1 SCOPe
This standard establishes the specific requirements for the design of flexible and rigid-flexible printed board applications and its forms of component mounting and interconnecting structures. The flexible materials used in the structures are comprised of insulating films, reinforced and/or non-reinforced, dielectric in combination with metallic materials. These interconnecting boards may contain single, double, multilayer, or multiple conductive layers and can be comprised wholly of flex or a combination of both flex and rigid.

1.1 Purpose The requirements contained herein are intended to establish specific design details that shall be used in conjunction with IPC-2221 and may also be used in conjunction with IPC-2222 for the rigid sections of rigid-flex circuits.

1.2 Classification of Products Classification type and use of products shall be in accordance with IPC-2221 and as stated in 1.2.1 and 1.2.2.

1.2.1 Printed Board Type This standard provides design information for different flexible and rigid-flex printed board types. Printed board types are classified as:

Type 1 Single-sided flexible printed board containing one conductive layer, with or without stiffener, and constructed using an adhesive or adhesiveless substrate (see Figure 1-1 and Figure 1-2).

Figure 1-1 Type 1 Single-sided Flexible Printed Board – Adhesive Substrate Construction

Note 1: Access Hole.
Note 2: Coverlay.
Note 3: Adhesive.
Note 4: Substrate.
Note 5: Copper Pad.

Figure 1-2 Type 1 Single-sided Flexible Printed Board – Adhesiveless Substrate Construction

Note 1: Access Hole.
Note 2: Coverlay.
Note 3: Adhesive.
Note 4: Adhesiveless substrate.
Note 5: Copper Pad.
Type 2 Double-sided flexible printed board containing two conductive layers with plated-through holes (PTHs), with or without stiffeners (see Figure 1-3 and Figure 1-4).

![Diagram of Type 2 Double-sided Flexible Printed Board – Adhesive Substrate Construction](image1)

**Figure 1-3 Type 2 Double-sided Flexible Printed Board – Adhesive Substrate Construction**

- **Note 1:** Access Hole.
- **Note 2:** Coverlay.
- **Note 3:** Adhesive.
- **Note 4:** Polyimide substrate.
- **Note 5:** Copper Pad.
- **Note 6:** Copper PTH.

Type 3 Multilayer flexible printed board containing three or more conductive layers with PTHs, with or without stiffeners, and constructed using an adhesive or adhesiveless substrate (see Figure 1-5 and Figure 1-6).

![Diagram of Type 2 Double-sided Flexible Printed Board – Adhesiveless Substrate Construction](image2)

**Figure 1-4 Type 2 Double-sided Flexible Printed Board – Adhesiveless Substrate Construction**

- **Note 1:** Access Hole.
- **Note 2:** Coverlay.
- **Note 3:** Adhesive.
- **Note 4:** Adhesiveless substrate.
- **Note 5:** Copper Pad.
- **Note 6:** Copper PTH.

**Type 3**

Note: When using Type 3 constructions, the number of layers should be limited due to the high adhesive content of the flexible adhesive systems (see 4.2.4.1).
**Type 4** Multilayer rigid and flexible material combinations containing three or more conductive layers with PTHs (see Figure 1-7 and Figure 1-8).
Figure 1-7 Type 4 Rigid-flex Printed Board – Adhesive Substrate Construction

Note 1: Coverlay.
Note 2: Adhesive.
Note 3: Adhesive based substrate.
Note 4: Copper Pad.
Note 5: Copper PTH.
Note 6: Prepreg.
Note 7: Rigid material.

Figure 1-8 Type 4 Rigid-flex Printed Board – Adhesiveless Substrate Construction

Note 1: Coverlay.
Note 2: Adhesive.
Note 3: Adhesiveless substrate.
Note 4: Copper Pad.
Note 5: Copper PTH.
Note 6: Prepreg.
Note 7: Rigid material.

Type 5 Flexible or rigid-flex printed board containing two or more conductive layers without PTHs (see Figure 1-9 and Figure 1-10)
Figure 1-9 Type 5 Flexible or Rigid-flex Printed Board without PTHs – Adhesive Substrate Construction

Note 1: Access Hole.
Note 2: Coverlay.
Note 3: Adhesive.
Note 4: Polyimide substrate.
Note 5: Copper Pad (Layer #1).
Note 6: Copper Pad (Layer #2).

Figure 1-10 Type 5 Flexible or Rigid-flex Printed Board without PTHs – Adhesiveless Substrate Construction

Note 1: Access Hole.
Note 2: Coverlay.
Note 3: Adhesive.
Note 4: Adhesiveless substrate.
Note 5: Copper Pad (Layer #1).
Note 6: Copper Pad (Layer #2).

1.2.2 Installation Uses Flexible circuit designs are unique in each application; however; the following are some typical classes of use. It is recommended that the intended use be specified on the fabrication drawing. It may be necessary to
define specific tests for design verification on the master drawing. These categories can be used individually or in a combination.

Use A Capable of withstanding flex during installation (flex-to-install). See 5.2.3.2

Use B Capable of withstanding continuous flexing for the number of cycles as specified on the master drawing (dynamic flex). See 5.2.3.2

Use C High temperature environment (over 105 °C [221 °F])

Use D UL recognition

1.3 Revision Level Changes Changes made to this revision of the IPC-2223 are indicated throughout by gray-shading of the relevant subsection(s). Changes to a figure or table are indicated by gray-shading of the figure or table header.

2 APPLICABLE DOCUMENTS

The following documents form a part of this standard to the extent specified herein. If a conflict of requirements exist between IPC-2223 and those listed in 2.1, IPC-2223 takes precedence. The revision of the document in effect at the time of solicitation shall take precedence.

2.1 IPC


2.4.18.1 Tensile Strength and Elongation, In-House Plating

IPC-HDBK-840 Solder Mask Handbook

IPC-SM-840 Qualification and Performance Specification of Permanent Solder Mask and Flexible Cover Materials

IPC-2152 Standard for Determining Current Carrying Capacity in Printed Board Design

IPC-2221 Generic Standard on Printed Board Design

IPC-2222 Sectional Design Standard for Rigid Printed Boards

IPC-2226 Sectional Design Standard for High Density Interconnect (HDI) Printed Boards

IPC-2615 Printed Board Dimensions and Tolerances

IPC-4101 Specification for Base Materials for Rigid or Multilayer Printed Boards

IPC-4202 Flexible Bare Dielectrics for use in Flexible Printed Wiring

IPC-4203 Adhesive Coated Dielectric Films for Use as Cover Sheets for Flexible Printed Wiring and Flexible Bonding Films

IPC-4204 Metal Clad Flexible Dielectrics for Use in Fabrication of Flexible Printed Wiring
IPC-4562 Metal Foil for Printed Wiring Applications

IPC-7351 Generic Requirements for Surface Mount Design and Land Pattern Standard

2.2 Joint Industry Standards

J-STD-001 Requirements for Soldered Electrical and Electronic Assemblies

1. www.ipc.org

3 GENERAL REQUIREMENTS

General requirements shall be in accordance with IPC-2221 and as recommended in 3.1 through 3.4.2.

3.1 Design Modeling The design cycle should include full-sized three dimensional modeling of the design to assure correct dimensioning and layout of flexible and rigid circuit areas (see Figure 3-1). It is recommended that physical mockups of the design be fabricated to evaluate stiffness, flexibility, fit or other properties before releasing the design for fabrication.

3.2 Design Layout The circuit layout depicts the physical size, flexible section, and location of all electronic and mechanical components, and incorporates the electrical schematic, and and the electrical requirements (e.g., impedance, amperage, voltage) to allow preparation of documentation and artwork.

Design activity should work closely with manufacturer to optimize producibility as early in the design process as possible.

Figure 3-1 Dimensional Modeling

Note 1: Prepare a model of the package to be wired.
3.2.1 Mechanical Layout Efficiency (Consider Final Panelization) Because flex designs may take a variety of shapes, it is recommended that the designer consider the use of folding to accomplish efficient panelization (see Figure 3-2). Layout efficiency is important because of the benefits in reduced processing cost and material usage. However, these savings can be offset by the labor required for the folding in the final assembly. Each fold will change the coverlay access opening orientation of the single-sided flexible circuit.

To achieve panel efficiency, consideration should be given to alternative interconnections such as discrete wiring. In many cases, it is possible to eliminate costly or lengthy flex extensions by the simple attachment of wiring or cable.

3.2.2 Fabrication Drawing Recommendations Separate views showing an installed flex configuration should be added to the fabrication drawing. The intent is to provide the fabricator with the locations of the critical areas that are to be folded or flexed.

Application specific requirements not covered in this standard shall be specified on the procurement documentation. Drawings should contain a detailed list and description of the materials contained in the flex construction (e.g., materials stack-up, finished copper weight, reinforced/stiffened areas and critical thickness areas). A cross-sectional view is recommended. Critical thickness areas (and measurement location) should be clearly defined. Critical bend radius locations should be identified with reference dimensions.

3.3 Schematic Pin out considerations should be considered early in the design to avoid complicated and costly crossovers in the flex connectivity. Pin out considerations can reduce layers and therefore reduce cost.

3.4 Test Requirement Considerations Electrical testing for flex circuits may include connectors and components. Specifications need to be given to the fabricator as to the electrical requirements of the end product. Design requirements for test considerations should be in accordance with IPC-2221.

Flex/Rigid-flex printed boards have some different/unique test considerations. Examples include:

- Fixturing – Because of the flexibility, tooling holes may be used to obtain stability during test
- Staying on grid is not a requirement
- Overlapping appendages – Areas of boards might be inaccessible by a bed of nails tester when folded
- Variable thicknesses among flex and rigid areas
Note: Flexible area thicknesses are often not specified.

3.4.1 Environmental The end use environment should be considered during design and material selection.

3.4.2 Mechanical/Flexural To evaluate flex life, it is recommended that flexural testing be done in a manner that reflects intended end product use.

Note: Materials with a high modulus have better dimensional stability than those with a lower modulus, though there is a trade-off in that the higher modulus materials have a relatively lower flex life.

4 MATERIALS

4.1 Material Selection Material type and construction is extremely important in designing flexible printed boards. Care should be taken in material selection to ensure compatibility with adjacent materials. All materials shall be specified on the master drawing. For clarification, it is suggested that cross-sectional views be used to highlight material selection. Examples of this are shown in Figure 4-1 and Figure 4-2.

For Type 4 (rigid-flex) constructions, in the areas where prepreg is used to define the rigid areas, the use of rigid laminate is not necessary.
Figure 4-1 Flexible Cross-Sectional Construction Examples

Note 1: Coverlay.
Note 2: Flex laminate.
Note 3: Bond ply.
Note 4: Type 1, single-sided construction.
Note 5: Type 2, double-sided construction.
Note 6: Type 3, three layer construction.
Note 7: Type 3, four layer construction.
Note 8: Type 3, five layer construction.
Note 9: Type 3, six layer construction.
4.1.1 Material Options At the fabricator’s option, flexible metal clad dielectrics and adhesive coated dielectric films may be manufactured using individual components per IPC-4562, IPC-4202, and IPC-4203. In addition, materials per IPC-4204 and IPC-4203 may be substituted where individual components are specified. These documents group materials into slash sheets that are generic in nature. This means that materials meeting the minimum requirements have widely different typical properties. It is important to research the various products to choose the one best meeting the design requirements. The attributes that should be considered are:

- Moisture absorption
- Fire retardancy
- Electrical properties
- Mechanical properties
- Thermal properties
Example: The dimensional stability of IPC-4204/1 states a maximum value of 0.20% for Method “B.” Some manufacturers of IPC-4204/1 may produce material with a dimensional stability of 0.05% or better. Therefore it is recommended to consult the manufacturer if specific attributes are required that exceed the stated minimum or maximum value. Many attributes are also gage (thickness) dependent. Where possible the slash sheets define a specific value for different gages, i.e. Peel Strength.

The designer and fabricator should concurrently review material selection for cost, performance, and producibility.

4.2 Dielectric Materials (Including Prepreg and Adhesives)

4.2.1 Preimpregnated Bonding Material (Prepreg) Prepregs are used in the fabrication of rigid-flexible printed boards. “No-flow” and “low-flow” types are typically used in the bonding of the rigid sections. The higher glass transition temperature (Tg) materials used in these prepregs offer higher operating temperatures and lower Z-axis expansion coefficient, which directly affect PTH reliability for > 8 layers (see 5.2.2.2). Disadvantages of using these materials include lower dielectric strength and reduced flexibility.

When preimpregnated bonding materials are used on flexible and rigid-flex printed boards, the material shall be in accordance with IPC-4101. The areas that require adhesive and those to be free of adhesive shall be defined on the master drawing.

4.2.2 Adhesives (Liquid) Adhesives such as flexible epoxy, acrylic, RTV silicone or polysulfide can be used to provide strain relief/fillet at rigid-flex transition areas or at the transition of Type 1, Type 2 and Type 3 printed boards with partial stiffeners (see 5.2.8). These materials may be a potential out-gassing issue for specific applications.

4.2.3 Flexible Adhesive Bonding Films (Cast Adhesive or Bondply) Flexible adhesive bonding films are typically used in bonding multiple flexible layers and attachments used for thermal management or structural support. These materials offer high bond strength to the flexible dielectric. These bonding films may be formulated from low Tg resins to enhance adhesion and flexibility. In rigid-flex designs, these materials should be minimized or removed from the rigid sections to eliminate the problem of excessive Z-axis expansion.

