are based upon PTFE resin formulations that have the desired properties, i.e. dielectric constant controlled to +/- .04 and dissipation factor to .0004. These values may vary somewhat depending on the material type and supplier. Some of the newer materials on the market do not contain PTFE resin, but still have controlled values that can be used for high frequency applications.

1. MATERIAL TYPES

There are three basic material types: non-woven glass, woven glass, and filled.
Military Specification designations are: GR (non-woven) and GT, GX, GY (woven).
Filled material has no designation.

A. Non-woven materials contain a dispersion of glass microfibers in the substrate. These are typically materials with low dielectric constants (2.20 - 2.35). They work very well at the higher frequencies, as the dissipation factor is low.

B. Woven glass materials are made using fine weave glass cloth. These materials have dielectric constants in the range of 2.40 to 2.60. Due to the glass cloth, the dissipation factor may be somewhat higher.

C. Filled materials have dielectric constants ranging from 2.94 to 10.8. The filler material may be a ceramic or other suitable material that can be used to raise the dielectric constant. These materials may also contain non-woven or woven glass. The type of filler and construction determines the dielectric constant as well as the dissipation factor. These materials tend to have a higher dissipation factor than the materials mentioned above.

2. COPPER FOIL TYPES

Two types of bonded copper foil are available. Electro-deposited (ED) and rolled-annealed (RA). The difference between these foils is the fabrication process and the treatment on the backside to enhance bonding adhesion.

A. Electrodeposited copper (ED) is made by electroplating onto a rotating drum that results in a vertically oriented grain structure. Opposite the drum surface, the copper plating is rough which is almost like “teeth” which is enhanced by further treatment. The “teeth” provide higher peel strength, but may increase conductor losses at high frequency because of the depth of the "teeth".

B. Rolled-annealed copper (RA) is made by rolling an ingot to very thin foils. The grain structure is horizontally oriented. The rolling causes stress in the foil and must be heat annealed to remove the stress. Rolled-annealed copper has the same finish on each side, which requires "teeth" artificially created on the backside. These "teeth" are about 1/2 the depth of the electrodeposited copper. This causes rolled-annealed copper to have about 1/2 the peel strength of electrodeposited copper. This means rolled-annealed copper performs better at frequencies above 13 GHz.

C. Peel strength is the amount of force, in pounds per inch, which must be applied to cause a 1" wide strip of the copper foil to peel from the substrate. Typical values for electrodeposited (ED) 1 ounce copper are 9-11 pounds per inch and rolled-annealed (RA) 1 ounce copper is 4-6 pounds per inch. As circuit traces become narrower the peel strength is reduced. Therefore, more care is required to solder and bond to narrow traces.

3. COPPER FOIL WEIGHT

Copper foil is designated by weight per square foot. Therefore, 1 ounce copper has 1 ounce of copper per square foot of surface area. This equates to approximately .0014” thickness. Copper foil is available in the following weights:

Foil Weight	Approximate Thickness
1/4 ounce	0.00035"
3/8 ounce	0.0005"
1/2 ounce	0.0007"
1 ounce	0.0014"
2 ounce	0.0028"

Thinner copper allows tighter etching tolerances (see ETCHING). However, copper thinner than half ounce requires special attention while processing and it also is more expensive.

Many times, when plated through holes are required, 1/2 ounce copper is used and plated up to the equivalent of 1 ounce. 1/4 and 3/8 ounce copper allow for tighter etch tolerances, but must be processed more carefully to obtain good yields.

4. SPECIFYING MATERIAL

When specifying substrates for use in microwave applications, the following information, if there is a vendor preference, should be stated on your blueprint or P.O.

A. Material type: woven, non-woven, or filled.

B. Dielectric constant

C. Substrate thickness

D. Copper foil weight and type

The following vendors all supply materials for the microwave industry:

Arlon Materials For Electronics Division 1100 Governor Lea Road Bear, Delaware 19701	Phone: 800-635-9333 Fax: 302-834-2574
Rogers Corporation Microwave and Circuit Materials Division 100 S. Roosevelt Avenue Chandler, Arizona 85226	Phone: 877-643-7701 Fax: 480-961-4533
Taconic Advanced Dielectric Division P.O. Box 69, Coonbrook Road Petersburgh, New York 12138	Phone: 800-833-1805 Fax: 800-272-2503

II. IMAGING

Imaging is a photographic process that requires circuit design data to be converted and plotted on silver film. The film is then used to expose the circuit image onto the substrate. It is very important that correct etch factors are applied to the artwork before the imaging process. We prefer to have the data received with no etch factor applied.

