AEROSPACE RECOMMENDED PRACTICE

ARP1612™	REV. A

Issued 1979-10
Revised 1990-01
Reaffirmed 2022-06

Superseding ARP1612

Polyimide Printed Circuit Boards
Fabrication of

RATIONALE

ARP1612A has been reaffirmed to comply with the SAE Five-Year Review policy.

1. SCOPE:

1.1 This document describes the materials, equipment, and processing techniques utilized in the fabrication of polyimide printed wiring boards. Included are recommendations for both double-sided and multilayer boards.

1.2 The processes described herein are the result of extensive evaluation and manufacturing experience. These recommendations reflect procedures that have proven effective in producing low-cost and reliable printed wiring boards.

1.3 Safety - Hazardous Materials: While the materials, methods, applications, and processes described or referenced in this document may involve the use of hazardous materials, this document does not address the hazards that may be involved in such use. It is the sole responsibility of the user to ensure familiarity with the safe and proper use of any hazardous materials and to take necessary precautionary measures to ensure the health and safety of all personnel involved (see 9).

2. APPLICABLE DOCUMENTS:

The following publications form a part of this document to the extent specified herein. The latest issue of Aerospace Material Specifications shall apply. The issue of other documents shall be as specified in AMS 2350.

2.1 SAE Publications: Available from SAE, 400 Commonwealth Drive, Warrendale, PA 15096.

2.1.1 Aerospace Material Specifications:

AMS 2350 Standards and Test Methods

TO PLACE A DOCUMENT ORDER: **Tel: 877-606-7323 (inside USA and Canada)**
Tel: +1 724-776-4970 (outside USA)
Fax: 724-776-0790
Email: CustomerService@sae.org
SAE WEB ADDRESS: http://www.sae.org

For more information on this standard, visit https://www.sae.org/standards/content/ARP1612A/

2.2 U.S. Government Publications: Available from Commanding Officer, Naval Publications and Forms Center, 5801 Tabor Avenue, Philadelphia, PA 19120.

2.2.1 Federal Specifications:

A-A-113 Tape, Pressure-Sensitive, Adhesive

2.2.2 Federal Standards:

FED-STD-406 Plastic, Methods of Testing

2.2.3 Military Specifications:

MIL-P-13949 Plastic Sheet, Laminated, Metal-Clad (For Printed Wiring),
MIL-P-13949/10 Plastic Sheet, Laminated, Metal-Clad (For Printed Wiring Boards), Base Material, Gi Glass Base, Woven, Polyimide Resin, Heat Resistant, Copper Clad
MIL-B-13949/13 Plastic Sheet, Laminated, Materials (For Printed Wire Boards), Gi Base Material, Glass Fabric Woven, Polyimide, Resin Preimpregnated (B-Stage)
MIL-F-14256 Flux, Soldering, Liquid (Rosin Base)
MIL-B-22191 Barrier Material, Transparent, Flexible, Heat Sealable

2.2.4 Military Standards:

MIL-STD-202 Test Methods for Electronic and Electrical Component Parts
MIL-STD-275 Printed Wiring for Electronic Equipment

3. GENERAL:

3.1 Polyimide-glass laminates are classified as high-performance materials and offer significant advantages over conventional laminate materials such as epoxy-glass.

3.2 The excellent properties of the polyimide material result from an addition-type thermosetting reaction. Actually, two chemical reactions take place. One is a Micheal-type addition between a low molecular weight diamine complex and bis-maleimide type complex at approximately 175°C (375°F). The second is a cross-linking of bis-maleimide complexes at approximately 230°C (450°F).