When flexible adhesive bonding films are used on flexible and rigid-flex printed boards, the material shall be in accordance with IPC-4203. The areas that require adhesive and those to be free of adhesive shall be defined on the master drawing.

4.2.4 Conductive Anisotropic Adhesives Adhesives of this type can be used to bond multiple layers/printed boards (rigid and flexible) and provide an electrical and mechanical connection between vertically adjacent pads. The lateral conductivity of these adhesives is low and can maintain electrical isolation of laterally adjacent pads. The products are supplied in film form.

4.2.4.1 Flexible Metal Clad Dielectrics (Flexible Substrate Materials) These materials are combinations of dielectric films and metallic foils. The foils can be attached to the dielectric by several means (i.e., adhesive resins or by direct deposition). The dielectric can be cast onto the metallic foil. The cast dielectric laminates and direct deposition laminates are called adhesiessless substrates. The traditional laminate configuration is constructed via the use of adhesive resin to bond the dielectric film to the metal foil. The Tg of these adhesive resins is usually lower than the dielectric film. High layer rigid-flex designs are currently employing adhesiessless substrates to minimize the impact of the low Tg adhesives (see Table 4-1). As such it is recommended to avoid adhesive based substrates in all Type 4 constructions or high temperature applications (Use C), and also in Type 3 constructions where the layer count exceeds eight layers. Flexible metal clad substrates shall be in accordance with IPC-4204 or combinations of IPC-4562, IPC-4202, and IPC-4203.

Table 4-1 Characteristics of Typical Flexible Dielectrics

<table>
<thead>
<tr>
<th>Characteristic</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tg</td>
<td>Low</td>
</tr>
<tr>
<td>Adhesion strength</td>
<td>Low</td>
</tr>
<tr>
<td>Flexibility</td>
<td>High</td>
</tr>
<tr>
<td>Electrical isolation</td>
<td>Low</td>
</tr>
<tr>
<td>Thermal management</td>
<td>High</td>
</tr>
<tr>
<td>Structural support</td>
<td>High</td>
</tr>
<tr>
<td>Application</td>
<td>High</td>
</tr>
<tr>
<td></td>
<td>Polyester (with Adhesive)</td>
</tr>
<tr>
<td>------------------------</td>
<td>--------------------------</td>
</tr>
<tr>
<td><strong>MECHANICAL</strong></td>
<td></td>
</tr>
<tr>
<td>Flexing (R ~ 2.0 mm [0.079 in])</td>
<td>FAIR</td>
</tr>
<tr>
<td>Thermal Forming</td>
<td>YES</td>
</tr>
<tr>
<td>Modulus</td>
<td>2800 – 5500 Mpa</td>
</tr>
<tr>
<td>Tear Strength</td>
<td>800 g</td>
</tr>
<tr>
<td>Peel Strength (Ambient)</td>
<td>1050 N/M</td>
</tr>
<tr>
<td><strong>CHEMICAL/ENVIRONMENTAL</strong></td>
<td></td>
</tr>
<tr>
<td>Caustic (&gt; 20%)</td>
<td>EXCELLENT</td>
</tr>
<tr>
<td>UV</td>
<td>POOR – PET</td>
</tr>
<tr>
<td>UV</td>
<td>FAIR – PEN</td>
</tr>
<tr>
<td>UL Recognition/Maximum Operating Temperature</td>
<td>PET², 85 °C</td>
</tr>
<tr>
<td>PEN³, 160 °C</td>
<td></td>
</tr>
<tr>
<td>Flame Retardancy</td>
<td>VTM-0 with FR Adhesive</td>
</tr>
<tr>
<td><strong>ELECTRICAL</strong></td>
<td></td>
</tr>
<tr>
<td>Dielectric Constant (1 MHz)</td>
<td>3.4</td>
</tr>
<tr>
<td>Dielectric Strength</td>
<td>4-5 Kv/25 µm</td>
</tr>
<tr>
<td>Insulation Resistance</td>
<td>10³ Ω-cm</td>
</tr>
<tr>
<td><strong>THERMAL</strong></td>
<td></td>
</tr>
<tr>
<td>Solder Processing</td>
<td>5 sec @ 205 °C</td>
</tr>
<tr>
<td><strong>ASSEMBLY</strong></td>
<td></td>
</tr>
<tr>
<td>Through Hole</td>
<td>LIMITED</td>
</tr>
<tr>
<td>Surface Mount (IR Reflow)</td>
<td>PEN, YES</td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td>Wire Bonding</td>
<td>NO</td>
</tr>
<tr>
<td>Chip (Direct Attach)</td>
<td>POOR</td>
</tr>
</tbody>
</table>

Note 1. The stated values are typical and will vary among different material suppliers. Consult the laminate manufacturer utilized by the fabricator for specific values.

Note 2. Polyethylene terephthalate.

Note 3. Polyethylene naphthalate.

4.2.4.2 Rigid Metal Clad Laminate Materials These laminate materials are combinations of metallic foils, resins, and woven or non-woven reinforcements. Resins used in these laminates offer a wide range of Tg (see IPC-2221 and IPC-4101).

4.2.5 Cover Materials Cover material is a generic term applied to thin dielectrics that are used to encapsulate circuitry, most commonly for flexible circuit applications. There are three types of cover materials, including coverlay, coverfilm and covercoat.

4.2.5.1 Coverlay The coverlay is a combination of applied film and integral adhesive made from separate layers of generically different chemistries that subsequently becomes a permanent flexible dielectric coating. The coverlay is used to
insulate/isolate the conductor layers on the surface of a flexible printed board. The coverlay is constructed of materials that can be flexed or formed in the intended use. The coverlay has access holes/windows to expose lands, PTHs and mounting holes. See IPC-4202, IPC-4203 and IPC-4204 for additional guidance.

4.2.5.2 Coverfilm The coverfilm is an applied film that subsequently becomes a permanent dielectric coating and is made from:

i) separate layers of generically similar chemistries,
ii) a homogeneous, single component, or
iii) a composite blend of two or more components.

4.2.5.3 Covercoat The covercoat is a liquid or semi-liquid coating that can be applied by dry film lamination, screening, spraying, or dipping/curtain coating and subsequently becomes a permanent dielectric coating. The resin can be formulated to be photoimageable. Intricate pad designs or tight pad spacing can be addressed or answered through the use of the photoimageable types. Some covercoats may be in accordance with IPC-SM-840.

NOTE: Covercoats may have reduced performance in tight bend radius applications.

4.3 Conductive Materials (Surface Finishes) Conductive materials shall be in accordance with IPC-2221 and as stated in 4.3.1 through 4.3.7.

4.3.1 Electrolytic Copper Plating

4.3.1.1 Flex-to-Install Applications Copper elongation should be 12% or greater for low flex life applications such as PTHs and circuit flex-to-install. The copper elongation is to be measured in accordance with IPC-TM-650, Method 2.4.18.1.

4.3.1.2 Dynamic Flex Applications Copper elongation above 18% is recommended if used in high flex life cycle applications. The copper elongation is to be measured in accordance with IPC-TM-650, Method 2.4.18.1. Additional electrolytic plating on the surface of the base material is not recommended on flexible areas. This suggestion is based on the fact that flexibility is adversely affected by thickness of the conductor. For dynamic applications, it is important to balance the circuit and coverfilms. For additional information, see 5.2.3.

4.3.1.3 Hole Plating Minimum average copper thickness for PTHs and vias are as shown in Table 4-2 and are the same for all classes. These are different from those in IPC-2221 because of the unique geometry of double-sided and low Tg materials used in multilayer and rigid-flex. For etchback of holes, see 9.2.2.1.

<table>
<thead>
<tr>
<th>Type</th>
<th>Overall Flexible Printed Board Thickness</th>
<th>Minimum Average</th>
<th>Minimum Thin Area</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>≤ 0.2 mm [0.008 in]</td>
<td>&gt; 0.2 mm</td>
<td>10 μm [394 μin]</td>
</tr>
<tr>
<td>Type 2</td>
<td></td>
<td>&gt; 0.2 mm [0.008 in]</td>
<td>20 μm [787 μin]</td>
</tr>
<tr>
<td>Type 3¹</td>
<td></td>
<td>&gt; 0.2 mm [0.008 in]</td>
<td>20 μm [787 μin]</td>
</tr>
<tr>
<td>Type 4¹</td>
<td></td>
<td>&gt; 0.75 mm [0.030 in]</td>
<td>30 μm [1,180 μin]</td>
</tr>
<tr>
<td>Type 3, 4²</td>
<td></td>
<td>&gt; 0.75 mm [0.030 in]</td>
<td>30 μm [1,180 μin]</td>
</tr>
</tbody>
</table>

Table 4-2 Minimum Average Copper Thickness, mm [in]

Note 1. Requirements for constructions with a ≤10% content of low Tg material.
Note 2. Requirements for constructions with a >10% content of low Tg material.

4.3.1.4 Selective Plating Requirements When the design is to be used in an application where conductor thickness is critical (e.g., flexibility, weight or tight impedance control), selective plating is suggested. A “pads only” (button plate) artwork shall be used only when defined by the design activity. This would be followed by a second resist and imaging application for
the etch process. These artworks can be generated by a number of methods. Special precautions may be needed when using clipped or shaved pads. Care should be taken in the design to use the largest possible pad size. It should be noted that some manufacturing processes will leave a thin amount of plating (flash) on the surface of the conductors. Certain areas may not tolerate any plating; these areas **shall** be stated on the master drawing (see Figure 4-3 and Figure 4-4).

**Note:** The amount of residual plating and copper crystalline structure on conductor surfaces can vary due to many plating variables (position in plating tank, position on plating rack, tank agitation, anode placement etc.).

![Figure 4-3 Plating for Adhesiveless Substrate Applications](image)

**Figure 4-3 Plating for Adhesiveless Substrate Applications**

- **Note 1:** Button (Spot) plate or pattern plate.
- **Note 2:** Panel plate.
- **Note 3:** Button (Spot) plate or pattern plate for dynamic applications.
- **Note 4:** Electrolytic copper.
- **Note 5:** Electrolytic flash copper.
- **Note 6:** Electroless copper.
- **Note 7:** Base copper.
- **Note 8:** Adhesiveless substrate.

**Note:** All views are prior to etching. Button (Spot) Plate or Pattern Plate for Dynamic Applications is applicable for installation use “B” (see 1.2.2).
4.3.2 Nickel Plating  Any nickel plating, external or as an undercoat, over the flexible section shall not be used due to its brittle nature. Cracks in the barrier coating will propagate and cause failure in the copper conductor.
4.3.3 Tin-Lead Plating Unless otherwise specified, all tin-lead plating shall be fused.

**Caution:** Not all flex circuit base materials are capable of surviving the temperatures required for fusing. See Table 4-1 for material selection.

4.3.4 Solder Coating Not all flex circuit-based materials are capable of surviving the temperatures required for solder coating, or soldering with any solder (RoHS compliant or non-compliant), without special precautions. See Table 4-1 for material selection.

**NOTE:** Table 4-1 is only a partial list of possible materials. It is recommended that the designer also check IPC standards for flexible, rigid and radio frequency (RF) materials or the manufacturer’s data sheets.

4.3.5 Other Metallic Coatings Other metallic coatings such as Electroless Nickel Immersion Gold (ENIG), Electroless Nickel Electroless Palladium Immersion Gold (ENEPIG), Immersion Tin, Immersion Silver, or Electrolytic Nickel-Gold shall be specified on the master drawing. Brittle plating material such as Nickel over the flexible section shall not be used (see 4.3.2).

4.3.6 Electronic Component Materials (Buried Resistors and Capacitors)

**Caution:** These materials should not be used in flex areas. They are primarily used in the rigid sections of rigid-flex printed boards.

4.3.7 Conductive Coatings for Shielding Conductive inks such as silver, copper, or carbon-filled polymers can be applied to the surfaces of dielectric layers to provide shielding. These coatings should be specified on the master drawing. Connection to ground can be made by direct attachment through an opening in the dielectric.

4.4 Organic Protective Coatings Organic protective coatings should be in accordance with IPC-2221 and as stated in 4.4.1 and 4.4.2.

4.4.1 Solder Mask Though commonly used in rigid printed boards, solder mask selection should be considered if used in any flex area. There are specific products that are designed to be used in a wide variety of flex applications. See IPC-HDBK-840 for additional guidance on using solder mask within flex areas.

4.4.2 Conformal Coating Conformal coatings should generally be omitted from flexible areas to avoid stiffening of the flex structure and flaking or de-bonding of the conformal coating from the flex substrate/surface.

4.5 Marking and Legend It is good practice to avoid marking in the flexible printed board dynamic area. Marking shall be defined on the procurement documentation or master drawing.

5 MECHANICAL AND PHYSICAL PROPERTIES

Mechanical and physical properties shall be in accordance with IPC-2221 and as stated in 5.1 through 5.4.1.

5.1 Fabrication Requirements See 3.1 through 3.2.1 for design modeling and mechanical layout efficiency.

5.1.1 Bare Printed Board Fabrication The manufacturer should be consulted on the panel sizes available and useable area on the panel.