In the electronic world of today it is most common to receive data files that contain all the information necessary to plot the circuit design data onto film. This section will only discuss the use of this type of data. If you prefer to supply film, please send a set of both negative and positive films on .007" stable base polyester and contact MCN for the correct etch factor information before plotting. Quality parts cannot be manufactured from poor artworks. Therefore, it is critical that customer supplied film meet the requirements of the drawing and allow for manufacturing tolerances.

1. METHOD OF RECEIVING DATA

Data can be sent by e-mail, FTP, or customer download site.

2. DATA FORMATS

AutoCAD
DXF
Gerber RS-274-X **(Preferred)**
ODB++
HPGL
IGES

If other formats are used, please contact MCN to be sure they can be converted.

3. DATA HANDLING AND PLOTTING

Once data is received it is converted, if necessary, to a format that can be used at our CAD stations. The data is reviewed to match the drawing. Please note that only the data/drawing accuracy can be verified, not the design accuracy. If the data/drawing accuracy is confirmed, the etch factor is added and the file is laser plotted on stable base film. If problems or discrepancies arise, the customer is notified by fax requesting disposition or clarification. The laser plotting machine uses a 1/4 mil or 1/8 mil pixel size.

4. FILM INSPECTION

Once the film is laser plotted, it is visually inspected for the following:

Accuracy of the image to the drawing
Quality of the image
Critical line widths and gaps
Accuracy of the step and repeat image (if more than one image is plotted)
Accuracy of the punched film to the panel and hole locations (if plated through holes are used)
Front to back registration

5. ANNULAR RINGS

A minimum of .010"of ring as measured from the hole edge or .020" overall larger than the hole diameter is recommended for isolated annular rings at the end of a plated through hole. This will result in a minimum .002" annular ring on the finished part. If a minimum annular ring of .005" is required then the film should be plotted at .030" overall above the nominal hole diameter plus the tolerance.

For example: A plated through hole of .030" diameter with a .002" minimum annular ring would require a pad size of .050" (.030"+.020"). If the minimum annular ring is .005" then the pad size should be .060" (.030"+.030").

Using unsupported plated through holes (no pad) is not recommended. Unsupported plating is not mechanically strong enough to prevent handling and assembly damage. In addition, unsupported edge plating of cutouts and cavities is also strongly discouraged.

6. FRONT TO BACK REGISTRATION

Front to back registration of imaged features is important for many microwave applications. Manufacturability is restricted for front to back registration tolerances less than 0.002" as well as for critical features spread over a large area.

Circuit traces that terminate at the edge of the part should be set back a minimum of 0.002" from the part edge or cutout for ease in manufacturing

7. IMAGING PROCESS

The application of the image to the substrate requires the use of photoresist. Photoresist is a material that is sensitive to high intensity UV light. The photoresist is applied to the substrate using heated rollers and light pressure. The film is then pinned to the substrate and placed in the exposure unit where a vacuum is drawn and both sides of the substrate are exposed to a high intensity UV light source. The UV hardens the exposed areas, while the unexposed areas are left to be developed away. The next step is to develop the image in a conveyorized machine using a solution to remove the unexposed photoresist. The exposed photoresist is not affected. The substrate is rinsed with water and dried. The substrate is now ready for further processing, either etching or plating depending upon the process required to complete the board.

The type of film used is dependent upon the process being used. For example, a board that simply requires etching will be exposed using a negative film while a board with plated through holes will be exposed using a positive film.

8. DEFINITIONS

Positive film - the black area on the film represents copper as it would appear on the finished board. Clear areas represent no copper on the finished board.

Negative film - the black area on the film represents no copper as it would appear on the finished board. Clear areas represent copper on the finished board.

UV - ultraviolet

Etch factor - a small correction applied to line widths and gaps to compensate for the copper removed during etching. This factor is added so that the final line or gap width will be as close to nominal as possible.

III. PANEL SIZES

Consideration during the design phase must be given to the final board size as it relates to the panel configuration that will be used. This increases substrate utilization and minimizes costs. While some designs require a specific board size due to system constraints, the design engineer should consider optimum utilization of substrate sizes.

Substrate materials are available in the following sheet sizes and vary by supplier.

12" x 18", 18" x 24", 18" x 36", 18" x 48", 10" x 10", 10" x 20", 20" x 20", 24" x 36", and 36" x 48".

Typically, non-woven substrates are available in 12" and 18" widths, and ceramic filled substrates are available in 10" and 20" widths. Most woven substrate materials are available in 24" or 36" widths. Additional information concerning sizes may be obtained by contacting MCN before making a final decision.