3.3 Polyimide laminates provide solutions to some basic problems that exist with conventional materials. In printed wiring board (PWB) applications the advantages of polyimide are as follows:

3.3.1 Higher glass transition temperature

3.3.2 Lower axis expansion coefficients

3.3.3 Decreased resin smearing during drilling

3.3.4 Greater delamination resistance and decreased pad lifting during soldering operations

3.3.5 Superior resistance to thermal stress damage in thermal shock and temperature cycling

3.3.6 Greater repairability

3.3.7 Lower life cycle costs of electronic assemblies

3.4 Polyimide double-sided boards may be processed in an identical manner to conventional epoxy-glass units; however, there are some differences for multilayer boards. Copper-clad polyimide laminates are treated prior to lamination to obtain a cupric oxide finish as opposed to the cuprous oxide utilized for conventional laminates. Slightly higher laminating temperatures of 175°C (350°F) and postcuring at 230°C (450°F) are required for polyimide. Polyimide boards do not require chemical smear removal treatments after drilling. Such treatments are commonly used for epoxy boards.

4. PROCESS RECOMMENDATIONS:

4.1 Design Considerations: No special design precautions are necessary when utilizing polyimide boards. The requirements established by MIL-STD-275 for board design may be utilized for any type of printed wiring board including polyimide.

4.2 Material Considerations: Copper-clad laminates for double-sided boards and
Ø multilayer boards should be procured to the requirements of MIL-P-13949/10 or as specified on the drawing. B-stage material should be procured to MIL-P-13949/13. Other materials utilized include commercial grade backer material, peanut oil, rosin flux (Type RA or Type RMA, MIL-F-14256), copper plating solution, tin-lead plating solution, resist stripper, cheesecloth, pumice cleaner, touch-up ink (plating resist), etch resist ink, electroless copper plating solutions, acid and alkaline cleaning solutions (for example, 1% or 25% sulfuric acid, ammonium persulfate solution), 1, 1, 1 trichloroethane (technical grade), dry film resist, and dry film developer, mold release, kraft paper, aluminum oxide paper or slurry, and heat-sealable plastic envelopes.

4.3 Equipment Considerations: No special equipment is required to process polyimide boards. Conventional equipment may be utilized and includes: dry film laminator, ultraviolet (UV) exposure unit, dry film developer, vapor degreaser, forced-air oven capable of maintaining a temperature of 230°C ± 5 (450°F ± 10), copper anodes, oxygen free, high conductivity (OFHC), tin-lead anodes (63/37% tin-lead nominal), plating tanks with racks and clamps as required, deionized water system, resist stripper, etching machine, board scrubber, end mill cutters, carbide drills and router bits, numerically controlled (NC) drill, fusing apparatus, silk screens, laminating press, liquid honing machine, various tanks for cleaning/rinsing operations, and solder bath.

4.4 Processing Recommendations: The following practices should be followed to ensure the quality of the printed wiring board:

4.4.1 Drilled holes shall be of the proper count and at the location specified by the part drawing. Holes shall be free of burrs and other anomalies.

4.4.2 The machine shop and drilling areas shall be clean and orderly.

4.4.3 Numerically controlled drills shall be checked at periodic intervals for accuracy. Drill bits shall be monitored for effective life and discarded or resharpened at specified intervals.

4.4.4 Master patterns shall be to the accuracy specified by the part drawing or other control documentation.

4.4.5 Registration shall be verified for double-sided boards by film overlay techniques or dimensional verification. X-ray techniques may be utilized to verify the registry of multilayer boards.

4.4.6 At regular intervals, a section of a sample board or test tab shall be solder floated and metallurgically examined for plating integrity, hole wall cracking, voids, or other signs of poor quality.

4.4.7 The application, exposure, and development of photoresist shall be performed in an area equipped with safelights.

4.4.8 All plating power supplies, meters, gauges, heaters, etc, shall be calibrated and maintained at regular intervals.

4.4.9 All plating anodes shall be examined periodically and replaced when depleted.

4.4.10 Boards shall be thoroughly rinsed after each processing step to ensure complete removal of processing solutions and to prevent contamination of subsequent solutions.

4.4.11 Ø Tape tests using tape conforming to A-A-113 shall be performed to verify plating adhesion.

4.4.12 All processing solutions shall be prepared and controlled within the limits specified by their suppliers.

4.4.13 All plating solutions should be monitored using appropriate methods (atomic absorption spectroscopy, wet chemistry) for their essential elements and maintained within manufacturer-recommended limits.