Consideration should be given to the following:

- Part Spacing
Panel utilization considerations become more important as production quantities increase and cost per printed board becomes a driver compared to the overall set-up cost of low quantity/prototype work.

5.1.2 Roll to Roll Fabrication The manufacturer should be consulted on web width and roll length, as the variations are determined by equipment and material thickness used in the application.

5.2 Product/Printed Board Configuration

**Note:** The printed board standardization figure in IPC-2221 may not apply to flexible or rigid-flex printed boards due to unique profiles. Nesting (non-rectangular shapes) should be considered as a more efficient utilization technique (see Figure 5-1).

5.2.1 Circuit Profile (Outline) The exterior outline of the proposed circuit should not waste raw material. If the circuit has peninsulas or fingers extending in many directions, material costs may be higher than necessary. Folds in the circuit may permit extensions to be manufactured near the main body of the circuit and provide a more compact, rectangular circuit outline for processing. Circuit profiles can be obtained using steel-rule dies, laser trim, routing or hard tooling.

5.2.1.1 Minimum Radius (Flexible Sections) The minimum radius on inside corners of the part profile should be 1.5 mm [0.060 in]; however, larger radii will make a more reliable part and be more resistant to tearing (see Figure 5-2 and Figure 5-3). Additional materials may be added to the inside radii to provide increased tear resistance (see Figure 5-4).
5.2.1.2 Hole to Edge Distance (Flexible and Rigid sections) The minimum distance between exterior edges and the edge of interior nonplated-through holes (NPTH) and cutouts should be designed such that the final residual material is no less than 0.5 mm [0.020 in] from the exterior edges. Feature location, size tolerance and outline tooling tolerances should be considered for the designed distance.

5.2.1.3 Hole to Edge Distance (Rigid-to-Flex Transition Section) The minimum distance from the rigid-to-flex transition area to the edges of clearance holes should not be less than 1.9 mm [0.075 in].

5.2.1.4 Counterbores and Countersinks Counterbores and countersinks should be avoided due to the complexities of accurate machining with practical thickness tolerances.

5.2.1.5 Variable Thicknesses The rigid laminated sections of a multilayer flex and rigid-flex printed board should be the same thickness to accommodate the processing of PTHs. Sequential lamination or varied thickness adds to processing and cost.

5.2.1.6 Strain Relief Slits and slots should terminate in a 1.5 mm [0.059 in] diameter or larger hole, as shown in Figure 5-5. This situation arises when adjacent portions of the flexible printed board must move separately.
5.2.2 Rigid Area Considerations

5.2.2.1 Bow and Twist Due to the nature of flexible and rigid material combinations, special construction, tooling and/or fixturing may be required to meet surface mount requirements. For any product grouped in pallets for assembly purposes, there may be cutouts that affect bow and twist; therefore bow and twist requirements apply to the entire panel and shall be as agreed between user and supplier (AABUS).

5.2.2.2 PTH Reliability To minimize the amount of Z-axis expansion, the percentage of low $T_g$ (e.g., acrylic) adhesive in the rigid section should be kept to a minimum. This can be accomplished with the use of adhesiveless based substrate materials and partial coverlays of the flexible layers (see Figure 4-2). The partial coverlays of the flexible layers should be overlapped by the rigid sections by 1.27 to 2.54 mm [0.050 to 0.100 in]. It is recommended that prepreg materials in accordance with IPC-4101 be used as bonding films in the rigid section.

5.2.2.3 PTHs to Rigid-Flex Interface PTHs in the rigid section should not be less than 3.18 mm [0.125 in] plus one half of the PTH pad diameter from the rigid to flex interface when measured from the PTH center to the edge of the rigid material, as shown in Figure 5-6.

Caution: Violating this recommendation may compromise PTH reliability.

Note 1: Rigid region.
Note 2: Flexible region.
Note 3: Rigid to flex interface region.
Note 4: Minimum recommended spacing of 3.18 mm [0.125 in] plus one-half PTH pad diameter from the PTH center to the edge of the rigid material.
5.2.2.4 Additional Dielectric Material  Additional dielectric material of equivalent characteristics (i.e., pouch or cocoon) may be added to the rigid section construction to aid in processing, provided the overall thickness requirements of the rigid sections are not violated. This material is added to prevent process solutions from being entrapped between the un-laminated flex layers and leaving possible conductive material that would be difficult to remove and which could cause foreign object debris (FOD) in the next level assembly.

Care should be taken in removing pouch material from the flexible areas after processing to prevent damage to the flex layers. The protective dielectric material should be kept to a minimum extending from the rigid area after removal.

5.2.3 Flexible Areas

5.2.3.1 Flexible Area Considerations  Factors to consider in determining the total number of layers required (along with other interrelated considerations) are:

- Quantity of signal conductors required across the flexible portion(s)
- Line widths required for current-carrying capacity
- Spacing required for voltage isolation
- EMI shielding
- Impedance
- Voltage drop requirements
- Mechanically defined "real-estate" for routing the conductors (i.e., the width of the flexible portions).

While higher layer counts in the flexible areas (four conductive layers and above) can be produced, they are not recommended due to the larger bend radii required for the thickness and induced stresses to the material(s). Should higher layer count designs be required, it is suggested that mechanical testing be performed. It is to be noted that, as one progresses from a single-sided layer to a multilayer, there is a decrease in flexibility. For dynamic flex applications (Installation Use B), a double-sided layer would be the thickest construction recommended (see Figure 4-1).

Multilayer flexible printed wiring is not very flexible. If flexibility is required, it can be achieved by not laminating together specific sections of the cable, as shown in Figure 4-2. This type of design should be used in multilayer constructions containing more than four layers of flexible material.

Caution Note: Consideration shall be given to the flexible area between bonded areas in regard to the number and construction of substrates, and the length of the flexible area. Bending a flexible area between bonded areas will cause compression buckling of the unbonded substrates. The shorter the distance between the bonded areas, the more pronounced the buckling effect will be. Also, the higher the substrate count, the more pronounced this buckling effect will be. Increasing the bend angle will further aggravate this condition. Areas of buckling may cause small, isolated areas of reduced radii on the innermost substrate. See Figure 5-7. It is advisable to combine circuitry layers to reduce the number of substrates in shorter flexible regions.
5.2.3.2 Bend Area Conductor Considerations The flexible printed board should not be flexed or formed in an area where there is discontinuity in the coverlay, termination of plating or potting, or any other stress-concentrating feature. The acceptability of this condition is determined by the thickness of materials, the radius of the bend, and the severity of the operating environment. A Type 1 flex in a dynamic installation is more susceptible to conductor failure than a multilayer rigid flex in a flex-to-install environment.

For maximum dynamic flex life (Use B) and maximum reliability for flex-to-install (Use A), conductors in the bend area (see Figure 5-8) should adhere to the following considerations:

- Perpendicular to the bend
- Evenly spaced across the bend area
- Maximized across the bend area
- Without additional plated metals
- Uniform in width
- The neutral axis, where possible, should be located at the center of the conductor in the construction of the laminate (see Figure 5-9 and Figure 5-10)
- Conductors in double-sided circuits should not be placed directly over each other, which produces an "I" beam effect. This condition may be necessary due to electrical considerations; however, mechanical installation requirements shall be considered (see Figure 5-9)
- The number of layers in a bend area should be kept to a minimum
- Vias and PTHs in bend areas should be avoided
- When using rolled annealed copper, running the grain (machine) direction parallel to the conductors that run through the flexing area will increase flex endurance. In dynamic flexing applications or bend radii that are less than those described in 5.2.3.3, grain direction should be specified on the drawing. Note that specifying grain direction can impact cost.
A balanced construction can be achieved by using materials of equivalent modulus values and thickness on each side of the conductor. This is critical in dynamic flexible printed applications. Several design techniques are popular for approximating this condition, such as using coverlays and interdigitizing conductors front to back (see Figure 5-9 and Figure 5-10).

**Figure 5-8 Conductors in Bend Areas**

- **Note 1:** Bend area.
- **Note 2:** Dynamic - preferred. Flex-to-install – preferred.
- **Note 3:** Dynamic – not preferred. Flex-to-install – acceptable.
- **Note 4:** Dynamic – non-conforming. Flex-to-install – not preferred.

**Figure 5-9 Bend/Cr ease Areas Center Lines**

- **Note 1:** Coverlay.
- **Note 2:** Conductors.
- **Note 3:** Dielectric.
- **Note 4:** Neutral axis.
- **Note 5:** Preferred.
- **Note 6:** Acceptable.
- **Note 7:** Not recommended for dynamic folds.
5.2.3.3 Estimation of Minimum Bending for Flexible Circuits with Coverlay

Every flexible circuit design should be thoroughly evaluated to determine if the circuit construction will be able to withstand the bends and forms required by the application. Many factors can play into a flexible circuit's ability to be formed reliably (see also A.1, A.2 and A.3). Some of these factors include, but are not limited to:

- Circuit thickness
- Bend radius
- Bend Angle
- Copper type (RA, ED, etc.), thickness and elongation
- Forming method (e.g. cold form vs. hot form)
- Static or dynamic application
- Stress concentration features.

Since no formula or calculation can possibly predict every possible construction and/or usage, it is recommended that the circuit design be evaluated using the circuit's bend ratio. Bend ratio is defined as the ratio of the bending radius measured to the inside of the bend, to the circuit overall thickness in the bending area as shown in Figure 5-11.
Below are generally accepted guidelines for minimum bending radius for static applications bent 90° from flat for fully bonded circuits:

- Single sided       10:1
- Double sided      10:1
- Multilayer         20:1

For multilayer flexible and rigid-flex designs that utilize unbonded (loose leaf) flexible layers, the minimum bend radius is calculated based on the thickest individual flexible layer in the unbonded area and uses the same general guidelines as provided above. Refer to 5.2.3.1, 5.2.5.2, and A.9.

For dynamic applications bent 90° from flat, the guidelines for minimum bending radius are:

- Single-sided       100:1
- Double-sided       150:1
- Multilayer         Dynamic flexing not recommended

These are general guidelines which can be exceeded; however it is recommended that the designer consult with the fabricator for specific recommendations or applications. While these recommendations are not positive indicators for success or failure, they are rather indicators that a design may have an elevated probability of success or failure. The further the circuit design deviates from the guidelines, the greater the probability of success or failure.

5.2.4 Preforming Bend

Preformed bends or creases can be used to aid in the assembly of a flexible printed board into a chassis. The bend or fold does not need to hold its shape perfectly like sheet metal, rather it just needs to cause the flex to bend or fold in a predetermined location during assembly. It is good design practice to minimize the amount of preforming and bending due to the memory characteristics of some flexible dielectric materials.

Bends or creases can be formed in flexible circuits. The principle concern in low flex and flex-to-install applications is high ductility of the copper. The formability and reliability of a circuit is dependent upon the thickness and available ductility of the copper foil, as well as the substrate material and adhesive system. Three methods of permanently forming circuits are employed: cold forming, thermal forming, and forming through reliance on the properties of the copper used either as a conductor or simply left unetched as a reinforcement. Thermal forming is generally only used with substrate/adhesive combinations from the same materials family (i.e., polyimide, polyester, etc.). Thermal forming of the dielectric materials is highly dependent on adhesive, copper, and dielectric thickness. Preforming requires expensive manufacturing, tooling, and shipping containers. When preforming is required, it should be performed just prior to installation in the final unit. The addition of a strain-relief bar or some other means of support shall be provided if bending occurs close to a solder joint, PTH, or rigid to flex transition point. Bend and/or crease line drawing requirements are as follows:

- Bend/crease lines should be shown on fabrication and assembly drawings.
- Bend/crease line dimensions shall be depicted as a reference. The assembly drawing should show the finished folded configuration as a reference only.
- Bend/crease lines should be depicted as center lines and described as “bend line corresponds to center of arc formed when folded.”

Most preformed flexes will relax to a flat condition, even at room temperature. Elevated temperatures will hasten the process. This should be considered when forming flex to avoid objects. After relaxing, the flex may interfere with those objects. Any dimension on formed flexible features should be noted as a “reference” dimension, or very loosely toleranced. Any measurements should be taken in a restrained condition.

5.2.4.1 Bends or Folds (Greater than 90°) Folds should be kept uniform and designed to follow the surfaces of the package. These types of folds are only recommended for single-sided flexes or double-sided flexes where there is no I-
beaming of copper in the fold area. Terminal rows on single-sided flexible printed boards can be made to appear double-sided by folding the circuit. A forming tool with a known radius may be employed to control the bend radius and prevent conductor cracking. Microsectioning should be used to validate the process and verify that there is no cracking of the copper. The folded portion of the circuit may be secured with an adhesive. A crease, if required, shall be formed only one time, it shall not be opened again, and it should be bonded in place to ensure that the fold will not be exercised. Keep in mind the bend allowances required in tight corners. Regular folds should not be allowed to “bind” against parts since such binding areas can be points of weakness under vibration (see Figure 5-12). A bend or fold greater than 90° is defined as one that results in an acute angle measured between two adjacent sections across the bend or fold. See 9.3.5 for details on land access openings.

![Figure 5-12 Irregular Folds](image)

Note 1. Stress on solder joint.
Note 2. Flexible circuit rubbing structure or adjacent component.
Note 3: Acceptable.
Note 4: Nonconforming.