Panel sizes are chosen to obtain even cuts from the vendor supplied sheet. For example, a 9" x 12" panel cuts evenly from nearly all sheet configurations except the 10" and 20" versions. The same is true for a 12" x 18" panel. The smallest panel size that can be processed is either a 6" x 6" or 5" x 5" depending upon the substrate.

Typically, panel sizes used in production are 9" x 12", 12" x 18", or 10" x 10". These sizes are chosen to minimize the dimensional changes that occur during processing. MCN are able to process panels 18” x 48” for special applications.

1. <u>PANEL LAYOUT</u>

MCN completes the layout of the panel. Usable panel size requires a 1/2" border around the panel for tooling holes and plating rack clamps. This reduces the usable panel size by 1" in both directions. For example: The usable panel size of a 9" x 12" panel is 8" x 11".

Part spacing - within the usable panel size the spacing of parts is determined by the process required to remove the parts from the panel.

A. Laser cutting - recommended spacing between parts is 0.030"

B. Routing or machining:

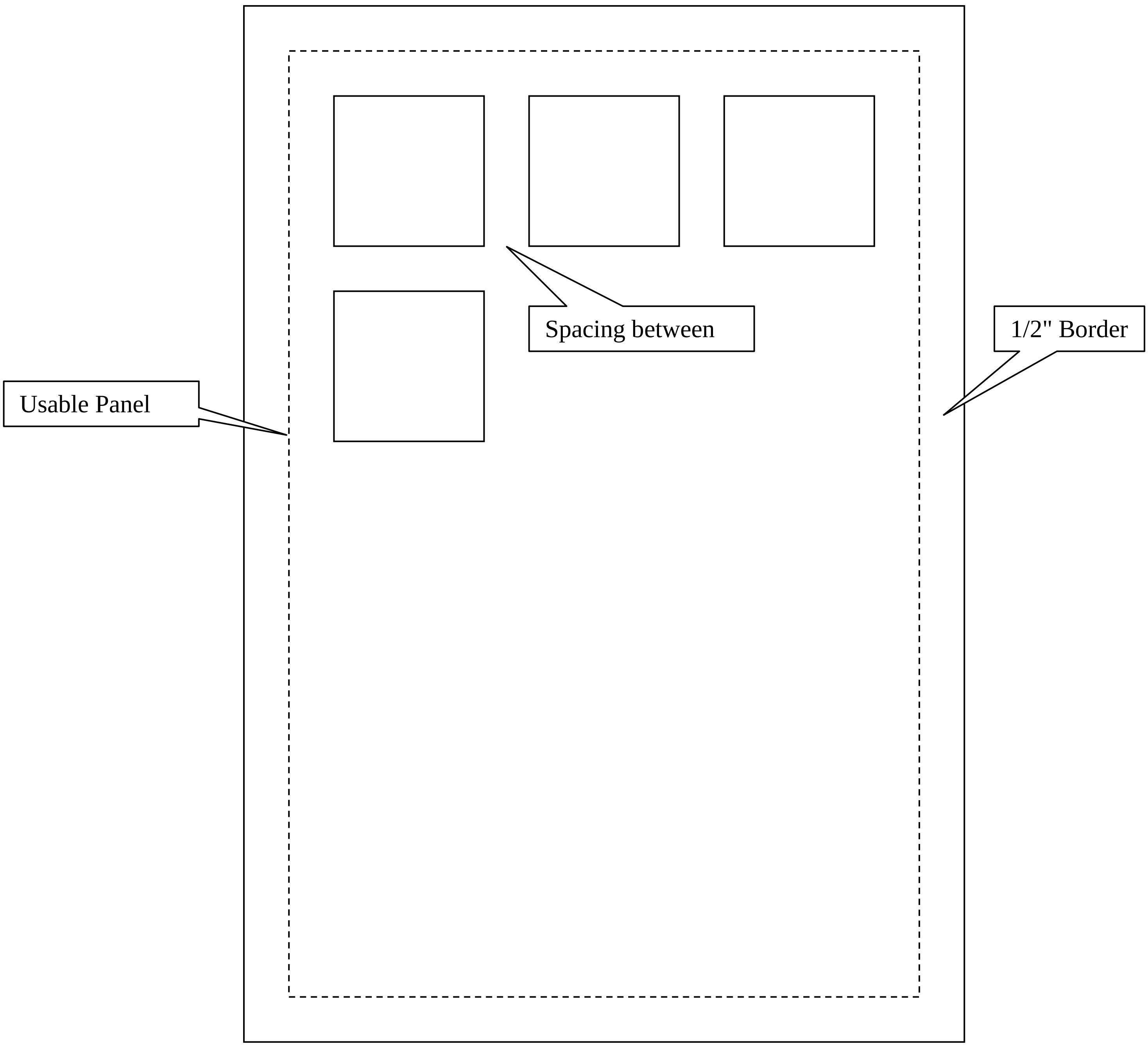
Modular Components National, Inc.