4.4.14 "B" stage material used for multilayer boards should be stored under controlled environmental conditions. When stored at 20°C (70°F) and 50% relative humidity, "B" stage materials may be safely used within 3 months of their receipt date.

4.4.15 Handle boards either with tongs or on the edges with clean, white, lint-free gloves, taking care not to touch or contaminate surfaces to be laminated, electroplated, or soldered.

5. MULTILAYER PRINTED WIRING BOARD FABRICATION CYCLE:

Fig. 1 illustrates the typical process steps involved in the fabrication of polyimide multilayer boards. The following is an elaboration of these general procedures:

5.1 Tooling Requirements:

5.1.1 Master Pattern Film: The film material shall be 0.0075 in ± 0.0005 (0.190 mm ± 0.013) thick, dimensionally stable polyester film as described in MIL-STD-275. The accuracy of the master pattern shall be such that the centers of all features (terminal areas, conductors, etc) shall be located within 0.002 in (0.05 mm) radius of the true grid position established for the layer. For the composite master pattern, the features of all layers shall coincide within 0.003 in (0.08 mm) radius of the true grid position, when measured at 20°C ± 1 (68°F ± 2) and 50% ± 5 relative humidity.

5.1.2 Drill Tapes: Punched tapes suitable for use with numerically controlled drill tapes may be generated to grid coordinate locations or to the centers of master pattern terminal areas.

5.1.3 Profile Templates/Tapes: Boards may be sized to the proper peripheral configuration by a variety of methods. A routing template may be fabricated from 0.25 in (6.5 mm) epoxy glass laminate or aluminum. Punched tapes may be prepared for numerically controlled routing machines or, if volume dictates, blanking dies may be utilized.

5.2 Innerlayer Preparation:

5.2.1 Prepare Index Holes: The proper type, size, and number of copper-clad innerlayers are selected and then punched or drilled to form indexing or tooling holes for use in subsequent operations, imaging, lamination, etc.

5.2.2 Cleaning: Copper-clad thin laminate innerlayers should be cleaned by degreasing in trichloroethane or other suitable solvent followed by a combination overflow rinse and deionized water. A 30 s immersion in 25% by volume sulfuric acid followed by a combination rinse is used. Scrubbing (machine or pumice) and rinsing to obtain a no-water-break surface are required prior to resist application.

5.2.3 Resist Application: Two commonly used approaches are screening and dry film. Because of its photographic precision, dry film is preferred for multilayer fabrication and is described herein.

5.2.3.1 Dry Film Lamination: Cleaned innerlayers are dried for approximately 5 min at 95°C (200°F). Dry film resists of various thicknesses are available and are used in roll laminators. Processing controls vary with manufacturers but key features are laminating temperature, traverse speed, and roll pressure. Using manufacturer-recommended settings, copper-clad innerlayers are laminated with dry film resist.

5.2.3.2 Dry Film Exposure: Dry films are sensitive to UV exposure. An image is imparted to the resist-coated innerlayer using the master pattern film and a UV exposure source. Negative films are utilized which allow hardening of the resist in the clear areas of the film. Typical controls are UV output and exposure time. Useful monitoring devices include UV output meters and exposure step scales.

5.2.3.3 Dry Film Development: The protective cover sheet is removed from the exposed dry-film-coated innerlayers which are then placed in conveyorized developer machines. Dry-film developing solutions may be either of a solvent or aqueous type. In either instance, the developer removes the unexposed areas of dry film (dark areas of master pattern film) leaving the copper-clad innerlayer with exposed resist defining all circuit features. Controls in this operation are solution concentration, temperature, life, and development time.

5.2.4 Touch-up: After development, resist-coated innerlayers are visually examined for anomalies, pin holes, scratches, and breaks, which may be touched up with silkscreen inks formulated to be etch resistant. If the degree of touch-up is deemed excessive, the innerlayers may be stripped of resist and reprocessed through the dry film application or scrapped.