5.2.4.2 Flex-to-Install Radii (Two Layers or Less) The bend radii should be kept as large as possible. The minimum bend radius should be 10 times the overall thickness of the completed flexible circuitry. It is preferable that a flexible printed board be allowed to follow its own natural bend. Bends greater than 90° and containing a small radius should be avoided.

5.2.4.3 Bonded Multilayer Bending A bonded multilayer flexible printed board is not as flexible as a single or double-sided flexible printed board. If flexibility is required, it can be achieved by not laminating specific sections of the cable together. This type of design should be used in multilayer constructions containing more than four layers of flexible material.

The maximum number of copper layers in a multilayer configuration should be limited to four layers if bending of the parts is required (see 5.2.3.1). A recommended bend radius for greater than 2 bonded layers in the bend area is 20 times the overall thickness. However, once bent, the flexible multilayer shall not be flattened or re-bent on the same axis. Since all of the materials are inherently less stable than those used to produce rigid multilayer printed boards (the multilayer flexible printed board depends on the conductors being able to bend freely without elongation), a third or fourth layer would bring the conductor too far from the neutral bend axis to prevent the copper from stretching.
5.2.5 Differential Lengths

5.2.5.1 Differential Lengths (Flexible Printed Boards) When an assembly application requires that two or more flexible printed boards be selectively bonded or fastened together at the same common termination points at either end and are bent at least 90°, but without forming an "S" or Omega bend “Ω” - shape in a single flex location, then differential board lengths should be incorporated in the design of the part (see Figure 5-13 and Figure 5-14). These differential board lengths are calculated for the bent portion of successive printed boards.

Figure 5-13 Differential Printed Board Lengths

Note 1. Basic inner bend radius (r₀) to the first flex layer (See 5.2.5.1.1)
Note 2. Effective bend radius (r₁) of the first flex layer (See 5.2.5.1.1)
Note 3. Flexible Printed Board Layer A
Note 4. Flexible Printed Board Layer B
Note 1. Basic inner bend radius to the first flex layer (See 5.2.5.1.1)
Note 2. Effective bend radius of the first flex layer (See 5.2.5.1.1)

Figure 5-14 Differential Printed Board Lengths, Rigid-Flex

The following is a general expression for the differential length calculation. The calculation assumes the following:

- Only a single bend radius per flexible printed board is considered
- The radius is assumed to be circular (i.e., a portion of the circumference)
- All flexible layers are assumed to be equal in thickness and separated by an adhesive layer of relatively small, uniform thickness $T$
- The effective bend radius of each layer is its neutral axis, $T$
- The radius dimension should be to the worst case location tolerance (longest distance ($r_0$)) of the two end points
- Staggered lengths are sometimes used to terminate multilayer flexible printed wiring to a connector.

5.2.5.1.1 Calculation The basic bend length or portion of a circumference is given by:

$$L_0 = 2\pi r_0 \left(\theta \div 360\right) \quad [\text{Equation 1}]$$

where: $L_0 = \text{length}$

$$r_0 = \text{basic inner bend radius to the first flex layer}$$

$$\theta = \text{the angle, in degrees, through which the flex is bent}$$

therefore, $L_0 = 2(3.1416) \left( r_0 \right) \left( \theta \div 360 \right) = 0.0175 \ r_0 \ \theta$

Therefore, the effective bend radius of the first flex layer ($r_1$) is:

$$r_1 = r_0 + T_L/2 \quad [\text{Equation 2}]$$

where: $T_L = \text{total thickness of the flex layer, including coverlay}$. Thus the effective bend radius is the distance from the center of the curvature to the neutral axis of the flex layer. The length of the curved portion of the first flexible layer is:
As successive layers are added to the first layer, the effective radius of the n\textsuperscript{th} layer is:

\[ r_n = r_1 + [(T_L + G) (n-1)] \]  \[\text{Equation 3}\]

where:  
G = thickness of the adhesive joint  
n = number of layers

**Note:** G is only used if the layers are to be laminated together. If the layers are to remain separated, then G = 0 and disappears.

The length of the curved portion of the n\textsuperscript{th} layer is:

\[ L_n = 0.0175r_n\theta \quad \text{or} \quad L_n= 0.0175\theta [r_1 + (T_L + G) (n-1)] \]  \[\text{Equation 4}\]

The differential length of interest is the difference in the curved length of the first flexible tape and the curved length of any successive layers:

\[ \Delta L = L_n - L_1 \]

From the above,

\[ \Delta L = 0.0175 \theta (r_n-r_1) \]

\[ = 0.0175 \theta [r_1 + (T_L + G) (n-1)] -0.01750 \theta [r_1] \]

Combining the above equation gives:

\[ \Delta L = 0.0175 \theta [(T_L + G) (n-1)] \]  \[\text{Equation 5}\]

**Note:** The expression for the bend radius (r) has dropped out of Equation 7 and \( \Delta L \) is governed to only be the degrees of the bend, the number of layers, the thickness of the flexible layers, and the associated adhesive bond joint thickness.

For example, if:

\[ \Delta L = \text{increase in length per layer per bend} \]
\[ \theta = 90^\circ = \text{angle of bend in degrees} \]
\[ T_L = 0.381 \{0.015\} = \text{thickness of layer in mm [in]} \]
\[ G = 0.025 \{0.001\} = \text{thickness of adhesive per layer in mm [in]} \]
\[ n = 4 = \text{number of layers} \]

Then, from Equation 7, the differential lengths required are as follows:

<table>
<thead>
<tr>
<th>n</th>
<th>( \Delta L )</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.</td>
<td>0.0175 (90) (0.381 [0.015] + 0.025 [0.001]) (1-1) = 0 mm [0 in]</td>
</tr>
<tr>
<td>2.</td>
<td>0.0175 (90) (0.381 [0.015] + 0.025 [0.001]) (2-1) = 0.639 mm [0.025 in]</td>
</tr>
<tr>
<td>3.</td>
<td>0.0175 (90) (0.381 [0.015] + 0.025 [0.001]) (3-1) = 1.278 mm [0.050 in]</td>
</tr>
<tr>
<td>4.</td>
<td>0.0175 (90) (0.381 [0.015] + 0.025 [0.001]) (4-1) = 1.918 mm [0.076 in]</td>
</tr>
</tbody>
</table>
5.2.5.2 Differential Lengths (Multilayer and Rigid-Flex) The bookbinder design of an unbonded flex area can be used in regions where a sharp bend (radius to thickness ratios < 6) is required. This technique uses progressive lengths in the flex area (see Figure 5-15) and is costly to manufacture because of tooling complexity, processing difficulties, and reduced yields. Calculations used are the same as those in 5.2.5.1.

Figure 5-15 Bookbinder

5.2.5.3 Staggered Flexible Layer Bands (Multilayer and Rigid Flex) This alternative to the bookbinder design can also be used in regions where a sharp bend (radius to thickness ratios < 6) is required. This technique uses staggered flexible layers in the flex area (see Figure 5-16 and Figure 5-17) to achieve a neutral axis bend for each of the flexible layer bands. The final minimum bend radius will be based on the thickest flexible layer band. This method has advantages over the bookbinder method: It does not require progressive lengths in the flex area and does not cause a hump in the process panel. Whereas the bookbinder is limited to one bend axis and can have two bend axes with extreme difficulty, this method can be used in multiple locations at multiple angles without added difficulty or risk. While there will be added cost for additional tooling and layer processing, the overall cost will be less than the bookbinder option.

Electrical requirements must be carefully considered when using this method due to the fact that conductors and spacing will most likely be necked down in the narrow bands. High speed designs will work if the line width and spacing can be maintained throughout the design, but will be affected if they change between the narrow bands and the rest of the board area.

The example in Figure 5-16 depicts a simple 8 layer rigid-flex with double-sided flex bands. 3 layer stripline bands can be used, although the acrylic bond ply will need to be transitioned to prepreg in the rigid area for Class 3 high reliability applications. This method can be used with more or less layers, provided there is adequate room to route conductors in the bands. Bands may be different widths as long as they do not overlap in the bend area. Allow at least a 0.381 mm [0.015 in] gap between bands to avoid chafing. Keep in mind the amount of room it will take to route all conductors and use the long edge of a rectangular board area for the flex transition if given a choice on which way to hinge the boards.
For higher layer counts it may be necessary to have more than one band occupy the same area in the transition. A technique called "stair stepping" is suggested to improve flexibility by maximizing vertical distance between bands. In Figure 5-17, for example, layers 8/9 would be positioned directly below layers 2/3 to achieve the maximum vertical gap between layers, and layers 10/11 would be directly below layers 4/5, etc. Keep in mind that this technique may only allow the rigid-flex to bend 90° depending on the transition length. If different copper weights or dielectric thicknesses are used in the construction, it is recommended as a best practice to make sure the thinnest and most flexible layers are used on the inside of the bend, and thicker layers are kept on the outside of the bend.

**5.2.6 Shielding** A conductive layer may be added to provide shielding effectiveness. This layer may be a metal particle filled resin applied in a thin coat over a signal layer or a metal foil. The shield shall be considered a layer in neutral axis calculations. To improve flexibility, the shield layer shall be kept to a minimum thickness. To further increase flexibility and increase adhesion, the shield layer may be latticed or crosshatched as electrical characteristics allow (see Figure 5-18 for a typical example).
5.2.7 Ground/Power Plane  When the ground or power is a separate layer or a large conductor, it is advisable to distribute, break up, or balance the conductor widths through the bend area. To improve flexibility and increase adhesion, the plane layer may be latticed or crosshatched as electrical characteristics allow. Shield openings shall be 0.635 mm [0.025 in] or larger, though shield openings larger than 1.78 mm [0.070 in] can affect EMI and performance. See Figure 5-19.

5.2.8 Stiffeners and Heat Sinks  When stiffener, heat sinking materials, or other attachments are required, material types (metallic and nonmetallic), size, thicknesses, and adhesive type shall be specified on the master drawing. Size and registration of access holes in the stiffener to termination holes in the flexible and rigid-flex printed board shall be defined on the master drawing. Allowances should be made for tooling holes common between the printed board and the stiffener/heatsink. Access holes in stiffeners and heat sinks to expose lands should be at least 0.254 mm [0.010 in] larger in diameter than the land to allow for registration tolerances and adhesive squeeze-out. The edge of the stiffener next to the flexible portion of the circuit should be radiused, chamfered, or adhesive filleted to prevent damage to the flexible printed board.

5.2.9 Strain Relief Fillet Guidelines for Flexible and Rigid-Flex Printed Boards  When designing rigid-flex printed boards, no-flow prepregs are often utilized to prevent excessive resin flow onto the flexible surfaces of the printed board during lamination. The cured resin, which has sharp glass-like qualities, can cut/tear the underlying flexible laminates if flexed at the interface. To prevent this damage from occurring, a strain relief fillet is often applied at the rigid/flex interface to extend the bending area past the resin squeeze-out.

When considering whether or not to apply a strain relief fillet it is important to consider the minimum distance between the outer surface of the flexible materials to the outer surface of the rigid portion of the rigid-flex printed board, as shown in Figure 5-19. A minimum vertical distance of 0.254 mm [0.010 in] is preferred in order to give the printed board fabricator enough space to physically apply a strain relief fillet. Distances less than this preferred range allow for the possibility of the fillet extending beyond the rigid surface, potentially interfering with component placement, or the fillet shape not being a fillet but rather a bead. Fillet must extend beyond the squeeze out area.
Figure 5-19 Stiffener Thickness Preferred in Order to Apply Strain Relief Fillet

*Note 1.* Minimum height of 0.254 mm [0.010 in] is preferred in order for the printed board fabricator to have enough space to apply the strain relief fillet.

When adhesive fillets (strain reliefs) are used at the transition of the flexible and rigid portions of Type 4 printed boards, or at the transition of Type 1, Type 2, and Type 3 circuits with partial stiffeners, the fillet requirement should be defined on the master drawing. A large tolerance is required due to the flow characteristics of these materials. This is recommended to reduce stress at the interface (see Figure 5-20). The materials can be flexible epoxies, acrylics, RTV silicones, polysulfides, etc. The strain relief length horizontal dimension is typically 1.0 - 2.5 mm [0.039 - 0.098 in] from the rigid to flex interface.

Figure 5-20 Strain Relief

*Note 1.* Type 1, 2 or 3 Flexible Circuit.
*Note 2.* Strain Relief.
*Note 3.* Stiffener.
*Note 4.* Typical strain relief length of 1.0 - 2.5 mm [0.039 - 0.098 in].

**5.3 Assembly Requirements**  Assembly requirements **shall** be in accordance with IPC-2221 and as stated in 5.3.1 through 5.3.7.

**5.3.1 Mechanical Considerations**  Specific mechanical characteristic data should be obtained from IPC-4202, IPC-4203, IPC-4204, and the material manufacturer’s data sheets.

**5.3.2 Array Sub-Pallets for Flexible and Rigid Printed Boards**  Flexible and rigid-flex printed boards may be partially routed with “hold-down” tabs and retained in a sub-pallet to facilitate subsequent handling, assembly and/or test procedures. Typical sub-pallets will include multiple parts. Often flexible printed board product with non-symmetrical outlines may be “nested” for best fit and sub-panel utilization.