Design Guide

1) Unsupported Boards - recommended spacing between parts is 0.150"
2) Supported boards - recommended spacing between parts is 0.250"
3) Edge plating requires a spacing of 0.250" along the edge to be plated.

If you have questions concerning panel layout, please contact MCN

2. PANEL LAYOUT EXAMPLE



105 E. Jarrettsville Road, Forest Hill, MD 21050 Phone: 410-879-6553 Fax: 410-638-7356
Contact: sales@modularcomp.com

IV. ETCHING

Etching is the process of removing the excess copper foil from the surface of the substrate. This is done by chemically dissolving the copper into the etching solution.
Either the photoresist or the plating on the circuit is used as an etch resist to protect the circuit during the etching process. The etchant chemistry is chosen so that the process does not attack either the photoresist or the final plating.

1. ETCHING TOLERANCES

Etching tolerances are dependent upon the copper thickness being etched. Typically, 1/2 ounce copper, with an approximate thickness of 0.0007", will have an etch tolerance of +/- .0005". One ounce copper, with an approximate thickness of 0.0014", will have an etch tolerance of +/- 0.001". The etch tolerance will vary with the manufacturing process. Boards with plated through holes will require more tolerance than a board that is processed by a print and etch method.

In general, the thinner the copper the tighter the etch tolerance. However, tighter etch tolerances reduces yields and impact costs.

The etch factor chosen for the film is used to compensate for the copper removal during the etching process. These values change based on the specific process.

Questions concerning etching tolerances should be directed to MCN during the design phase to assure tolerance exceptions and economic feasibility.

2. CIRCUIT LINE AND GAP WIDTHS

Circuit line and gap widths of 0.004" and larger do not affect manufacturability and are considered standard features. Line widths and gaps below 0.004" do affect yields and board cost. The minimum line width and gap that can be processed is 0.002" Designers should consider the tradeoffs between minimum line and gap widths and manufacturability, particularly if the program is going to require large volumes of boards.

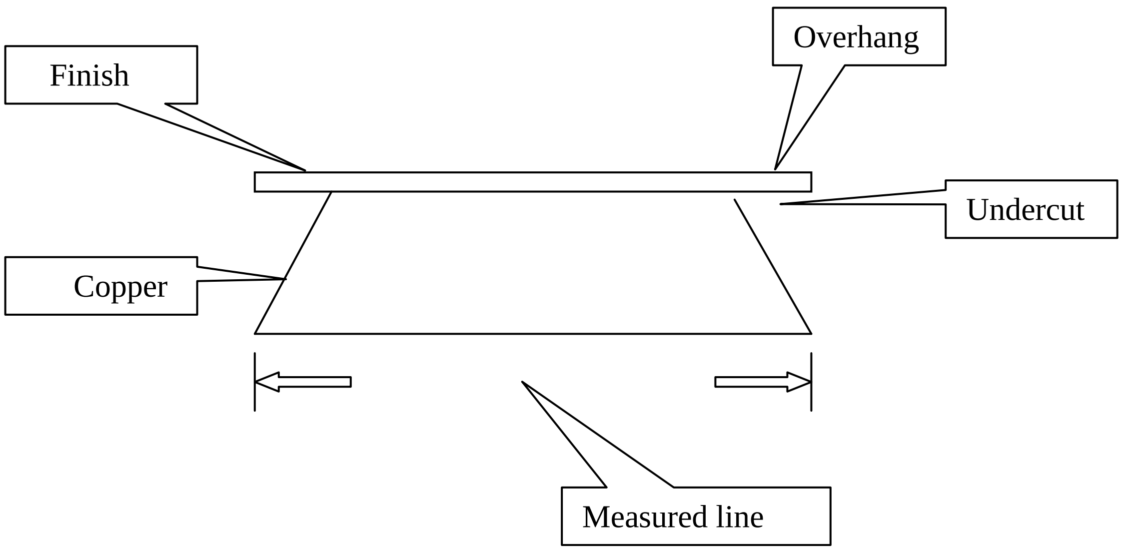
General Guidelines:

A. Gaps are more difficult to image and etch than lines. For high circuit density applications, specify gaps larger than the associated line widths.
B. Isolated fine lines with large associated gaps are easier to etch than clusters of fine lines grouped together.