5.2.5 Etching: A variety of alkaline and acid etchants are used in industry. The purpose of the etching step is to remove unwanted copper from the copper-clad innerlayer. In the process, the innerlayer is fed into a conveyorized etching machine wherein etching solution is sprayed on both top and bottom surfaces. The dry-film resist protects the copper conductor areas while the unprotected areas are removed, thereby defining the circuit image. Most etching machines incorporate internal rinsing and air drying steps after etching. Controls include etchant solution concentration, life, temperature, and conveyor speed.

5.2.6 Resist Removal: Depending on the type of resist selected, a solvent or aqueous solution is used to strip the resist from etched innerlayers. This is normally a conveyorized machine operation wherein innerlayers are stripped, rinsed, and air dried. Controls include conveyor speed, stripper solution concentration, life, and temperature. Innerlayers should be carefully inspected at this point for defects such as broken conductors, over or under etching, and excess copper. Defects missed at this stage will show up at the completed board or assembly level as opens or shorts and may result in expensive scrap.

5.2.7 Oxide Treatment: The etched copper-clad innerlayers are oxide treated to promote bonding during subsequent lamination. The oxide treatment must not be degraded by the 230°C (450°F) postcure treatment given polyimide laminated boards. The following process has been used successfully:

5.2.7.1 The innerlayers are cleaned in a commercial alkaline cleaning solution for 3 to 5 min followed by a combination overflow and deionized water rinse. The innerlayers are immersed in ammonium persulfate for 30 s followed by a combination rinse and successive immersion in a 25% by volume sulfuric acid solution and combination rinse.

5.2.7.2 After preparatory cleaning, the oxide treatment is imparted by immersing the innerlayers in a 90°C ± 3 (195°F ± 5) cupric oxide forming solution. The makeup of the solution is as follows:

Sodium Chlorite ($NaClO_2$):	30 g/L
Sodium Hydroxide (NaOH):	10 g/L
Trisodium Phosphate ($Na_3 PO_4$):	5 g/L
Deionized Water	remainder

The duration of treatment is approximately 5 min at which time the copper turns uniformly black. After oxide treatment, the innerlayers are combination rinsed, dried in air for not less than 30 min followed by baking at 120°C ± 3 (250°F ± 5) for not less than 1 h. Essential controls are solution composition, temperature, and treatment time.

5.3 Lamination:

5.3.1 Lay-up:

5.3.1.1 In this process, the oxide treated innerlayer laminates are combined with "B" stage materials to form the multilayer composite. The partially cured "B" stage material bonds the composite at the proper pressure and temperature and provides the dielectric spacing between layers. The selection of various plies of "B" stage also controls the overall thickness of the multilayer board. The required number of "B" stage sheets are sheared to size and punched to match tooling holes contained in the innerlayer laminates.

5.3.1.2 A thin coat of mold release is applied to the inside surface of the laminating plates (typically aluminum or steel), one of which contains pins matching the tooling holes of the innerlayer and "B" stage materials. The bottom innerlayer is placed over the tooling pins of the laminating plate. This is followed by the desired number of "B" stage sheets. This stacking sequence is followed until all layers are stacked on the laminating plate. The top laminating plate is then dropped over the tooling pins. Laminating plastic and kraft paper are placed next to each laminating plate to facilitate release and to provide temperature lagging.

5.3.2 Pressing Conditions: Typically, polyimide lamination is achieved using a hydraulic press with electrically heated platens. The press platens are allowed to stabilize at 65°C ± 3 (150°F ± 5). The lay-up is placed in the press and 10 psi (68 kPa) pressure applied with increasing temperature until the stack reaches 130° to 140°C (270° to 280°F). At this temperature, the pressure is increased to 600 psi (4100 kPa) and the temperature is allowed to reach 175°C ± 3 (350°F ± 5). The 600 psi (4100 kPa) pressure is maintained for 90 min after the temperature of 175°C (350°F) is reached. After this time, the press is cooled until the plated temperature is less than 40°C (100°F). The pressure is released and the laminated board removed. The laminated plastic and kraft paper are discarded and any residue cleaned from the platens. An effective control of this process is the use of a thermocouple to monitor the actual temperature reached by the center of the lay-up stack coupled with the time required to achieve the desired temperature.