Array sub-pallets, if required, **shall** be part of the design documentation and required features / dimensions **shall** be documented. Additional tooling holes, conductive patterns and areas for coupons may be added by the supplier within the non-part portion of the sub-pallet. “Keep-out” zones for any supplier-added features or other restrictions on the array sub-pallet **shall** be documented in the design. The maximum number of defective parts (i.e., “x-outs”) within the array **shall** be documented.
Standard “break-away” procedures used for part removal from rigid printed board sub-pallets may not be effective on flexible or rigid-flex constructions. Flexible printed board materials will not “crack” and separate compared to rigid-only constructions. Use of cutting tools at the break-away tab locations during de-panelization are recommended to prevent tearing or delamination.

After assembly, depalletization methods such as mechanical or laser routing should be used for tab removal. Tab removal using “break-away” methods shall not be used on flex or rigid-flex product, as damage to the flex layers could result.

5.3.3 Single Part Sub-Pallets. Rigid-flex or flexible printed boards with rigid stiffeners procured as a “1-up” or single part may be supplied with “rigid rails” (see Figure 5-21). Similar to multi-part arrays, rigid rails will aid in handling and part installation, and provide protection against over-bending of the flex portion during storage and assembly.

Rigid rails, if required, shall be part of the design documentation. Standard “break-away” procedures used for removal of the rigid rails may not be effective on flexible or rigid-flex printed boards. See 5.3.2 for additional guidance.

![Figure 5-21 Rigid Rails for Single Part Sub-Pallets](image)

Note 1: Flex.
Note 2: Rigid rail.

5.3.4 Non-Palletized Flexible and Rigid-Flex Printed Boards Special fixturing may be required for assembly operations on flex and rigid-flex printed boards. Provide layout considerations for holding the assembly securely.

5.3.5 Moisture If the dielectric is not free of moisture, soldering temperatures may cause entrapped moisture to boil. Depending on the amount of moisture present when the flexible printed board reaches the soldering temperature, delamination may be violent, literally blowing the circuit apart, or mild, producing small blisters, which could grow into serious delaminations.

Bake out shall be a standard process for mitigating moisture/blow-out risk. The lapsed time between baking and subsequent soldering operations is also important. Generally, flex and rigid-flex printed boards require longer bake out times than equivalent rigid printed boards.

5.3.6 Infrared Preheats and Reflow Polyimide films absorb IR energy quickly because of their color. They must be monitored carefully when using IR preheats or reflow to prevent excessive heat buildup within the flexible printed board. External fixturing and shielding may be required.
5.3.7 Adhesive $T_g$ Flexible printed boards can contain adhesives with very low $T_g$. For this reason, all thermal excursions during assembly should be monitored to prevent damage such as delamination and blistering. Preheat and soldering dwell times should be kept to a minimum. Special heatsinks or heat shields may be required.

5.4 Dimensioning Dimensioning systems should be in accordance with IPC-2221, IPC-2615 and as stated in 5.4.1.

5.4.1 Datum Features Datum feature or origins should be designated in each rigidized or localized termination area. The dimensions referenced to a datum for hole patterns should not include a flexible section.

The use of multiple datums, as shown in Figure 5-22, is a common technique for reducing or eliminating the effects of variable shrinkage or process distortion in flexible materials. One of the primary values of flexible printed boards is their ability to be formed into three dimensions. With this flexibility comes inherent dimensional instability relative to rigid printed boards or solid metal objects. Providing more positional tolerance from datum to datum, while specifying a tighter tolerance within each datum or termination area, allows for ease of manufacturing of the circuit without compromising subsequent installation of components in each termination area.

In assembly, the variable distance between datums can easily be accommodated by designing a small amount of slack or a service loop between the termination areas. For this reason, there is no loss of dimensional control or inferior quality associated with multiple datums.

| Note 1 | Datum lines within printed board (Non-functional holes may be used). |
| Note 2 | Additional datums (Supported or Constrained Condition) |
| Note 3 | Datum 1. |
| Note 4 | Datum 2. |

Figure 5-22 Establishing Datums

6 ELECTRICAL PROPERTIES

Electrical properties shall be in accordance with IPC-2221 and as stated in 6.1 and 6.2. See IPC-2152 for considerations in calculating current carrying capacity of conductors. The tables and charts contained in IPC-2221 and IPC-2152 should be considered as guidelines only.
6.1 Electrical Considerations  Continuous improvements in material and circuit technologies may allow reduced electrical clearances and/or greater current capacity. Specific electrical characteristic data should be obtained from the material manufacturer’s data sheets.

6.2 Impedance and Capacitance Control  Due to the high speeds required of electronics today, controlled impedance requirements for conductors are commonplace. It is important to understand the factors that affect impedance in flex/rigid-flex printed board applications as to avoid signal integrity issues as described in 6.2.1 through 6.2.8.

6.2.1 Impedance Modeling Software  Separate impedance models will need to be created for rigid-flex designs for the same required signal in both rigid and flexible areas for each controlled impedance requirement. Impedance modeling software permits optimizing key variables of impedance model type (micro-strip, stripline, etc.), material thickness, material types (with associated dielectric constants) and line width / spacing to achieve the desired results consistent with design objectives. It is recommended that the printed board design be coordinated with fabricators to optimize conditions.

6.2.2 Material Thickness and Stack up  The thickness and stack-up of the materials used in the flex/rigid-flex construction dictate the required line width and spacing to meet the specified impedance. It is recommended to design using readily available material thickness options.

6.2.2.1 Dielectric Constant (Dk, Er)  Most materials used in flex / rigid-flex constructions will have dielectric constants between 2.0 and 4.5. Selection of the best materials for an application will depend on the end use, cost and fabrication considerations. Obviously, in most rigid-flex constructions, dielectric constant will be different for materials in the rigid versus the flex sections. Coordinate with the supplier as dielectric constants published by the material supplier may not always align with the supplier’s typical empirical results.

6.2.2.2 Cut-back Coverlay/Bondply/Adhesive:  Coverlay and bondply containing adhesive and adhesive sheets are commonly pulled back from the rigid section in a rigid-flex design to maximize PTH reliability. It is recommended to permit the fabricator to adjust line widths at these transitions regions to accommodate for the different dielectric constants of the rigid and flexible materials. Both the designer and fabricator, when evaluating controlled impedance models for both the rigid and flexible regions, should adjust the dielectric thickness to minimize line width difference between the rigid and flexible regions, as shown in Figure 6-1. Differences in the line width of 0.025 – 0.051 mm [0.001 – 0.002 in] between rigid and flex regions generally will not compromise signal integrity, although it should be noted that impedance calculations should be performed at each point where the dielectric thickness, type, or conductor width changes to ensure that reflections and noise due to mis-matched impedance are kept to a minimum. The fabricator should also consider the effects of the increased thickness of the flex dielectric material on flexibility.
6.2.3 Conductor Pitch

The conductor pitch for differential pairs used for controlled impedance should be kept constant along the length of the conductor to avoid unwanted signal noise / signal mismatch. The impedance coupon for a given layer / signal also is based on a consistent differential pair pitch. Pads are often wider than the signal conductors, so keep this in mind when laying out the necessary pitch for differential lines. Figure 6-2 shows preferred and non-preferred layouts for differential lines. Device and signal requirements of “loosely coupled” (about 0.38 mm [0.015 in] edge to edge spacing) or “tightly coupled” (about 0.127 mm [0.005 in] edge to edge spacing) differential signals will obviously impact conductor pitch.
Note 4. Differential pair 2.
Note 5. Ground.
Note 6. Pitch 1 for differential pair 2.
Note 7. Pitch 2 for differential pair 2.
Note 8. Pitch 1 and Pitch 2 are not constant for differential pair 2, which is a non-preferred layout.
Note 9. A constant pitch among differential pairs is a preferred layout.
Note 10. It is recommended to leave copper behind for both shielding and dimensional stability.

6.2.4 Narrow Conductors When modeling for impedance control, a dielectric thickness able to achieve a finished line width of 0.127 mm [0.005 in] or greater on 0.5 oz. copper (or less) is recommended to minimize variation related to etch processing. In addition to the reduced flex reliability of narrow conductors, there are challenges with respect to maintaining a controlled impedance, especially for moderate and long line lengths (> 102 mm [4.0 in]).

6.2.4.1 TDR Measurement Considerations for Narrow Conductors Normally a Time Domain Reflectometer (TDR) is used to measure characteristic impedance ($Z_0$) of a controlled impedance line. For very narrow line widths, DC resistance can significantly contribute to the measurement. The measured effect of DC resistance is also doubled due to the fact that a signal must make a round trip in order to be detected by the TDR. The TDR Waveform in Figure 6-3 shows a Type 2 flex that is 102 mm [4.0 in] long and has a very narrow (0.051 mm [0.002 in]) line width. The design software predicts that the impedance of this line should be about 47 ohms. This model is accurate only at the beginning of the line. It is recommended that the effect of DC resistance be understood and openly discussed by fabricators and designers for narrow line widths (< 0.127 mm [0.005 in]) over long lengths (> 102 mm [4.0 in]).

![Figure 6-3 TDR Waveform for Type 2 Flexible Printed Board](image)

Note 1. $Z_0$ (ohms) vs Time (ps).
Note 2. $Z_0$ (ohms).
Note 3. Propagation Time (ps).
Note 4. 102 mm [4.0 in] length.
Note 6. DC resistance of 3.5 ohms can appear as a 7 ohm rise on TDR.

6.2.5 Differential Impedance Edge-coupled impedance models are generally preferred to broadside-coupled differential models as there will be no front to back layer alignment variation. Edge coupled signals are present on the same layer by definition. If a broadside-coupled design must be used, the signal layers should use the same core for best registration.
6.2.6 Unbonded Flexible Impedance Controlled Layers (“Loose-leaf layers“) Unbonded flexible layers (typical of double-sided micro-strip constructions), when flexed, naturally buckle, creating uncontrolled spacing between the individual flex circuits across the length of the flex as shown in Figure 6-4. When unbonded flexible impedance controlled layers are used in rigid-flex designs, impedance values may vary from the target value due to uncontrolled spacing between flex layers if the reference ground plane/impedance signal layers are placed on adjacent sides of the air gap. The overall impact in the flex region usually is not significant based on the fact that the closest reference plane layer (present on the same layer core) exerts the most influence on the controlled impedance value. If the application requires a tighter than recommended bend, or a need to eliminate the uncontrolled spacing between layers, then the design should consider using differential length flex layers (bookbinder).

![Figure 6-4 Buckling of Unbonded Flexible Layers](image)

6.2.7 Modifications to Shield Layers One of the major items that drive the thickness of dielectric layers in a flex circuit is the impedance requirement. Often it is desirable to sacrifice some shielding performance of a signal reference (or “ground”) layer for the sake of flexibility. This trade-off can be accomplished in several ways. Three techniques discussed in this section are by cross-hatching (etched patterns), applying thick-film conductor, or applying shielding film.

It is recommended that the designer and fabricator jointly understand the requirements that the part has for signal loss, impedance tolerance, electromagnetic interference (EMI), and flexibility since all of these factors are affected significantly by modifications to the shield layer. Available modeling software may not precisely predict absolute impedance or EMI characteristics of thin controlled impedance flex structures. In addition, the techniques applied to shielding vary significantly between designers and fabricators. Due to these realities, it is usually necessary for some empirical study, if the results of the variable(s) under consideration (cross hatch shield, for example) have not been previously validated.

6.2.7.1 Cross Hatching The advantages of this technique as shown in Figure 6-5 are that it has the least negative effect on loss and no novel materials (e.g. silver epoxy shielding) are required to implement. The disadvantage of this technique is that it results in the most degradation in EMI performance, especially at high frequencies. To maximize success with this technique, the following guidelines are suggested:

- Just like any other etched feature in a flex area, sharp corners on signal routing should be avoided. These act as potential points of failure when the material bends.
- In general, smaller openings in the cross hatch plane may offer improved impedance consistency. The opening should be sized based on dielectric thickness and line width objectives to meet the targeted impedance. Often suppliers will have suggested cross hatch impedance models with successful history.
6.2.7.2 Thick-Film Conductor Shielding (Silver Epoxy) The advantages of this technique are that it can be applied as an additive process. Electrical connections to the shield are made through contact of the shielding material to a copper (ground) conductor in the layer below. The connection to the silver epoxy layer shall not be made through a PTH. To minimize effects of the higher resistance of the shielding material compared to copper conductors, redundant connections of the shield layer to a ground guard line are recommended. The disadvantage of this technique is that loss may be higher at high frequencies, compared to a full copper reference plane.

6.2.7.3 Shielding Film The advantages of this technique are that no pattern design is required. The conductor in a shielding film cannot be patterned and thus it will have a uniform effect. EMI performance is generally superior to the other techniques. The disadvantage is the increase in signal loss when compared to a cross-hatched ground. This increase in signal loss can negatively impact the impedance, especially for longer lengths. Also, these films are specialized and not commonly used in traditional flex processing.

6.2.8 Dielectric Constant Changes Over Frequency Knowledge of the operating frequency of a design is necessary to address what dielectric constant is selected for the various materials of construction. While adhesiveless polyimide has very low loss and the dielectric constant does not change with frequency, this is not true for adhesives commonly used in flex applications. There are two reasons for this:

- The chemical structure of acrylics and pure epoxies cause the material to absorb electromagnetic energy as frequencies increase;
- The adhesives absorb a lot of moisture from the air. Water also absorbs electromagnetic energy especially at frequencies between 2-25 GHz.