C. Sharp corners where lines change direction are more difficult to image and etch than more gradual curves or 45 degree angled corners.

3. LINE AND GAP WIDTH MEASUREMENT

In microwave applications, measurement of line or gap widths is always done at the base of the line or gap. (See illustration below)



V. PLATING

Microwave circuit boards generally require some type of final plating on the circuitry although in some instances the final finish is bare copper. This section addresses the plated through hole process and the various final plating finishes available.

Substrates containing PTFE resins require special treatment prior to plating, since PTFE resin is by nature a non-wettable surface. The substrate surface is activated with a sodium etch, or alternately a plasma etch. Either method converts the non-wettable surface to one that accepts various cleaning and plating solutions.

1. ELECTROLESS COPPER

After drilling or machining, the panel is sodium or plasma etched to make the surface wettable. Following a series of cleaning and prepping solutions a three step process electroless deposits a thin layer in holes or on edges. The process consists of a catalyst that seeds the surface followed by an accelerator and then the electroless copper solution. The electroless copper deposits a thin layer (10-20 microinches) that covers the hole walls, edges, and surface of the panel. As the name implies, this solution requires no current flow. This process is followed by an electroplating step to deposit additional copper (up to 200 microinches).

2. COPPER PLATING

Electroplated copper builds up copper thickness in the holes to the 0.001" minimum industry standard. The minimum thickness in the hole can be varied from 0.0005" to 0.0015" as required. Thinner copper allows for better control of etching tolerances. Typically the copper is pattern plated, that is, only the circuit and holes are plated, with the background copper, protected by photoresist, remaining unplated. Electroplated copper is governed by MIL-C-14550B.

3. TIN/LEAD PLATING

Tin/lead plating is an alloy of tin and lead with the composition ranging from 55% to 65% tin to lead. This deposit may be used as plated or fused. When reflow is required, 60/40 + /– 4 should be used. The as plated surface is flat where the fused surface is mounded. Typical specified thickness is 0.0003" to 0.0005".

Another process used with tin/lead is Hot Air Leveling. A thin coating of tin/lead is applied on the circuitry (approximately 0.0001") and then leveled using hot air.

Tin/lead plating is governed by MIL-P-81728B.

105 E. Jarrettsville Road, Forest Hill, MD 21050 Phone: 410-879-6553 Fax: 410-638-7356
Contact: sales@modularcomp.com

4. NICKEL PLATING

Two types of nickel are plated, electroless nickel and electroplated nickel.

Electroless nickel is generally the first step in plating aluminum. It can also be used as a final finish on aluminum backed parts. Typical thickness is 50 to 1000 microinches.

Electroplated nickel is used as a barrier between copper circuitry and gold. The electroplated nickel is plated from a sulfamate nickel solution. When plated under gold, nickel serves two purposes: (1) as a barrier against copper migration and (2) as a hard surface under the gold for wear purposes. Typical thickness is 50 to 200 microinches.

Electroless nickel is governed by MIL-C-26074E

Electroplated nickel is governed by QQ-N-290A Class I

5. GOLD PLATING

Only Type III gold, which is 99.99% pure, is plated. Type III gold has a Knoop hardness of 90 maximum and lends itself well to wire or ribbon bonding processes. Type III gold is solderable; however, it suggested the deposit thickness be kept below 75 microinches to minimize gold embrittlement problems.

Gold plating is governed under MIL-G-45204B. This specification covers the Type, Grade, and Class to specify the final thickness and type.

Gold can be plated in thicknesses varying from 10 microinches to over 200 microinches. Consideration should be given to the cost of the gold as the thickness increases.

6. TIN PLATING

Electroplated bright/matte tin is a solderable finish that offers a flat surface and has a lustrous surface. Typical thickness is 0.0001" to 0.0003" for solderability and 0.0003" to 0.0005 for corrosion resistance. Tin plating is governed by MIL-T-10727B.

7. OTHER AVAILABLE FINISHES

Chemical Film - this is a chromate conversion coating that is applied to aluminum. Chemical films are governed by MIL-C-5541C. Chemical film coating provides protection to the exposed aluminum surfaces.

Immersion Tin - this is a thin coating applied by immersion. Thickness varies from 10 to 50 microinches maximum. This finish has good short term solderabilty, but has limited shelf life. Immersion tin plating is governed by MIL-T-81955.