5.3.2.1 An alternate cure cycle using a hot steam heated press at 182°C ± 3 (360°F ± 5) and allowing the inside temperature to reach 160°C ± 3 (320°F ± 5) before applying pressure of 600 psi (4100 kPa) with a 2 min rise time has been found to be equivalent to the procedure in 5.3.2.

5.4 Postcure: After lamination, the boards require postcuring at 230°C (450°F) for 8 h to develop the full properties of the polyimide material. This should be accomplished in a forced-air oven with the boards laying flat. After postcure, the resin flash should be removed from all sides of the board followed by immersion in a 1% by volume sulfuric acid solution. The board should be manually or machine scrubbed, rinsed, and dried.

5.5 Numerically Controlled Drilling:

5.5.1 Set Up: Postcured boards are pinned to unclad backer material such as hard board or aluminum coated press board. Epoxy/glass backing material should be avoided because of its tendency to smear during drilling. Carbide drill bits of the proper size are selected and preferably should be new because used or resharpened bits may cause drilling problems (smears, rough holes, etc). The proper speeds and feeds are selected for each bit size and the upper and lower stroke elevation settings are established.

5.5.2 Drill Operation: Using the drill tape prepared for the specific board design, all holes are drilled as required. Drill bit life should be monitored, and 1200 hits (holes) per drill are an effective maximum limit. It is recommended that the first board drilled of each production lot be subjected to X-ray for registration verification. If the hole location is acceptable, the remainder of the lot may be drilled. If not, adjustments may be made to the floating zero to move the hole pattern to align it to the internal pads.

5.5.3 Deburring: After drilling, it is necessary to remove drill burrs which might interfere with subsequent operations. This may be accomplished manually by sanding with 180 grit (80 µm) or finer aluminum oxide paper or in an appropriate machine using brushes coated with an abrasive material such as aluminum oxide.

5.6 Liquid Honing: Pressure blasting with an abrasive slurry of 240 mesh (60 µm) aluminum oxide grit and water has proven effective in removing any drill debris from the interior of the drilled holes. This practice is recommended to remove a chaff which develops during drilling and which may interfere with later plating operations. Rinsing is required to remove all traces of aluminum oxide grit. Controls in this operation are typically pressure, grit concentration, and conveyor speed.

5.7 Electroless Copper Plating:

5.7.1 The purpose of electroless copper plating is to provide an initial deposition of copper on the insulative polyimide glass, hole walls. The initial step involves manual or machine scrubbing of the drilled boards followed by racking of the board for electroless plating.

5.7.2 The various process steps are as follows:

5.7.2.1 Immerse boards in alkaline cleaner (various types available) for 10 min.

5.7.2.2 Rinse in overflow water for 2 min plus deionized water for 30 s.

5.7.2.3 Immerse in an acid conditioner (various types available) for 5 min.

5.7.2.4 Rinse in overflow plus deionized water.

5.7.2.5 Immerse for 1 to 1.5 min in ammonium persulfate solution of the following composition:

Ammonium persulfate ($[NH_4]_2S_2O_8$):	1.5 - 2.0 lb (680 - 907 g)
Sulfuric acid (H_2SO_4), technical grade:	2.0 fl oz (60 mL)
Water, to make	1 gal (3.8 L)

5.7.2.6 Rinse in overflow water (30 s), then deionized water (30 s).

5.7.2.7 Immerse in 25% by volume sulfuric acid solution for 1 to 1.5 min.

5.7.2.8 Rinse in overflow water plus deionized water.

5.7.2.9 Immerse in 25% by volume hydrochloric acid solution for 5 min.

5.7.2.10 Without an intermedite rinse, directly immerse in a catalyst solution (various types available) for 5 min. This solution deposits colloids of a tin covered noble metal (palladium) on the interior hole surfaces.