TDR equipment used in fabrication shops may have long rise times and may not adequately measure effects at high frequencies. Generally an OEM’s TDR equipment is more advanced and operates with very short rise times. With these shorter rise times, effects at higher frequency are often observed. This can lead to discrepancy of measurement between OEMs and fabricators, so it is recommended that the measurement equipment be agreed upon by the fabricator and designer prior to agreement on controlled impedance specifications.

7 THERMAL MANAGEMENT

Thermal management shall be in accordance with IPC-2221.

Note: Thin dielectric materials have high electrical and low thermal resistance. The thermal characteristics of the specific flex dielectric should be reviewed for thermal transfer, $T_g$, and temperature index.

8 COMPONENT AND ASSEMBLY ISSUES
Because of the wide range of $T_g$ materials, the design should be reviewed for thermal management with respect to component assembly methods and techniques.

8.1 General Placement Requirements  General placement requirements shall be in accordance with IPC-2221 and as stated in 8.2 through 8.6.

The visual fiducials used in the placement of parts should be located in rigidized or localized component mounting areas.

8.2 Standard Surface Mount Requirements  Standard surface mount requirements shall be in accordance with IPC-2221 and as stated in 8.3.

8.3 Lands for Surface Mounting  Surface mount should only be used in rigid portions of rigid-flex circuits or nonbending areas of flexible circuits. The selection of the design and positioning of the land geometry in relation to the part may significantly impact the solder joint. The designer must understand the capabilities and limitations of the manufacturing and assembly operations (see IPC-7351). Special consideration should be taken for coverlay openings to compensate for adhesive squeeze-out over surface mount lands.

8.4 Constraints on Mounting to Flexible Sections  Designs shall not place a component in an area of continuous flexing or in an area that will be flexed, folded, or flexed as part of a flex-to-install. Leads mounted through flexible material should be fully clinched. If unclinched leads are required, supporting hardware, encapsulant, or stiffeners should be designed as a part of the flexible printed board to ensure that no flexure related stresses are exerted on the solder joints.

8.5 Interfacial Connections  Interfacial connections on Type 2 flexible printed boards should be made by the use of clinched wires, followed by soldering, or PTHs. Interfacial connections on Type 3 and Type 4 flexible and rigid-flex printed boards should be made with PTHs only. Standoff terminals, eyelets, rivets, or pins should not be used to provide electrical connections and shall not be used to provide signal and power connections. Clinched wires used as interfacial connections are considered to be part of the assembly and shall be identified on the flexible printed board assembly drawing.

8.6 Offset Lands  Lands, when used in conjunction with clinched leads, may be located adjacent to (not surrounding) the lead termination hole. The land shall be of a sufficient distance from the hole to allow clipping of the part lead prior to unsoldering the part lead from the land.

9 HOLES/INTERCONNECTIONS

Holes and interconnections shall be in accordance with IPC-2221 and as stated in 9.1 through 9.3.

Note: For more information on materials and construction, see 5.2.2.2.

9.1 General Requirements for Lands with Holes  General requirements for lands with holes shall be according to IPC-2221 and as stated in 9.1.1 through 9.2.2.1.

9.1.1 Land Requirements  Land requirements shall be in accordance with IPC-2221. Filleting of conductor to land transition is recommended in flexible printed boards so as to reduce stress conditions and increase manufacturing tolerances (see Figure 9-1).
### 9.1.2 Annular Ring Requirements

Annular ring requirements **shall** be in accordance with IPC-2221.

The standard fabrication allowance for interconnection lands given in IPC-2221, as used for determining the worst-case land-to-hole relationship, **shall** be substituted with the fabrication allowance for interconnection lands given in Table 9-1.

#### Table 9-1 Minimum Standard Fabrication Allowance for Interconnection Lands, mm [in]

<table>
<thead>
<tr>
<th>Level A</th>
<th>Level B</th>
<th>Level C</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.5 mm [0.020 in]</td>
<td>0.4 mm [0.016 in]</td>
<td>0.3 mm [0.012 in]</td>
</tr>
</tbody>
</table>

**Note 1.** For copper weights > 1 oz/sq.ft., or 34 µm [1,339 µin], add 0.05 mm [0.002 in] minimum to the fabrication allowance for each additional oz/sq.ft or µm of copper used to allow for increased undercut due to the chemical etching process.

**Note 2.** For more than eight layers, add 0.05 mm [0.002 in].

**Note 3.** For designs with multiple laminations to create additional (blind or buried) via structures, add 0.05 mm [0.002 in].

**Note 4.** See IPC-2221 for definitions of Level A, B, and C producibility levels.

Level C may not be producible or may be cost prohibitive for large (> 304.8 mm [12.0 in]) printed boards or printed boards with high (> 10) layer counts. If Level C is required, consider Land-to-Conductor Transition rules in 9.1.9.

### 9.1.3 Eyelet or Standoff Land Area Considerations

When eyelets or standoff terminals are used, the lands on Type 1, Type 2, and external layers of Type 3 and Type 4 **shall** be designed so as to have a minimum diameter of at least 0.5 mm [0.020 in] greater than the maximum diameter of the projection of the flange, eyelets, or standoff terminals. Special consideration should be taken for coverlay openings to compensate for adhesive squeeze-out. Eyelets or standoff terminals should not be used for electrical connections (see 8.5).

### 9.1.4 Land Size for Non-Plated Component Holes

The land size to be used **shall** be the largest possible land that will meet the spacing requirements. The minimum standard fabrication allowance **shall** be per Table 9-1.

### 9.1.5 Land Size for Plated-Through Component Holes

The land size to be used should be the largest possible land that will meet the spacing requirements, such as finished hole size plus 0.75 mm [0.030 in]. The land size used on the surface layers should be larger than the internal layers, if possible. The minimum standard fabrication allowance **shall** be per Table 9-1.

### 9.1.6 Thermal Relief in Conductor Planes

When a ground and/or power plane is used, the thermal relief pad **shall** meet the requirements of IPC-2221.
9.1.7 Surface Mount Components See IPC-7351 for surface mount land pattern design recommendations.

9.1.8 Nonfunctional Lands Nonfunctional lands are lands on internal layers that do not make an electrical connection. Nonfunctional lands should be included on internal layers for all PTHs except for high (> than 10) layer count designs and rigid-flex designs whose thickness exceeds 1.6 mm [0.063 in].

The plated barrels shall meet the same minimum spacing requirements as that for conductors on internal layers. The design shall use the standard fabrication allowance to account for PTH position and artwork misregistration, even though nonfunctional lands are not present, as the barrel of the hole is a conductor.

PTHs passing through internal ground, voltage and thermal planes without nonfunctional lands shall meet the same minimum spacing requirements as conductors on internal layers. It is preferred that the space to PTH clearance not change because the lands are missing.

When nonfunctional lands are removed, it is recommended that they are removed on alternate layers, preferably starting in the middle third of the printed board.

If nonfunctional lands are to be removed during the design process, it should be delayed until routing is completed in order to maximize conductor to copper barrel clearances. If nonfunctional lands are to be removed by the fabricator, it is recommended that a drawing note indicate the layers on which such removal is authorized.

Note: While fabricators should not remove nonfunctional lands without specific authorization, it is a common practice.

Table 9-2 provides considerations and concerns about using nonfunctional lands in the design.

<table>
<thead>
<tr>
<th>Reasons to Retain Nonfunctional Lands</th>
<th>Reasons to Remove Nonfunctional Lands</th>
</tr>
</thead>
<tbody>
<tr>
<td>Clearance is required for signal and plane layers for the maximum hole size, misregistration, manufacturing tolerance and electrical spacing. This is the same clearance as that for functional lands with the exception of the annular ring requirement.</td>
<td>Each conductor in close proximity to a land is an area susceptible to potential etching process defects. The presence of more lands increases the potential for more defects.</td>
</tr>
<tr>
<td>Omitted lands allow some routing tools to run conductors through the holes.</td>
<td>The pad stack is thicker than the areas of the board without lands. Lamination pressure will tend to generate resin starvation and cracked glass fibers.</td>
</tr>
<tr>
<td>Drawing review of interferences is easier when all holes can be visually identified by their lands.</td>
<td>Reduction of noise as stub lines.</td>
</tr>
<tr>
<td>The potential for resin cracks is reduced, especially for heavy copper layers.</td>
<td>Removal by the design function assures configuration control.</td>
</tr>
<tr>
<td>Lands on internal layers provide ribbed or three point support for the hole. There will be less propensity for plating separation.</td>
<td>As copper is heavier than dielectric, printed boards utilizing fewer lands will be lighter.</td>
</tr>
<tr>
<td>For designs that have acrylic adhesive in the PTH areas, the removal of nonfunctional lands can cause additional removal of the acrylic during plasma etch which may create plating folds and stress points after plating and possible fractured PTHs. This problem will occur after temperature cycling or thermal stress, particularly if the PTHs do not have component leads that are soldered in the hole. Further, this problem will not be noticed in the cross-sectional analysis unless the same exact construction is reviewed.</td>
<td>Lands can take up considerable space within a database.</td>
</tr>
</tbody>
</table>
9.1.9 Land-to-Conductor Transition The transition between a conductor and a supported or unsupported land should be strain relieved to prevent broken conductors at this transition interface. An adequate strain-relief can be attained in several ways, depending on the design density. The primary design requirement is that the transition should not be on or within the coverlay access hole diameter unless the land is filleted or of a teardrop shape, as shown in Figure 9-1. There are two acceptable designs (see Figure 9-1).

9.1.10 Unsupported Edge Conductors/Fingers Unsupported edge conductors/fingers are areas of circuits that are not encapsulated by coverlay or base materials and are designed specifically for connections.

The absence of these materials creates free-floating conductors that are accessible from either side of the flex substrate. This allows for easier direct connections to components or printed boards as heat for connections does not need to be applied through or into substrate materials. These fingers can be formed and designed in a variety of configurations, with some examples provided in Figure 9-2 through Figure 9-5. There are many other possibilities beyond those shown here.
Figure 9-5 Straight Fingers for Lap Soldering to Printed Board

When designing a flexible circuit for unsupported fingers, the foil options in Table 9-3 should be considered.
<table>
<thead>
<tr>
<th>Circuit Type Application</th>
<th>Thickness</th>
<th>UNS(1) Foil Alloy</th>
<th>Foil Type</th>
<th>Specification(s)</th>
<th>Slash / Temper</th>
<th>Temper description</th>
<th>Pros</th>
<th>Cons</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Type 1 Single Layer or Type 2 Double sided with two conductive layers</td>
<td>1 to 2 Oz</td>
<td>C11000 Rolled (Wrought)</td>
<td>IPC-4562</td>
<td>/5(3) or /8(3)</td>
<td>/5(3)</td>
<td>Hard</td>
<td>Higher tensile strength and elongation than /7</td>
<td>Thinner foil allows creation of fine line circuitry</td>
<td>Delicate and fragile.</td>
</tr>
<tr>
<td>Conventional Flex exposed on one or both sides</td>
<td>3 to 5 Oz</td>
<td>C11000 Rolled (Wrought)</td>
<td>IPC-4562</td>
<td>/5(3,4,5) or /6(3,4,5)</td>
<td>/5(3,4,5)</td>
<td>Hard</td>
<td>Very stiff fingers</td>
<td>Not recommended for forming</td>
<td>Potential for cracking foil</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>ASTM-B-152(6)</td>
<td>H04</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>IPC-4562</td>
<td>/5(3,4,5) or /6(3,4,5)</td>
<td>¼ Hard or ½ Hard</td>
<td>Recommended for forming</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>ASTM-B-152(6)</td>
<td>H01 or H02</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>IPC-4562</td>
<td>/7(3,4,5)</td>
<td>Soft (Annealed)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>ASTM-B-152(6)</td>
<td>O60</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>C17200 (Be Cu)</td>
<td></td>
<td>ASTM-B-194(6)</td>
<td>TD01 or TD02</td>
<td>¼ Hard or ½ Hard</td>
<td>Material properties allow fingers with better stability to align during assembly</td>
<td>Foil requires soldered fingers to be pre-plated with copper flash before soldering</td>
<td>May requires separate etchant tank Possible HES / OSHA concerns</td>
<td></td>
</tr>
<tr>
<td></td>
<td>C11000</td>
<td></td>
<td>IPC-4562</td>
<td>/5(3,4,5) or /6(3,4,5)</td>
<td>Hard</td>
<td>Very stiff fingers</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

(1) UNS: Uniform Number System
Sculptured Flex may be exposed one or both sides

<table>
<thead>
<tr>
<th>Sculptured Flex</th>
<th>Rolled (Wrought)</th>
<th>ASTM-B-152(6)</th>
<th>H04</th>
<th>Not recommended for forming</th>
<th>Potential for cracking foil</th>
</tr>
</thead>
<tbody>
<tr>
<td>≥ 6 Oz. (0.20 mm [0.008 in]) full thick areas with reduced areas in the (0.051 – 0.127 mm [0.002 to 0.005 in] range</td>
<td>IPC-4562</td>
<td>/5(3,4.5) or /6(3,4.5)</td>
<td>⅞ Hard or ⅜ Hard</td>
<td>Recommended for forming</td>
<td>Preferred foil for unsupported fingers</td>
</tr>
<tr>
<td></td>
<td>ASTM-B-152(6)</td>
<td>H01 or H02</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>IPC-4562</td>
<td>/7(4,5)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>ASTM-B-152(6)</td>
<td>O60</td>
<td></td>
<td>Soft (Annealed)</td>
<td>Not recommended</td>
</tr>
</tbody>
</table>

Note 1: The Unified Numbering System (UNS) is the accepted alloy designation system in North America for wrought and cast copper and copper alloy products. The three-digit system developed by the U.S. copper and brass industry was expanded to five digits following the prefix letter C and made part of the Unified Numbering System for Metals and Alloys. UNS designations are simply expansions of the former designations. For example, Copper Alloy No. C110 (Electrolytic Tough Pitch - ETP) or C172 (Copper-Beryllium Alloy) in the original three-digit system respectively became C11000 and C17200 in the UNS System. The UNS is managed jointly by the American Society for Testing and Materials (ASTM) and the Society of Automotive Engineers (SAE). Because these old numbers are embedded in the new UNS numbers, no confusion need result. The designation system is an orderly method of defining and identifying coppers and copper alloys; it is not a specification. It eliminates the limitations and conflicts of alloy designations previously used and at the same time provides a workable method for the identification marking of mill and foundry products.

Note 2: Electrodeposited.

Note 3: Copper foil annealing at 170 °C is complete after 30 minutes. Typical laminations exceed 170 °C for at least 1 hour. Once annealed, the properties of the foil do not change significantly – subsequent laminations (even at substantially higher temperatures) have little or no effect. This claim is based on works published in white paper “Conversion of as Rolled Foil (W5) to Annealed Foil (W7) During Lamination” by Arvind Partha (Olin / Somers) and Richard Zwierlein (DuPont). This paper was presented in 1999 at the IPC Fifth Annual National Conference on Flexible Circuits.

Note 4: IPC-4562 slash sheets do not provide temper data above the thickness of 2 Oz. Therefore, temper requirements are AABUS.

Note 5: Thicker foils require thicker adhesive which will add to additional squeeze out.

Note 6: Although materials procured to ASTM specifications will meet physical and chemical properties, other considerations may be required for use in printed board applications. Designers should keep in mind the following additional items: Surface condition (e.g. mill oils, pits, dents, scratches, flatness, edge quality) and that generally these materials are not produced with any surface treatments.
9.1.10.1 Processes

There are two basic processes used to create these unsupported fingers: build-up using substrate/foil/coverlay materials, and laser ablation.

The build-up process involves pre-punching an access window into the base dielectric, laminating copper foil to the base dielectric, masking the copper in the pre-punched dielectric window before etch to prevent back side etch of the copper, etching the circuitry, and then aligning and laminating a pre-punched coverlay to the etched circuitry base. This will result in some adhesive squeeze out where fingers exit the insulation as shown in Figure 9-6.

![Figure 9-6 Adhesive Squeeze Out Where Fingers Exit Insulation](image)

Note 1: Adhesive squeeze-out on fingers.

The laser ablation process involves ablating the top and bottom dielectrics after lamination and before final “blanking” to create the window as shown in Figure 9-7 (this eliminates the adhesive squeeze out that is created during the build process).

![Figure 9-7 Laser Ablated Window](image)

Regardless of fabrication process, these unsupported conductors are prone to handling damage during final processing (e.g. electrical test and profiling/blanking) as shown in Figure 9-8 and will require special packaging and handling to ensure a usable product, both before and after final assembly. The preferred way to minimize this handling damage is to connect all the fingers with a buss bar of foil or insulator that holds all fingers in alignment as shown in Figure 9-9 and Figure 9-10. The copper buss bar is removed after assembly. This removal can be aided with notched fingers or scoring of a V-groove (see IPC-2222) along the ends of the fingers/conductors assembly. If a dielectric buss bar is utilized, it may remain as shown in Figure 9-11.

![Figure 9-8 Broken Fingers Resulting from Poor Handling](image)
Another way to minimize the risk of handling damage when using the build-up process is to stagger the edges of the base and coverlay (preferred) as shown in Figure 9-12. This breaks up the sharp stress riser on the conductor caused by solder and the aligned covers. A good dimension to use for the offset is a minimum of five times the copper thickness. A larger offset can be used based on available working length. When soldering to a printed board it is recommended that the side with the shorter exposed length is against the board to avoid issues with cleaning and inspections.
This offset opening technique may not be possible with laser processed openings. In addition, this cover offset is not necessary with thicker or more rigid finger designs such as those found in sculptured flex. There is no need to use the offset edge at the busbar end of the opening.

![Figure 9-12 Staggered Base/Coverlay Edges Used to Reduce Stress Points](image)

**Notes:**
- **Note 1:** Concentrated stress location.
- **Note 2:** Minimum 5X copper thickness.
- **Note 3:** Acceptable.
- **Note 4:** Preferred.

### 9.1.10.2 Design Options

There are several other design possibilities that can be used when creating unsupported fingers. They each offer different advantages and design constraints:

a. **Sculptured:** Similar to punched or laser-ablated cover/base windows with the exception that a thicker copper is used (typically 0.254 mm [0.010 in]) and all areas other than the fingers are pre-etched to reduce the copper thickness to aid in flexibility as shown in Figure 9-13. Final product will have 0.20 – 0.254 mm [0.008 - 0.010 in] thick fingers but in the remaining portion of the circuit, the copper will be approximately 60-70% thinner. Fingers will be more robust due to their thickness, but will still be very fragile and prone to damage if not properly packaged or shipped.

![Figure 9-13 Sculptured Flex Showing the Foil Thickness Transition](image)

b. **Brazed/Welded/Soldered Fingers or Pins:** Fingers or pins are manufactured separately from the circuit fabrication, and are either brazed, high-temp soldered or parallel gap welded to circuit lands that run to the edge of the circuit as shown in Figure 9-14. The unsupported conductors with this variation are typically more robust than standard unsupported fingers but are also more expensive due to the added time and labor involved with attaching the fingers or pins. Frequently, these designs are used where the final flex connection is with a method other than soldering, and a wide variety of materials, finishes, and shapes may be used for the fingers to suit a particular application as
described in Table 9-4. Finger connections to the flex may sometimes be covered with a patch or a curable insulating material to offer mechanical strength or isolation. These processes will require internal development to assure that all materials, processes, and final uses are compatible.

![Figure 9-14 Fingers Brazed to Circuit Lands](image)

**Note 1:** Finger.  
**Note 2:** Brazing material.  
**Note 3:** Flex coverlay/adhesive.  
**Note 4:** Base foil.  
**Note 5:** Flex Substrate.

<table>
<thead>
<tr>
<th>Table 9-4 Common Brazed Finger Options</th>
</tr>
</thead>
<tbody>
<tr>
<td>Finger materials</td>
</tr>
<tr>
<td>Finger Shapes</td>
</tr>
<tr>
<td>Finger Finishes</td>
</tr>
</tbody>
</table>

9.1.10.3 Mounting Flex to a Printed Board  
Due to the fragile nature of unsupported fingers, it is recommended that the connection area be ruggedized after being soldered to the printed board to ensure that any mechanical stresses on the flex circuit are not transferred to the unsupported fingers. Connections may be ruggedized in a variety of ways, but the most common methods are bonding the circuit to the printed board with a thermal setting or pressure sensitive adhesive, overcoating the entire connection area with a semi-rigid epoxy after soldering, or to clamp the termination area as shown in Figure 9-15.

![Figure 9-15 Bonding of Unsupported Fingers to Printed Board with Reinforcement](image)

9.2 Holes  
Alternative hole formation in flexible printed boards may use die punched, laser cut, cluster punched, plasma etched, and chemical etched holes. Designers should consult fabricators for the best process for application and hole size.

9.2.1 Unsupported (Non-plated) Component Holes  
When a single-sided flex circuit is used for through-hole mounted component leads, the hole size is typically 0.15 to 0.35 mm [0.006 to 0.014 in] larger than the component lead diameter.
9.2.2 Plated Component Holes When PTHs are used for component leads, the finished hole size is typically 0.25 to 0.5 mm [0.001 to 0.020 in] larger than the component lead diameter.

9.2.2.1 Etchback For Type 3 or Type 4 flexible printed boards, etchback, if required, shall be specified on the master drawing.

9.2.3 Copper Filled Vias Copper filled microvias and vias should be avoided in areas where surface planarity is critical. Surface topography and dimensional stability associated with flexible product could make surface planarization difficult. Designers should consult fabricators regarding the planned construction to ensure that fabrication capability exists. Refer to IPC-2221 and IPC-2226 for additional microvia information.

9.2.3.1 Microvia Stacking For guidance on microvia stacking, refer to IPC-2226.

9.2.4 Back-drilled Holes Back drilling of holes is a method of reducing the overall length of any plated hole by drilling out a portion of the plated hole from either side to a predetermined depth, for purposes of signal integrity or circuit isolation. Recommendations for back-drilled holes include but are not limited to the following:

- Stub length and tolerance from the specified layer should be identified on the drawing preferably with an exploded view (e.g., 0.203 +/- 0.127 mm [0.008 +/-0.005 in] from layer 4). Drill files should be included for each desired location.
- Back-drill diameters should be no less than 0.254 mm [0.010 in] larger than the drilled hole size.
- Conductors should be routed to provide for minimum lateral spacing plus standard fabrication allowance between conductor and subsequent back-drill, unless cap plated. Reference 10.1.1 of IPC-2222.
- Back-drill to back-drill hole wall remaining web spacing below 0.50 mm [0.020 in] may impact structural integrity.
- Allowance for land size reduction to be back-drilled should be accommodated within the design or drawing 0.15 mm [0.006 in] smaller than back-drill diameter. Lands not intended to be back-drilled should follow normal design rules.
- When designating the specified layer, consider the position of the press-fit contact point relative to that layer and use an applicable safety margin.

Note: Unused lands may be removed from back-drill spans.

9.3 Coverlay Access Openings

9.3.1 Coverlay Access, Unsupported Lands Access openings in coverlays to expose unsupported (non-PTH) lands should be at least 0.25 mm [0.001 in] smaller than the land to assure "pad capture" by the coverlay. It is recommended to capture the land on at least two locations (see Figure 9-16). The land size shall be sufficiently larger than the component to allow for the solder fillet and pad capture. See IPC-J-STD-001 for examples of dimensional and solder fillet requirements.
9.3.2 Coverlay Access, Holes Coverlay access holes should be sized to the annular ring requirements of IPC-2221. It is recommended that openings for unsupported hole access in coverlayers for exposing lands should be at least 0.25 mm [0.001 in] larger than the land to allow for registration tolerances and adhesive squeeze-out (see Figure 9-17 and Figure 9-18). When total pad capture is required the access hole shall provide clearance for adhesive squeeze-out and annular ring. Supported PTHs do not require a coverlay pad capture.

9.3.3 Coverlay Access Spacing The limitation for the web left between two adjacent access openings is 0.25 mm [0.001 in] for both die cut and drilled coverlays. Alternative processes are required for spacings less than 0.25 mm [0.001 in] (i.e., windows covering multiple features, laser skiving, and photoimageable coatings).

9.3.4 Land Access/Exposed Lands Access to lands may be accomplished by windows (cutouts) (see Figure 9-16) or access holes in the coverlay. Windows may be used, and are preferred, if they expose only the lands, because conductor paths exiting from under the coverlay into a window area tend to break at the edge of the coverlay. Bending or forming should be avoided at the point of termination of the window. A stiffener is recommended for placement at the opposite side of the window opening to avoid damage to the conductors.
9.3.5 Type 1 Land Access, Opposite Sides Access openings to lands may be on one or both sides of a single printed board layer. Access openings on both sides of a single layer are more costly, due to the additional tooling and processing required.

10 CONDUCTORS

10.1 Conductor Characteristics Conductor characteristics shall be in accordance with IPC-2221 and as stated in 10.1.1 and 10.1.2.

10.1.1 Conductor Routing Conductors should not exit from a rigid to flexible section at an angle. Conductors should not pass through a designed bend zone at an angle and should run perpendicular to the bend. Minimum spacing of an angular conductor to a bend or rigid-flex transition should be no less than 5.0 mm [0.197 in] (see Figure 10-1).

Note: It is preferred that all conductor routing in flex areas should have a radius to reduce stress conditions.
10.1.2 Conductor Edge Spacing Except for edge-board contacts, the minimum distance between conductive surfaces, PTH and the edge of the finished printed board, unsupported through holes or cutouts in the rigid section shall not be less than the minimum spacing specified in Table 6-1 of IPC-2221 plus 0.5 mm [0.020 in] from the edge of the finished printed board. In the flexible section, to promote adequate edge seal of coverlay and base material, the finished product value should not be less than 0.4 mm [0.016 in] for conductive surfaces to an edge. Outline tooling tolerances should be considered for the designed distance. Ground and heat sink planes may extend to the edge when required by design. Special design applications in areas such as high voltage, surface mount, and radio frequency (RF) technology may require variances to these requirements. For tighter edge distance, alternative tooling can be used.

10.2 Land Characteristics Land characteristics shall be in accordance with IPC-2221.

10.3 Large Conductive Areas Unless otherwise specified, large conductive areas on external or internal layers shall be in accordance with 5.2.6 and 5.2.7.

11 DOCUMENTATION

Documentation shall be in accordance with IPC-2221.

12 QUALITY ASSURANCE

Quality assurance and conformance coupon design and location shall be in accordance with IPC-2221.
Appendix A
Design Tutorial

A.1 Flexing and Forming The successful design of a flexible circuit requires that the designer carefully consider both the electrical and mechanical requirements of the finished product at the design stage. Unlike rigid printed boards, flexible circuits are required to bend and flex to fit the specific application, making them as much a mechanical device as an electrical device. Most failures that occur in finished flexible circuitry are due to these bending and flexing operations. Also, many circuit features that enhance the electrical performance of a flex circuit (such as controlled impedance, shields, etc.) will decrease the circuit's mechanical performance, and vice versa. For this reason, it is imperative that the designer understand the forces and material interactions that are occurring in the flex circuit when it is bent or flexed.

Whenever a flex circuit is bent or flexed, the circuit material in and adjacent to the bending area will experience a variety of stresses. The materials on the inside of the bend will be subjected to compression forces, and the materials on the outside of the bend will be subjected to stretching or tension forces. The greater the thickness of the flex circuit, the greater these forces will become. If any of these forces exceed the limits of the flex materials, the result will be wrinkles, delamination, and/or cracks and tears. Somewhere inside the circuit, there will be a planar region referred to as the neutral bend axis that does not experience any tension or compression forces. Any material in a bending zone that falls to the inside of the neutral bend axis will experience compression forces, and any material that falls to the outside of the neutral bend axis will experience tension forces (see IPC-2223 Figure 5-10). The further a material is from the neutral bend axis, the more severe these forces become. In most applications, the ideal location for the neutral bend axis is at or near the center of the circuit. This will spread the forces generated by the bending operation equally between tension and compression. Many factors can affect the location of the neutral bend axis. Copper greater than 2 oz. or dielectric film greater than 0.0762 mm [0.003 in] will tend to shift the neutral bend axis in the direction of those materials. As an example, consider a 4 layer flex circuit that has controlled impedance between layers one and two requiring a 0.127 mm [0.005 in] polyimide core between those layers. If this circuit is formed to a 90° bend, with layer 1 on the inside of the form, the result would be that the neutral bend axis would shift toward layer one, and the outside of the bend would experience exaggerated tension forces (possible cover tears and/or conductor cracks). If this same circuit was formed to a 90° bend with layer four on the inside of the bend, the neutral bend axis would again shift toward layer one, and the inside of the bend would experience exaggerated compression forces.
(possible wrinkles and/or delamination). As the previous example shows, it is extremely important to be aware of the approximate location of the neutral bend axis, and how its location can affect the performance of the flex circuit.

A.2 Material Selection Guidelines

- **Copper Weight:** Copper weight should be selected based on electrical current requirements and also overall circuit thickness/flexibility requirements. Higher electrical current will require wider or thicker conductors. Be aware that thicker copper also requires thicker adhesive (approximately 0.0254 mm [0.001 in] of adhesive per 1 oz. of copper weight), which drives up the overall circuit thickness.

- **Copper Type:** Rolled and annealed copper will typically perform better in a flexing area than electro-deposited copper.

**Caution Note:** Due to lower mechanical properties, ½ oz. copper may fail prematurely relative to 1 oz. in bending areas.

- **Base Dielectric Type/Thickness:** The base dielectric should be selected based on overall circuit thickness requirements, electrical requirements such as impedance, and thermal requirements (for rigid-flex and high layer count multilayer flex). Higher impedance requirements drive thicker base dielectrics, and high layer count designs should employ adhesiveless base dielectrics. See IPC-2223 4.2.4.1.

- **Cover Thickness:** Covers for flexible printed boards are typically a combination of dielectric film and b-stage epoxy or acrylic adhesive film. Cover dielectric thickness requirements are driven by mechanical concerns such as abrasion, with (0.0254 mm [0.001 in] dielectric thickness being the most common choice, though greater thicknesses are available and can be used). Cover adhesive thickness is driven by copper weight of the conductors. It is important to use sufficient adhesive to completely encapsulate the conductors.

**Caution Note:** The printed board designer should make every effort to reduce or eliminate low Tg materials (such as acrylic or epoxy thermo-setting film adhesives) in PTH areas of Type 4, Class 3 circuits. This also holds true for any Type 4 circuit, regardless of performance Class, which will be subjected to temperature cycling or lead-free reflow processing due to concerns of excessive Z-axis expansion which affects PTH reliability.

- **Reference Plane Materials:** Reference planes can be formed using copper, conductive inks, or shielding films. Using conductive inks or shielding films will generally yield a more flexible finished product than can be attained when using copper. In applications where memory is desired for forming, copper may be a better choice.

A.3 Stress Concentration Features (Stress Risers) Another critical factor affecting the reliability of a formed flex circuit is stress concentrating features. Flex circuit construction should remain constant with no changes of any kind in, or in close proximity to, a bend area. Examples of stress concentrating features in a bend area would be attributes such as conductor width or directional changes, cover access openings, vias or holes of any type, circuit profile changes, and terminations of platings or coatings. Even small changes in circuit features in a bend area that may seem inconsequential can lead to catastrophic failure of the flex circuit if it is formed sharply in that area. Virtually all failures that occur in flex circuitry are a result of mechanical forces that can be traced to some type of stress concentrating feature. The importance of thoroughly reviewing the circuit construction in and around a bending or forming zone cannot be overstressed to ensure a reliable design.

A.3.1 Dual Row Zero Insertion Force (ZIF) Connectors

ZIF (Zero Insertion Force) connectors are a commonly used connection method for flexible circuitry. There are two main styles—single row contact and dual row contact as shown in Figure A-1 and Figure A-2.
Single row contact ZIF connectors typically specify FPC contact finger width of 0.35 mm [0.014 in] and overall thickness of 0.3 +/- 0.05 mm [0.012 +/- 0.002 in]. Dual row contact ZIF connectors typically specify flexible printed board contact area with trace widths as low as 0.10 mm [0.004 in] and overall thickness of 0.2 +/- 0.05 mm [0.008 +/- 0.002 in]. Contact areas of both ZIF styles are typically Ni/Au plated.

Designers should use caution when specifying dual row style ZIF connectors. The combination of reduced overall thickness, very fine conductors and nickel underplating in the connection area make flexible circuits utilizing this style of connector highly prone to damage during installation.

Single row contact ZIF connectors have not demonstrated this predisposition for damage during assembly due mainly to the greater overall thickness in the connection area and the greater width of the copper conductors in those areas.

Industry history with the dual row ZIF style connector has shown that it is very easy to bend the connection area of the flexible circuit during the installation process. The nickel plating on the conductor surfaces in the connection area is very brittle and prone to cracking if bent or flexed. Since many of the conductors in the connection area are naturally fragile due to the fine width (0.10 mm [0.004 in]), adding nickel plating exacerbates the problem. Surface cracks can form in the nickel layer after a single mild bend (under 45 °). These cracks can propagate over time through the copper layer resulting in open traces. A bend of 90 ° or more will nearly always result in cracks through the nickel, copper, or both.

Empirical evidence has shown that 1 oz. copper is more robust than thinner copper foils, so it is recommended that 1 oz. copper be utilized whenever possible on dual row ZIF style connector. This will not eliminate the issue but may reduce the frequency of occurrence.

A.4 Overall Size & Shape Flex circuits are fabricated in panel or roll form from flexible copper-clad materials that are laminated together, similar to a rigid board. These flexible layers are different from rigid layers, however, because they are not glass reinforced and have less predictable registration during fabrication. The clad materials are also more expensive, and there are more processing steps and more labor involved. As a result, the cost per square inch for a flex board will generally be higher than a rigid board of the same size. For this reason it is important to keep the flex circuit profile as compact as possible. For panel processing, panel sizes vary between suppliers, but most prefer to keep panel sizes to 457 x 610 mm [18.0 x 24.0 in] or less (approximately 406 x 559 mm [16.0 X 22.0 in] useable area). These limitations are driven by material availability as well as processing equipment. It is recommended to work with the flex circuit supplier during the
design stage to ensure that the circuit size will match the supplier's panel size and processing equipment so as to not incur significant cost penalties.

A.5 Allowances Rigid-flex and multilayer flex designs require more annular ring allowance than rigid board designs. Rigid boards are comprised of only glass reinforced rigid layer cores, with better dimensional stability and layer-to-layer registration during processing. Rigid-flex and multilayer flex, on the other hand, are comprised of polyimide film cores and coverlayers that have no glass reinforcement and are prone to misregistration during processing. It is also a common error in mechanical design to treat the flex like a sheet metal part when allocating space for it. The standard manufacturing tolerance for a flex profile is +/- 0.51 mm [0.020 in], so when solid modeling a flex it is a good idea to leave at least a 0.762 mm [0.030 in] gap between the flex outline and the adjacent housings and mechanical parts. A flex profile feature should never be used as a mechanical reference point for other components or structures.

A.6 Symmetry Avoid designing symmetrical flex outlines or pinfield land patterns to prevent incorrect installation of flex or components. This applies to mounting hole locations as well. It is recommended to always use one offset hole when working with circular pin patterns or bolt hole patterns.

A.7 Length A limited number of suppliers can produce a circuit with lengths over 559 mm [22.0 in]. The layer count has a greater impact on producibility at these lengths due to the fact that flexible base materials can have as much as 0.0254 mm [0.001 in] per 2.54 mm [1.0 in] of growth or shrinkage during processing and lamination. If annular ring is tight, misregistration can become the limiting factor on yield and cause much higher cost penalties than the larger material usage. For this reason, it is best to limit layer count on large panel sizes and maximize annular ring on the design. It is also highly recommended to maintain a narrow overall part profile to provide better material usage. Panel sizes larger than 457 x 610 mm [18.0 x 24.0 in] may drive processing equipment restrictions, electrical testing challenges, and reduced yield, driving costs higher as well.

A.8 Rigid Areas (Types 1, 2 and 3) Circuit areas that require additional structural support such as connector areas and regions with SMT components should have mechanical stiffeners applied. The most common materials used to create mechanical stiffeners are glass reinforced FR4 and polyimide film. Stiffeners are typically bonded to the circuit using thermal setting acrylic or epoxy adhesives, or with pressure sensitive adhesive (PSA). Thermal setting adhesives will result in a stronger bond between the circuit and stiffener than PSA, but is also more costly. It is important to remember that these stiffeners are for mechanical support only. Complete, void free lamination is not required to achieve the desired stiffening function.

A.8.1 Rigid Areas (Type 4) It is preferred for material balancing that rigid areas on each side of the board be the same thickness. It is HIGHLY recommended that all rigid areas on a given side be the same thickness. PTHs in rigid areas should be located far enough from the rigid-flex interface so that the plated barrel of the hole is at least 3.18 mm [0.125 in] plus one half of the PTH diameter from the rigid edge. It is also highly recommended that the designer reduce or eliminate flexible adhesive systems in rigid areas due to the thermal expansion characteristics of these adhesive systems. Differential rigid thicknesses in Type 4 circuits will significantly increase costs. However, selective removal of conductive layers without removal of the carrier substrate may not have cost impact.

A.9 Flexible Areas Flexible layers between rigid areas of a Type 4 flex are generally comprised of double-clad core material with coverlays. The most typical construction is the loose-leaf (unbonded) construction where these two-layer inner layers are not bonded together. There is a practical limit to how short the flexible area can be on a rigid-flex. The supplier will have a difficult time producing a flexible area less than 8.89 mm [0.35 in] (typically) between rigid interfaces. This will only work with a single two-layer inner layer. As a general rule, the minimum typical recommended span is 12.7 mm [0.5 in], and for each additional two-layer unbonded inner layer, 12.7 mm [0.5 in] typically should be added if the bend in this area will be 90°. Shorter lengths and thicker layer constructions will cause the flex to become very stiff and difficult to bend, and will impart very high stresses on the laminates. The designer should plan for this when modeling the mechanical layout and leave room for as much flexible length between rigid areas as possible to avoid problems.

A.10 Banding A technique called banding, or staggered bands, can be used to accommodate a short flexible area if there is no room for a service loop. Instead of using the full width of the flexible region between rigid areas for each inner layer of a Type 4 flex, the area can be divided into smaller bands of equal width for each inner layer, with an approximately 0.381 mm
[0.015 in] gap between bands. This eliminates buckling and stress in the flexible area and allows the flex to be bent at a radius as small as 6 times the thickness of inner layer, and allows flex lengths down to the practical minimum. See IPC-2223 5.2.5.3 and IPC-2223 Figure 5-16.

A.11 Symmetrical Lay Ups Due to the wide variety of materials used in rigid-flex construction and the corresponding wide range of expansion coefficients, rigid-flex material stackups should be kept as symmetrical as possible. If the lay-up is not symmetrical, the varying expansion properties of the different material can cause the finished board to bow, or twist.

A.12 Reference Literature Additional flexible circuit design information can be freely obtained from the following sources:


2. “Ask the Flexperts” Technical Column – Printed Circuit Design and Fab magazine, Mark Finstad, Mark Verbrugge – www.pcdandf.com