White Tin - this is an immersion coating with superior solderability and long shelf Life and can be used as a replacement for Hot Air Leveling

Plating thickness is measured by either X-Ray fluorescence (non-destructive) or micro-section (destructive). X-Ray fluorescence measurements may be made anywhere on the board surface as long as the area being measured is twice the width of the beam. Micro-sections are best done using a test coupon that is removed from the panel, but is representative of the plating on the panel. If coupons are required it should be specified in the drawing notes and coupon artwork should be included in the CAD data.

Drawing notes should specify the type of plating finish and thickness. See examples below:

"Plated through holes to be plated with 0.001" minimum copper plating."

"Circuitry to be plated with Gold, Type III, 50 microinches minimum over low stress nickel, 50 microinches minimum."

Questions concerning how to properly specify plating finishes can be addressed by MCN.

8. SPECIAL PLATING PROCESSES

Holes can be copper plated to the required thickness with minimal surface plating. This process plates holes only to the proper thickness while adding a small amount of additional copper to the surface. This is especially useful with critical circuits, such as filters. Contact MCN for further information.

9. DEFINITIONS

Electroless plating - Plating without the use of a rectifier (power supply)

Electroplating - Plating that requires a rectifier, using the board as the cathode in the plating solution.

Fused surface - Tin/lead plating heated to the melting point and then allowed to solidify. This process results in longer term solderability than as-plated tin/lead, but also causes the tin/lead surface to be mounded at the center of the circuit trace.

Hot-Air Level - Molten solder is applied to the surface of the circuit and the excess is removed with a hot-air knife. This results in a flat surface, and a remaining solder coating of approximately100 microinches thick.

X-Ray Fluorescence - A non-destructive measurement method to determine plating thickness, using a small x-ray beam.

Test coupon - Added to one side of the artwork to be used for microsections. The coupon should contain the smallest diameter plated through holes.

VI. MACHINING

This section deals with all the mechanical operations necessary to fabricate the microwave circuit board. Included is drilling, laser cutting, machining and punching.

1. DRILLING

Drilling of microwave substrates requires different parameters than drilling epoxy based materials because the material is softer and more difficult to drill cleanly. Drilling is accomplished with high speed CNC machine. Maximum spindle speeds are 80,000 rpm. Controlled spindle speed and feed rate are critical to achieve clean holes, acceptable for plating.

Hole diameters are dictated by the overall substrate thickness. Consideration must be given to the aspect ratio of hole diameter to total substrate thickness. As a general rule, aspect ratios of 5:1 are standard, with increasing difficulty up to 7:1. For example, a 0.020" hole in a .060" thick substrate is 3:1. A 0.015" hole in the same substrate is 4:1 with increased complexity. As the drill size decreases, it becomes more difficult to get good solution flow through the holes for the plating processes. The minimum recommended hole size is .012".

Hole diameters can be specified before or after plating. Plating closes down the hole diameter by twice the plating thickness. The tolerance on the hole diameter after plating is limited by the combined tolerances of the drilling and plating processes.
Unless otherwise specified on the drawing, hole sizes are measured after plating.

Hole size after plating can be difficult to accurately predict due to variations in plating current distribution. Hole size, hole density, and sizes and shapes of adjacent circuits and ground planes will dictate metalization thickness in a particular hole due to variability in the local current density. Often, specifying hole sizes before plating and specifying a minimum (but not maximum) plating thickness will improve manufacturability. This is most appropriate where plated through holes function as mode suppression grounding or inner layer interconnection vias. An overall bilateral tolerance should be used on holes designated for leaded components.

The ability to hold positional tolerances varies significantly depending on material type (inherent dimensional stability), thickness, and overall part dimensions. A true position diameter of .010" is most common and readily attained. Manufacturability is adversely affected by tolerances tighter than .006"

2. ROUTING AND/OR CNC MACHINING

Tolerances on feature location are the same as for through hole location. Tolerances on feature size are dictated by cutter tolerances and positional tolerances, and are typically +/-.005" for CNC routing machines, and +/- .003" for CNC milling machines. Tighter tolerances may be obtained at increased cost.

Depth tolerances are dictated by the machine tolerance and, more important, material thickness tolerance. Typical depth tolerances are +/-.002". Tighter tolerances of +/-.001" is attainable at an increased cost.

Since cylindrical tools produce all pockets and slots, inside corners must have a specified radius. Larger radii permit the use of larger cutting tools and higher feed rates that can reduce manufacturing costs. A .047" radius is preferred, and a minimum of .031" is recommended. One of the methods shown below is recommended for square corners.

Hardened steel punch and die sets can maintain tolerances of +/-'002" and are recommended for high volume unsupported boards.

3. LASER CUTTING

Laser cutting of soft substrate materials is a standard procedure, capable of maintaining +/-.001” tolerances. Most of the materials used in the microwave industry can be laser cut.
In programmed routing, the part periphery is programmed into the laser similar to normal CNC operations. The laser beam must be able to access bare dielectric, since any copper or plating in the beam path will reflect the laser beam and damage the substrate material. It is recommended that circuitry maintain .002” to .005” pull back from all sides of the board. Ground plane (if any) may be programmed to final size.

VII. MULTILAYERS

There are several types of multilayers: Conventional construction with plated through holes, conventional construction with access to internal layers from the top surface, composite construction, and special processes using different interconnect methods.

Multilayers using microwave dielectric materials differ from conventional epoxy multilayers in several ways.

A. The substrate material is dimensionally unstable. The thinner the substrate, the more it will change during processing. Shrinkage occurs as the copper is removed during the etching process. There are processes available to control the dimensional changes but cannot eliminate it.

B. Bonding of microwave multilayers is typically done using special dielectric bonding films that have matching properties to the substrate material. Both thermoplastic and thermoset films are available. Typically these films are non-supported films, i.e., they have no glass fiber in them. Therefore, copper layer to copper layer bonding is not recommended as this would result in potential shorts between layers. A circuit layer is generally opposite an unclad layer.

C. Multilayers based on PTFE materials cannot use an etchback process such as used with epoxy based materials. The plated through holes can be desmeared using a plasma etch. Also, plated through holes must be processed either by sodium etching or plasma to make the hole walls wettable to accept the electroless copper plating. Sodium or plasma etching is also necessary prior to bonding to obtain a bondable surface for increased bond integrity of the finished board.

Modular Components National, Inc.

Design Guide

Bonding of multilayer boards is done in a lamination press under heat and pressure. The press also is enclosed in a vacuum chamber so that all air is removed prior to activating the press cycle. This allows closer control of the bonding process and also allows use of lower pressures to minimize any mechanical distortion to the boards.

There are several bonding films available, both thermoplastic and thermoset, with different melt point temperatures. These temperatures range from 570 F to 250 F.
Dielectric constant of these films range from 2.15 to 4.0

1. CONVENTIONAL CONSTRUCTION

Conventional construction consists of three or more layers bonded together in a single bonding process. The internal layers may have plated through holes (buried vias) connecting a layer from side to side, or all the interconnects will be made after the board is bonded using plated through holes from the external layers.

A variation of this process is the use of sequential bonding where a series of layers are bonded together and then this bonded unit is bonded again to another layer or layers. The use of sequential bonding allows more freedom in the design, but requires the use of bonding films that have different bonding temperatures.

2. CONVENTIONAL CONSTRUCTION WITH ACCESS TO INNERLAYERS

This method differs slightly from conventional construction in that access holes are cut into the external layers to allow access to one or more internal layers. Plated through holes and edge plating can still be used with this method. Access to innerlayers can be made from both sides of the PWB. Where necessary for shielding, plugs can be inserted into the access holes after the component mounting in those areas is complete. Adhesive flow into access cavities during bonding is difficult to predict. Therefore, some allowance for adhesive flow must be made. Typically, a bead of .010" to .020" is allowed.

3. COMPOSITE CONSTRUCTION

Composite construction consists of bonding a layer or layers of microwave substrate material to an epoxy or polyimide based material. This method is used primarily when a PWB contains both RF and DC circuitry that needs to be interconnected. The epoxy or polyimide PWB can also be a multilayer. This method of fabrication results in a very compact PWB that will replace to individual PWB's. Again, plated through holes are used to make the interconnections. Access holes can also be used for this method. Typically the bonding material used for composite PWB's is standard epoxy or polyimide prepreg. Composite PWB's require desmearing of the holes prior to plating.

4. SPECIAL MULTILAYER PROCESSES

MCN has special processes available that will allow any combination of blind or buried via interconnections. These processes are considered Company Proprietary, but can be discussed after negotiating a signed non-disclosure agreement. Up to 18 layers PWB's have been fabricated using this technology. Applications range from very critical filters to power divider/combiners. It is important to manage the hole diameter aspect ratio within the limits discussed earlier in the drilling section. The hole diameter needs to be sufficient to allow proper plating and assure internal connections between layers. Please contact MCN for further information about multilayer applications.

VIII. CONDUCTIVE BONDING

To enable our customers to obtain a more complete assembly, MCN offers conductive bonding of microwave substrates to carriers or housings. The carriers or housings can be provided by the customer or MCN’s in-house CNC machine shop. Conductive bonding is generally less expensive than using a prebonded material.