5.7.2.11 Rinse in deionized water, overflow water, and deionized water spray.

5.7.2.12 Immerse in an accelerator solution (various types available) for 10 min. The purpose of the accelerator is to strip the tin from the deposited colloids, exposing the noble metal catalyst.

5.7.2.13 Rinse in water overflow plus deionized water spray.

5.7.2.14 Immerse in electroless copper solution (various types available) for 20 min. In this step, copper is deposited at noble metal sites and forms autocatalytically to continuous sheet thickness.

5.7.2.15 Rinse in overflow water plus deionized water spray.

5.7.2.16 Boards should be transferred to the next copper deposition as soon as possible. They may be held in a 1% by volume sulfuric acid solution for approximately 2 h if delay is required.

5.7.2.1.6.1 Subsequent copper plating as specified in 5.7.2.16, 5.8.1 and 5.8.2 may not be necessary if a heavy electroless copper plate is used.

5.7.3 Controls for the above operation include the processing of a predetermined area of board surface followed by dumping and replenishing of the various cleaning solutions. Color standards and replenishments may be used for electroless copper solutions.

5.8 Copper Pre-plate:

5.8.1 After the electroless copper deposition, electroplated copper is deposited in the holes to a thickness of usually 0.0003 to 0.0005 in (8 to 13 µm). A number of commercially available copper plating solutions may be used. Parameters for a commonly used pyrophosphate solution are as follows:

Copper: 3.0 oz/gal (22.5 g/L)
Pyrophosphate/Copper ratio: 7.7 : 1
Orthophosphate: 12 oz/gal (89.9 g/L)max
Ammonia as NH_3: 0.2 oz/gal (1.5 g/L)
pH: 8.35
Operating Temperature: 50°C ± 3 (125°F ± 5)
Agitation: Air and mechanical
Filtration: Continuous

5.8.2 Process steps are as follows:

5.8.2.1 Rack boards on electroplating racks (variety available).

5.8.2.2 Rinse in deionized water (30 s).

5.8.2.3 Clean in 25% by volume sulfuric acid for 30 s.

5.8.2.4 Rinse in overflow water plus spray deionized water.

5.8.2.5 Immerse boards in copper plating bath with current set at low level (1 to 2 A) and tighten rack clamps securely to the cathode bar.

5.8.2.6 Increase the plating current slowly to the recommended 35 A/ft^2 (376 A/m^2) current density, as required for the area to be plated.

5.8.2.7 After plating for the time required to achieve the desired plating thickness, the current is reduced and the racks are removed from the bath, rinsed in overflow water, and allowed to dry.

5.8.3 Controls for this operation include chemical analyses to ensure the bath is of the proper composition. When organic addition agents are added, they are controlled by a variety of means (amp-hour, plating area, etc) and are removed periodically by carbon filtration. Test tab samples may be examined metallurgically for adequacy of copper thickness.

5.9 Image Transfer:

5.9.1 Cleaning: Copper preplated boards are cleaned following the process outlined in 5.2.2.

5.9.2 Resist Application: Resist is applied as described in 5.2.3, 5.2.3.1, 5.2.3.2, and 5.2.3.3. One exception to the procedure outlined for innerlayers is that preplated boards are imaged using positive rather than negative films. In this instance, positive films acts as a plating-resist allowing plating deposition in the defined circuit areas.

5.10 Touch-Up: Boards are touched up as described in 5.2.4 except that a commercially available silkscreen plating ink is used to repair anomalies.

5.11 Copper-Final Plate: With the resist coated boards outlining open circuitry areas, copper is deposited to a thickness of not less than 0.001 in (25 µm) in the plated-through-holes. The general description and controls described in 5.8 apply in this step also. Some specific sequential operations are described below:

5.11.1 Rack boards on plating racks and immerse in an acid-type (various types available) cleaner.

5.11.2 Rinse in overflow water plus deionized spray water rinse.

5.11.3 Clean in ammonium persulfate solution for 30 s.

5.11.4 Rinse in overflow water plus deionized water spray.

5.11.5 Clean in 25% by volume sulfuric acid solution for 30 s.

5.11.6 Rinse in overflow water plus deionized water spray.

5.11.7 Place racked boards in copper plating tank with current on and set at 1 to 2 A.

5.11.8 Gradually increase the current to the required amperage to achieve a current density of 35 A/ft^2 (376 A/m^2) and plate for a time sufficient to achieve the desired copper thickness.