Aluminum, copper, and brass are the most common materials used for carriers or housings.

Two methods are available for conductive bonding: soldering or conductive epoxy.
Both methods have been proven in use up to and including 38GHz. The soldering method offers lower cost, with some limitations. Conductive epoxy is more expensive, but overcomes the limitations experienced with soldering.

Conductive bonding allows access to the backside circuitry of the PWB through the backside of the carrier. Bonding into housings that are 1/2" to 1" deep is feasible.

Both conductive bonding methods require heat and pressure. This requires a press with heated platens and hydraulic pressure control. The finished bond is uniform and void free in the bond area.

1. SOLDERING

Soldering requires the carrier or housing to have a solderable finish such as nickel or iridite.

Typically the PWB backside is plated with tin lead alloy (60/40) to a minimum thickness of 350 microinches to solder to the carrier or housing. If a higher temperature application is desired, an alloy such as 96.5 Tin (Sn) or 3.5 Silver (Ag) is applied. This is done using a thin foil applied between the PWB and the carrier or housing.

Usually the PWB is soldered to the carrier in panel form prior to final machining or processing. The soldering process is done on individual PWB's, bonding, requiring the using fixtures to hold and align the PWB during the soldering process.

It is necessary to allow for volatiles from the flux to escape during the bonding process. Generally there are sufficient holes in the PWB to allow this. However, in some cases, it may be necessary to add holes. Customers are contacted to obtain concurrence for adding additional holes.

As it is difficult to control solder flow, it is recommended that machining of pockets or other features on the surface of the PWB be done after the solder bonding has been completed. There should be allowance in housing for solder flow between the edges of the PWB and the sidewalls of the housing. Contact MCN for specific recommendations on tolerances and allowances for a particular design.

2. <u>CONDUCTIVE EPOXY</u>

MCN uses a silver loaded conductive epoxy for bonding. This material is purchased as an unsupported film, .002' or .003" thick. It is more expensive than soldering, but the flow characteristics can be controlled.

Conductive epoxy is best bonded to a plated surface rather than bare metal. It is not compatible with bare aluminum. It is recommended that the carrier or housing be plated with nickel, gold, silver, or tin.

The bonding procedure is similar to the soldering process, requiring heat, pressure and similar fixturing and equipment.

Conductive epoxy is a thermoset material. Once the cure cycle is complete, temperatures above the cure temperature are not a problem, contrary to the soldering process.

Contact MCN for specific questions on tolerances and allowances.

IX. SUMMARY

1. **Substrate Materials** – These are usually determined and requested by our customers on their RFQ, however, we can help determine the best vendor based on price and delivery. The most important is information regarding the dielectric constant, loss tangent and size of material that is needed. By specifying the type of material (GR, non-woven, GT, GX or GY woven,) the DK, copper thickness and base material thickness, we will quote all of the suppliers that produce that type material. Also, it is important to know if the material is pre-bonded to a metal base material or 2 sided material that is to be bonded to metal after processing.

2. **Imaging** – Artworks are produced with data supplied by the customers by disk, modem (410 879-6215), e-mail (cam@mcn-mmpc.com), FTP (ftp.mcn-mmpc.com, Login – anonymous, Password – Customer's e-mail).

   Minimum annular ring required is .002" on the finished product.

   Minimum front to back registration is .003".

3. **Panel sizes** - These are determined by MCN based on manufacturability and material availability.

4. **Etching -** Tolerances are determined by the thickness of the copper at the time of etching. Basically, +/- .0005" on ½ ounce copper, +/- .001" on 1 ounce copper. Minimum lines and gaps are .003".

5. **Plating** – We have in house plating tanks for Electroless Copper, Copper, Tin/Lead, Nickel, Gold and Electroless Tin (Including White Tin).

6. **Drilling -** We prefer an aspect ratio for plated through holes to be 5:1, however, we will entertain 7:1 with increased costs to process.

   Minimum drilled hole size is .010"

7. **Machining –** Typical tolerances for the CNC milling machines are +/- .003". Tighter tolerances may be obtained at a premium. Typical depth tolerances are +/- .002". Radius minimum .031" with .047" preferred.

   Laser cutting will maintain a +/- .002" tolerance on all features.

8. **Multilayers –** We produce multilayers with PTFE on all layers with blind and buried vias up to 14 layers. We also produce mixed composite boards (PTFE and FR4) with the RF circuits on the PTFE and the digital on the FR4 layer.

9. **Conductive Bonding –** We produce bonded product (Circuit to Metal base) with 2 different methods, soldering using tin/lead plating and conductive epoxy.