5.11.9 Remove racks from the tank and rinse in overflow water plus deionized water spray.

5.11.10 Remove copper plated panels from racks and place in 1% by volume sulfuric acid solution.

5.12 Tin-Lead Plating:

5.12.1 Tin-lead is plated to provide a metallic etch resist and to facilitate soldering attachment of components during assembly. Tin and lead are codeposited from a variety of commercial plating solutions, normally to a ratio of 63% tin to 37% lead.

5.12.2 The following is a brief description of a typical bath:

Stannous tin:	2.0 oz/gal (15.0 g/L)
Lead:	1.34 oz/gal (10.0 g/L)
Fluroboric acid:	46.73 oz/gal (350 g/L)
Boric acid:	3 oz/gal (22.5 g/L)
Stabilized peptone:	2.18 fl oz/gal (17.0 mL/L)
Temperature:	Room
Agitation:	Mechanical

5.12.3 Boric acid is typically placed in a dynel bag and suspended in the plating solution to maintain saturation. Peptone, which is added for grain refinement, is replenished on a plating area, amp hour, or other use basis. At some regular interval all peptone is removed by carbon filtration and replaced, for example, for a new bath. Periodically the tin-lead deposit is analyzed for tin-to-lead ratio. Ideally, this ratio should correspond to the eutectic composition (63 to 37) of bulk solder; however, a ± 10% variation is generally acceptable. The stannous tin, lead, and acid content of the bath are analyzed chemically on a periodic basis and additions are made to keep the bath in balance. Test tabs may be metallurgically analyzed for proper plating thickness of both the tin-lead and copper plating. A plating thickness of 0.0003 in. (8 µm) is required for tin-lead.

5.12.4 The following is a brief sequential process description of tin-lead plating.

5.12.4.1 Rack boards on plating racks.

5.12.4.2 Rinse in overflow water plus deionized water spray.

5.12.4.3 Clean in a commercially available acid cleaner.

5.12.4.4 Rinse in overflow water plus deionized water spray.

5.12.4.5 Immerse in ammonium persulfate solution for 30 s.

5.12.4.6 Rinse in overflow water plus deionized water spray.

5.12.4.7 Clean in 25% by volume sulfuric acid solution for 15 s.

5.12.4.8 Rinse in overflow water plus deionized water spray.

5.12.4.9 Immerse in 3% fluoboric acid solution.

5.12.4.10 Place racked boards in tin-lead plating tank with current set to a low level (1 to 2 amp).

5.12.4.11 Plate at desired current density, normally 25 amp per sq ft (272 A/m^2), to a thickness of 0.0003 in. (8 µm).

5.12.4.12 Remove boards from plating tank and rinse in overflow water and deionized water spray.

5.13 Resist Removal: Depending on the type of plating resist used, resist may be removed by either an aqueous- or solvent-type stripper. This typically takes place in a conveyorized machine which incorporates scrubbing, rinsing, and drying steps. It is particularly important to remove all traces of resist along tin-lead plated circuits. Any resist remaining will interfere with etching and will result in superfluous copper or irregularly etched circuit areas.

5.14 Touch-up: Tin-lead plated circuits are visually examined for scratches, pin holes, and other defects. Any such anomalies are touched up with screen ink as described in 5.2.4.

5.15 Etching: In this operation the tin-lead plating protects its underlying copper while all other copper (preplate and foil) is removed, thus leaving only the circuitry areas. Alkaline- and acid-type etching solutions perform as described in 5.2.5.

5.16 Cleaning: All touch-up ink is removed from the board surfaces by manual cleaning with solvent or by running through the resist stripping machine. After cleaning, the boards may be manually scrubbed with a commercially available scrub cleaner followed by thorough water rinsing and drying at approximately 120°C (250°F) for not more than 15 minutes.

5.17 Reflow Tin-Lead: Reflow is the process whereby codeposited tin and lead plating is subjected to a temperature where melting or fusing occurs. Fusing results in a solder alloy surface which promotes greater corrosion protection during storage and improves solderability. Fusing may be accomplished by either of two methods. One involves the use of a conveyorized infrared fusing machine which incorporates fluxing, preheating, and fusing steps. Normally the boards are spray water rinsed to remove acid flux residues which can cause corrosion during storage. In the other approach, boards are fluxed (MIL-F-14256, Type RA rosin flux) and immersed in a hot fusing fluid (peanut oil) at approximately 220°C (425°F) until fusion occurs. After fusing, these boards are vapor degreased using trichloroethane to remove traces of flux and fusing fluid.

5.18 Final Machining: After tin-lead reflow, the boards are sized to the configuration required by the part drawing. Boards are normally sized by routing but may be sized by blanking with special care and tooling.

5.19 Preparation for Delivery: Following sizing, the boards are water rinsed and dried, then marked as specified by the part drawing. Boards may be packaged in heat sealed plastic envelopes conforming to MIL-B-22191.

6. DOUBLE-SIDED PRINTED WIRING BOARD FABRICATION CYCLE: Fig. 2 illustrates the typical process steps involved in the fabrication of polyimide double-sided boards.

6.1 Tooling Requirements: The tooling requirements described in 5.1 for multilayer boards are applicable for double-sided boards.

6.2 Remaining Process Steps: Double-sided boards are treated very similarly to multilayer boards after the lamination step. Therefore, the remaining process sequence for double-sided boards is adequately described by the procedures in 5.5 through 5.19 for multilayer boards.

7. QUALITY ACCEPTANCE REQUIREMENTS, POLYIMIDE PRINTED WIRING BOARDS:

7.1 Ø Materials: Copper-clad materials utilized for double-sided boards and multilayer boards shall be in accordance with MIL-P-13949 and MIL-P-13949/10. The copper foil thickness shall be not less than 0.5 oz per sq ft (0.015 g/cm^2), 0.0007 in. (18 µm). B-stage material for multilayer boards shall meet the requirements of MIL-P-13949/13 Type Gi. Copper thickness shall be not less than 1 oz per sq ft (0.03 g/cm^2), 0.0014 in. (36 µm) on internal layers except the first and last internal layers shall be 2 oz per sq ft (0.06 g/cm^2), 0.0028 in. (71 µm).

7.2 Plating Requirements:

7.2.1 Recommended Thicknesses: The following minimum thicknesses are recommended:

Copper:	0.001 in. (25 µm)
Tin-lead (Unfused):	0.0003 in. (8 µm)
Tin-lead (Fused):	coverage

7.2.2 Plating Quality:

7.2.2.1 Plating shall be free of slivers or cracked overhang.

7.2.2.2 Plating shall be free of excess roughness which would impair function or solderability.

7.2.2.3 There shall be no evidence of chipping, peeling, or cracking.

7.2.2.4 Fused tin-lead plating shall be smooth, continuous, and free of voids that expose the underlying copper.

7.2.2.5 Plating voids shall not exceed 5% of the hole length or 10% of the total surface area and no more than 3 voids per hole. Voids shall not be allowed at terminal areas to plating interfaces or on both sides of a hole in the same plane.

7.3 Registration Requirements: Registration of all layers shall not exceed 0.014 in. (0.36 mm), determined by computing the difference of center-lines of terminal areas shifted to extreme positions (vertical cross-sections of hole). External terminal areas shall have a minimum annular ring of 0.005 in. (0.13 mm). Internal annular rings of multilayer boards shall be 0.002 in. (0.05 mm).

7.4 Conductor Pattern Requirements - Line Widths and Spacings: The conductor pattern shall be clearly defined. The minimum conductor width shall be 0.008 in. (0.20 mm). Conductor spacings should be as specified by the part drawing and as shown below: