



IPC-6012E

2020 - March

**Qualification and Performance
Specification for Rigid
Printed Boards**

Supersedes IPC-6012D

September 2015

An international standard developed by IPC

Association Connecting Electronics Industries





IPC-6012E

Qualification and Performance Specification for Rigid Printed Boards

Developed by the Rigid Printed Board Performance Specifications Task Group (D-33a) of the Rigid Printed Board Committee (D-30) of IPC

Supersedes:

IPC-6012D - September 2015
IPC-6012C - April 2010
IPC-6012B with
Amendment 1 - July 2007
IPC-6012B - August 2004
IPC-6012A with
Amendment 1 - July 2000
IPC-6012A - October 1999
IPC-6012 - July 1996
IPC-RB-276 - March 1992

Users of this publication are encouraged to participate in the development of future revisions.

Contact:

IPC

Table of Contents

1	SCOPE	1	3.2.8	Polymer Coating (Solder Mask)	13
1.1	Statement of Scope	1	3.2.9	Fusing Fluids and Fluxes	13
1.2	Purpose	1	3.2.10	Marking Inks	13
1.2.1	Supporting Documentation	1	3.2.11	Hole Fill Insulation Material	13
1.3	Performance Classification and Type	1	3.2.12	Heatsink Planes, External	13
1.3.1	Classification	1	3.2.13	Via Protection	14
1.3.2	Printed Board Type	1	3.2.14	Embedded Passive Materials	14
1.3.3	Selection for Procurement	1	3.3	Visual Examination	14
1.3.4	Material, Plating Process and Surface Finish	3	3.3.1	Edges	14
1.4	Terms and Definitions	4	3.3.2	Laminate Imperfections	14
1.4.1	Back-Drilling	4	3.3.3	Plating and Coating Voids in the Hole	15
1.4.2	Stub (Plated Hole)	4	3.3.4	Lifted Lands	15
1.4.3	High Density Interconnects (HDI)	4	3.3.5	Marking	15
1.4.4	Microvia	5	3.3.6	Solderability	16
1.4.5	Design Data	5	3.3.7	Plating Adhesion	16
1.5	Interpretation	5	3.3.8	Edge Printed Board Contact, Junction of Gold Plate to Solder Finish	16
1.6	Presentation	5	3.3.9	Back-Drilled Holes	17
1.7	Design Data Protection	5	3.3.10	Workmanship	17
1.8	Revision Level Changes	5	3.4	Printed Board Dimensional Requirements	17
2	APPLICABLE DOCUMENTS	6	3.4.1	Hole Size, Hole Pattern Accuracy and Pattern Feature Accuracy	17
2.1	IPC	6	3.4.2	Annular Ring and Breakout (External)	17
2.2	Joint Industry Standards	8	3.4.3	Bow and Twist	19
2.3	Federal	8	3.5	Conductor Definition	19
2.4	Other Publications	8	3.5.1	Conductor Width and Thickness	20
2.4.1	American Society for Testing and Materials	8	3.5.2	Conductor Spacing	20
2.4.2	Underwriters Lab	8	3.5.3	Conductor Imperfections	20
2.4.3	National Electrical Manufacturers Association	8	3.5.4	Conductive Surfaces	20
2.4.4	American Society for Quality	8	3.6	Structural Integrity	22
2.4.5	AMS	8	3.6.1	Thermal Stress Testing	23
2.4.6	American Society of Mechanical Engineers	8	3.6.2	Requirements for Microsectioned Coupons or Printed Boards	24
3	REQUIREMENTS	9	3.7	Solder Mask Requirements	39
3.1	General	9	3.7.1	Solder Mask Coverage	39
3.2	Materials	9	3.7.2	Solder Mask Cure and Adhesion	39
3.2.1	Laminates and Bonding Material	9	3.7.3	Solder Mask Thickness	40
3.2.2	External Bonding Materials	9	3.8	Electrical Requirements	40
3.2.3	Other Dielectric Materials	9	3.8.1	Dielectric Withstanding Voltage	40
3.2.4	Metal Foils	9	3.8.2	Electrical Continuity and Isolation Resistance	40
3.2.5	Metal Planes/Cores	9	3.8.3	Circuit/Plated Hole Shorts to Metal Substrate	40
3.2.6	Base Metallic Plating Depositions and Conductive Coatings	9	3.8.4	Moisture and Insulation Resistance (MIR)	40
3.2.7	Surface Finish Depositions and Coatings – Metallic and Non-Metallic	10			

3.9 Cleanliness 41

3.9.1 Cleanliness Prior to Solder Mask Application 41

3.9.2 Cleanliness After Solder Mask, Solder, or Alternative Surface Coating Application 41

3.9.3 Cleanliness of Inner Layers After Oxide Treatment Prior to Lamination 41

3.10 Special Requirements 41

3.10.1 Outgassing 41

3.10.2 Fungus Resistance 41

3.10.3 Vibration 41

3.10.4 Mechanical Shock 41

3.10.5 Impedance Testing 41

3.10.6 Coefficient of Thermal Expansion (CTE) 42

3.10.7 Thermal Shock 42

3.10.8 Surface Insulation Resistance (As Received) 42

3.10.9 Metal Core (Horizontal Microsection) 42

3.10.10 Rework Simulation 42

3.10.11 Bond Strength, Unsupported Component Hole Land 42

3.10.12 Destructive Physical Analysis 42

3.10.13 Peel Strength Requirements (For Foil Laminated Construction Only) 42

3.10.14 Design Data Protection 42

3.10.15 Performance Based Testing for Microvia Structures – Structural Integrity During Thermal Stress 43

3.11 Repair 43

3.11.1 Circuit Repairs 43

3.12 Rework 43

4 QUALITY ASSURANCE PROVISIONS 43

4.1 General 43

4.1.1 Qualification 43

4.1.2 Sample Test Coupons 43

4.2 Acceptance Tests 44

4.2.1 C=0 Zero Acceptance Number Sampling Plan 44

4.2.2 Referee Tests 44

4.3 Quality Conformance Testing 44

4.3.1 Coupon Selection 45

5 NOTES 50

5.1 Ordering Data 50

5.2 Superseded Specifications 50

APPENDIX A 51

Figures

Figure 1-1 Example of a Back-Drilled Hole (Not To Scale) 4

Figure 1-2 Example of a Shallow Back-Drill 4

Figure 1-3 Microvia Definition 5

Figure 3-1 Annular Ring Measurement (External) 19

Figure 3-2 Breakout of 90° and 180° 19

Figure 3-3 External Conductor Width Reduction 19

Figure 3-4 Example of Intermediate Target Land in a Microvia 19

Figure 3-5 Rectangular Surface Mount Lands 20

Figure 3-6 Round Surface Mount Lands 21

Figure 3-7 Printed Board Edge Connector Lands 21

Figure 3-8 Plated Hole Microsection (Grinding/Polishing) Tolerance 23

Figure 3-9 An Example of Plating to Target Land Separation 23

Figure 3-10 Copper Crack Definition 26

Figure 3-11 Separations at External Foil 26

Figure 3-12 Plating Folds/Inclusions – Minimum Measurement Points 26

Figure 3-13 Microsection Evaluation Laminate Attributes 27

Figure 3-14 Measurement for Etchback 27

Figure 3-15 Measurement for Dielectric Removal 28

Figure 3-16 Measurement for Negative Etchback 28

Figure 3-17 Annular Ring Measurement (Internal) 29

Figure 3-18 Microsection Rotations for Breakout Detection 29

Figure 3-19 Comparison of Microsection Rotations 29

Figure 3-20 Example of Non-Conforming Dielectric Spacing Reduction Due to Breakout at Microvia Target Land 30

Figure 3-21 Surface Copper Wrap Measurement for Filled Holes (Over Foil) 30

Figure 3-22 Surface Copper Wrap Measurement for Filled Holes (Over Laminate) 31

Figure 3-23 Surface Copper Wrap Measurement for Non-Filled Holes 31

Figure 3-24 Wrap Copper (Acceptable) 31

Figure 3-25 Wrap Copper Removed by Excessive Processing, e.g., Sanding/Planarization/Etching (Not Acceptable) 32

Figure 3-26 Copper Cap Thickness 33

Figure 3-27 Copper Cap Filled Via Height (Bump) 33

Figure 3-28 Copper Cap Depression (Dimple) 33

Figure 3-29 Copper Cap Plating Voids 33

Figure 3-30 Nonconforming Via Fill Between Copper Cap Plating Layers 33

Figure 3-31 Acceptable Via Fill Between Copper Cap Plating Layers 33

Figure 3-32 Example of Acceptable Voiding in a Cap Plated, Copper Filled Via 34

Figure 3-33 Example of Acceptable Voiding in a Copper Filled Microvia without Cap Plating 34

Figure 3-34 Example of Nonconforming Void in a Cap Plated, Copper Filled Microvia 34

Figure 3-35 Example of Nonconforming Void in a Copper Filled Microvia 34

Figure 3-36 Microvia Contact Dimension 35

Figure 3-37 Exclusion of Separations in Microvia Target Land Contact Dimension 35

Figure 3-38 Unintended Piercing of Microvia Target Land (Laser Drilled) 35

Figure 3-39 Intentional Piercing of Microvia Target Land (Mechanically Drilled²) 35

Figure 3-40 Overhang 37

Figure 3-41 Metal Core to PTH Spacing 37

Figure 3-42 Measurement of Minimum Dielectric Spacing 38

Figure 3-43 Fill Material in Blind/Through Vias When Cap Plating Not Specified 38

Figure 3-44 Void in Fill Material at Hole Wall Interface 38

Tables

Table 1-1 Technology Adders 2

Table 1-2 Default Requirements 3

Table 3-1 Metal Planes/Cores 9

Table 3-2 Maximum Limits of SnPb Solder Bath Contaminant 10

Table 3-3 Surface Finish and Coating Requirements 12

Table 3-4 Surface and Hole Copper Plating Minimum Requirements for Buried Vias > 2 Layers, Through-Holes, and Blind Vias 13

Table 3-5 Surface and Hole Copper Plating Minimum Requirements for Microvias (Blind and Buried) 13

Table 3-6 Surface and Hole Copper Plating Minimum Requirements for Buried Cores (2 layers) 13

Table 3-7 Plating and Coating Voids in the Hole 15

Table 3-8 Edge Printed Board Contact Gap 17

Table 3-9 Minimum Annular Ring 18

Table 3-10 Plated Hole Integrity After Stress 25

Table 3-11 Cap Plating Requirements for Filled Holes 32

Table 3-12 Microvia Contact Dimension (Laser Drilled) 35

Table 3-13 Microvia Contact Dimension (Mechanically Drilled) 35

Table 3-14 Internal Layer Foil Thickness after Processing 36

Table 3-15 External Conductor Thickness after Plating 37

Table 3-16 Solder Mask Adhesion 40

Table 3-17 Dielectric Withstanding Voltages 40

Table 3-18 Insulation Resistance 40

Table 4-1 Qualification Test Coupons 44

Table 4-2 C=0 Sampling Plan per Lot Size 45

Table 4-3 Acceptance Testing and Frequency 46

Table 4-4 Quality Conformance Testing 50

Qualification and Performance Specification for Rigid Printed Boards

1 SCOPE

1.1 Statement of Scope This specification establishes and defines the qualification and performance requirements for the fabrication of rigid printed boards.

1.2 Purpose The purpose of this specification is to provide requirements for qualification and performance of rigid printed boards based on the following constructions and/or technologies. These requirements apply to the finished product unless otherwise specified:

- Single-sided, double-sided printed boards with or without plated-through holes (PTHs).
- Multilayer printed boards with PTHs with or without buried/blind vias/microvias.
- Active/passive embedded circuitry printed boards with distributive capacitive planes and/or capacitive or resistive components.
- Metal core printed boards with or without an external metal heat frame, which may be active or non-active.

1.2.1 Supporting Documentation IPC-A-600, which contains figures, illustrations and photographs that can aid in the visualization of externally and internally observable acceptable/nonconforming conditions, may be used in conjunction with this specification for a more complete understanding of the recommendations and requirements.

1.3 Performance Classification and Type

1.3.1 Classification This specification establishes acceptance criteria for the performance classification of rigid printed boards based on customer and/or end-use requirements. Printed boards are classified by one of three general Performance Classes as defined in IPC-6011.

1.3.1.1 Requirement Deviations Requirements deviating from these heritage classifications **shall** be as agreed between user and supplier (AABUS).

1.3.1.2 Space Requirement Deviations Space performance classification deviations are provided in the IPC-6012ES Addendum and are applicable when the addendum is specified within the procurement documentation.

1.3.2 Printed Board Type Printed boards without PTHs (Type 1) and with PTHs (Types 2-6) are classified as follows and may include technology adders as described in Table 1-1:

Type 1 — Single-Sided Printed Board

Type 2 — Double-Sided Printed Board

Type 3 — Multilayer Printed Board without blind or buried vias

Type 4 — Multilayer Printed Board with blind and/or buried vias (may include microvias)

Type 5 — Multilayer metal core Printed Board without blind or buried vias

Type 6 — Multilayer metal core Printed Board with blind and/or buried vias (may include microvias)

1.3.3 Selection for Procurement Performance Class **shall** be specified in the procurement documentation.

The procurement documentation **shall** provide sufficient information to fabricate the printed board and ensure that the user receives the desired product. Information that should be included in the procurement documentation is to be in accordance with IPC-2611 and IPC-2614.

The procurement documentation **shall** specify the thermal stress test method to be used to meet the requirement of 3.6.1. Selection **shall** be from those depicted in 3.6.1.1, 3.6.1.2 and 3.6.1.3. If not specified (see 5.1), the default **shall** be per Table 1-2.

When tested in accordance with IPC-TM-650, Method 2.4.28.1, the maximum percentage of cured solder mask lifting from the G coupon or printed board **shall** be in accordance with Table 3-16.

Table 3-16 Solder Mask Adhesion

Surface	Maximum Percentage Loss Allowed		
	Class 1	Class 2	Class 3
Bare Copper	10	5	0
Gold or Nickel	25	10	5
Base Laminate	10	5	0
Melting Metals (Tin-lead plating, fused tin-lead, and bright acid-tin)	50	25	10

3.7.3 Solder Mask Thickness Any requirement for the measurement of solder mask thickness **shall** be AABUS. If a thickness measurement is required, instrumental methods may be used or assessment may be made using a microsection of the parallel conductors on the E coupon or Destructive Physical Analysis (DPA) (see 3.10.12, thermal stress not required). For additional guidance on solder mask thickness measurement options, see the IPC-6012 Automotive Addendum.

3.8 Electrical Requirements When tested as specified in Table 4-3 and Table 4-4, the printed boards **shall** meet the electrical requirements detailed in the following paragraphs.

3.8.1 Dielectric Withstanding Voltage Applicable test coupons or printed boards **shall** meet the requirements of Table 3-17, without flashover, or breakdown between conductors, or conductors and lands. The dielectric withstanding voltage test **shall** be performed in accordance with IPC-TM-650, Method 2.5.7. The dielectric withstanding voltage **shall** be applied between all common portions of each conductor pattern and adjacent common portions of each conductor pattern. The voltage **shall** be applied between conductor patterns of each layer and the electrically isolated pattern of each adjacent layer. For embedded passive device capacitor materials, the dielectric withstanding voltage for adjacent plane layers that are not electrically common **shall** be per IPC-6017.

Table 3-17 Dielectric Withstanding Voltages

	Class 1	Class 2 and Class 3
Voltage for Spacing 80 μm [3,150 μin] or greater	No requirement	(500 +15 -0) V (dc)
Voltage for Spacing less than 80 μm [3,150 μin]	No requirement	(250 +15 -0) V (dc)
Time	No requirement	30 sec +3, -0

3.8.2 Electrical Continuity and Isolation Resistance Finished printed boards **shall** be tested in accordance with IPC-9252. Electrical continuity and isolation resistance testing of blind and buried structures is not required for in process checks.

3.8.3 Circuit/Plated Hole Shorts to Metal Substrate Printed boards **shall** be tested in accordance with 3.8.1 except that polarizing voltage of (500 +15 -0) V (dc) **shall** be applied between conductors and/or lands and the metal substrate (heat sink or core) in a manner such that each conductor/land area is tested (e.g., using a metallic brush or aluminum foil). The printed board **shall** be capable of withstanding (500 +15 -0) V (dc) between circuitry/plated holes and the metal core substrates. There **shall** be no flashover or dielectric breakdown.

3.8.4 Moisture and Insulation Resistance (MIR) Test coupons **shall** be tested in accordance with the procedure outlined below. The test coupon **shall not** exhibit subsurface imperfections in excess of that allowed in 3.3.2. Insulation resistance **shall** meet the minimum requirements shown in Table 3-18 (at (500 +15 -0) V (dc)). Non-component flush printed boards **shall** have a minimum requirement of 50 M Ω for all classes.

Table 3-18 Insulation Resistance

	Class 1	Class 2	Class 3
As received ¹	Maintain electrical function	500 M Ω	500 M Ω
After exposure to moisture	Maintain electrical function	100 M Ω	500 M Ω

Note 1. This measurement is taken after conformal coating material has been applied to the test coupon in accordance with IPC-TM-650, Method 2.6.3.

The moisture and insulation resistance for printed boards **shall** be performed in accordance with IPC-TM-650, Method 2.6.3. Conformal coating in accordance with IPC-CC-830 **shall** be applied to the external conductors prior to chamber exposure. Final measurements **shall** be made at room temperature within two hours after removal from the test chamber. All layers

版权等原因，不能全部发布。

此为样本文件，如需更多内容，完整版：

[单击进入](#)

www.file123.top

1395833280@qq.com

微信：IPCSTD



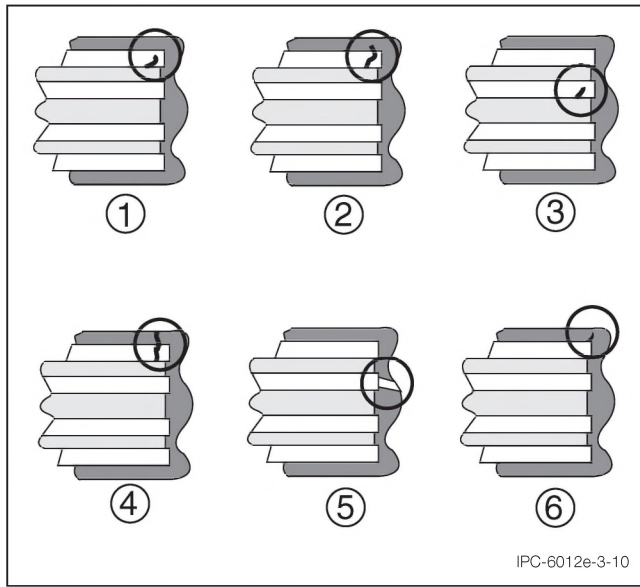


Figure 3-10 Copper Crack Definition⁷

- Note 1.** "A" cracks - crack in external foil.
- Note 2.** "B" cracks - crack does not completely break plating.
- Note 3.** "C" cracks - crack in internal foil.
- Note 4.** "D" cracks - complete fracture.
- Note 5.** "E" cracks - barrel crack in plating only.
- Note 6.** "F" cracks - corner crack in plating only.
- Note 7:** Copper plating as shown may include multiple plating layers or cap plating.

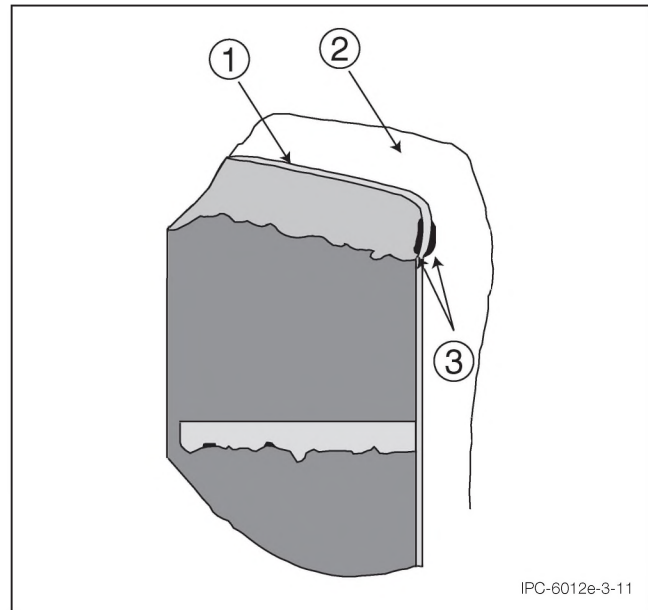


Figure 3-11 Separations at External Foil

- Note 1.** Conductive coatings.
- Note 2.** Electrolytic copper.
- Note 3.** Area of acceptable separation along the vertical edge of the external foil.

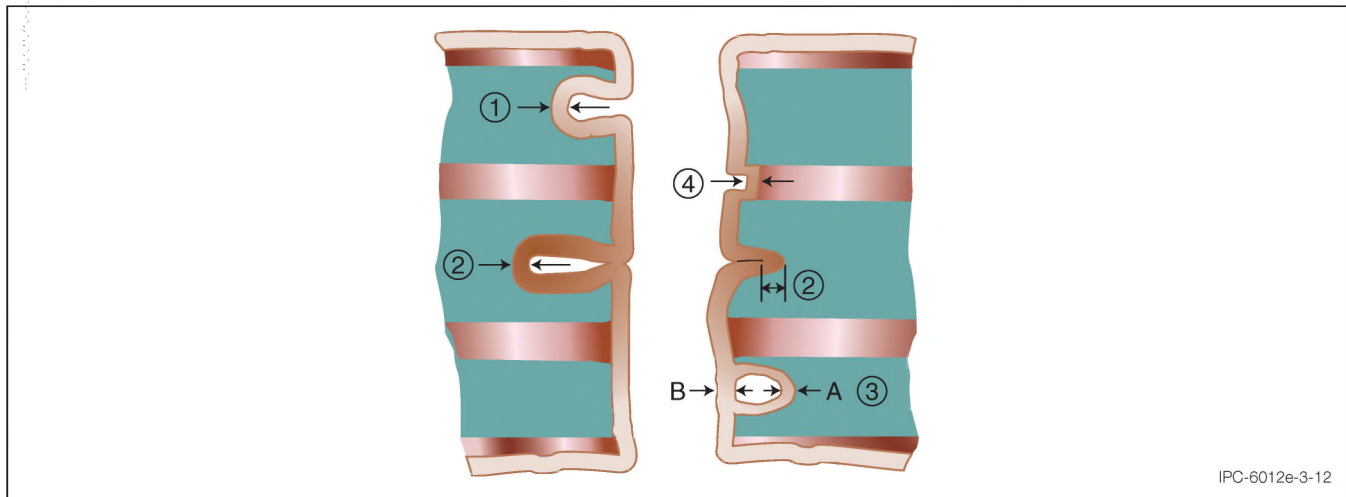


Figure 3-12 Plating Folds/Inclusions – Minimum Measurement Points

- Note 1.** Minimum copper plate measurement point. Plating folds that are not enclosed and where the minimum copper plate thickness in Tables 3-4, 3-5, or 3-6 is met are acceptable.
- Note 2.** Enclosed plating folds (inclusions) with demarcation line visible. Measure and accept per **Note 1**.
- Note 3.** Enclosed plating fold with no visible demarcation line. The thickness measurement A+B shall comply with the minimum copper plate thickness in Tables 3-4, 3-5, or 3-6.
- Note 4.** Minimum copper plate measurement point for negative etchback.

3.6.2.3 Laminate Voids For Class 2 and Class 3 products, there **shall** be no laminate voids outside of any thermal zones (see Figure 3-13) in excess of 80 µm [3,150 µin]. For Class 1 products, voids allowed outside of any thermal zone (see Figure 3-13) **shall not** exceed 150 µm [5,906 µin]. Boundary line voids that overlap into a thermal zone **shall not** be in excess of 80 µm [3,150 µin] for Class 2 or Class 3 products and 150 µm [5,906 µin] for Class 1 products. Multiple voids between two adjacent plated holes in the same plane **shall not** have a combined length which exceeds these limits. Voids between conductive patterns that are not electrically common in either the horizontal or vertical direction **shall not** decrease the minimum dielectric spacing.

Market Your Business



USE IPC'S GLOBAL reach and exclusive member benefits to increase your company's visibility.

- Build your brand by using the IPC member logo to highlight your company's involvement, plus get free links from IPC's Web site to your Web site.
- Be listed in IPC's online membership directory available to all IPC members.
- Put your products, services and qualifications in front of key customers with a listing in IPC's Products and Services Index (PCB and EMS companies only).
- Exhibit in IPC's annual trade show and conference, IPC APEX EXPO, plus the IPC Midwest Conference & Exhibition at a special member price.
- Gain valuable exposure by sponsoring market research conferences and Management Meetings at discounted rates.

Contain Costs

TAKE ADVANTAGE OF exclusive savings opportunities that can easily offset your annual membership investment.

- Get discounts of up to 50 percent on IPC standards, publications and training materials.
- Save money on subscription purchases of IPC standards through IHS. Save up to 25 percent.
- Enjoy dramatic discounts on registration fees for meetings, technical conferences, workshops and tutorials.
- Benefit from preferred pricing on exhibit space at IPC trade shows and events.

Join the Leaders in IPC

WHATEVER SEGMENT OF the industry you are in — PCB design, board manufacturing, electronics manufacturing services (EMS), original equipment manufacturer (OEM), industry supplier, government agency, educational institution — IPC membership has something to offer virtually everyone at your site. Put the resources of the entire industry behind your company by joining IPC today!

For more than 50 years, company leaders have looked to IPC – Association Connecting Electronics Industries® for the tools, information and forums they need to thrive in the ever-changing electronic interconnect industry.

As a member-driven organization and leading source for industry standards, training, market research and public policy advocacy, IPC supports programs to meet the needs of the estimated \$1.5 trillion (U.S. dollars) global electronics industry.

Whatever your role in the industry — a small start-up company, an educational or government institution or a Fortune® 500 firm — you can join more than 2,700 companies worldwide that enjoy access to unparalleled opportunities to participate in and shape the direction of our collective future.

**To learn more about
IPC membership
and apply online,
visit www.ipc.org.**

Benefits of IPC Membership

Put IPC Membership to Work for YOU

Benefits of IPC Membership

Stay Current

BE INFORMED OF what's happening in the industry now and in the future ... and what you need to do to prepare and respond.

- Get **FREE** access to members-only online resources, including a searchable archive of original articles on the latest technical issues and industry/market trends.
- Access market research studies and reports and learn about the latest trends in markets and technology through IPC market research. Participate in statistical programs available for EMS, PCB, laminate, process consumables, solder and assembly equipment. Conferences are held each year in Asia, Europe and North America.
- Attend IPC workshops and educational conferences held throughout the year at special member rates.
- Stay abreast of critical technology changes and supply chain challenges stemming from global environmental directives, legislation and regulations.

Get Connected

NETWORK WITH YOUR peers, and put the resources of a global industry behind your most challenging technical and business issues.

- Get involved in committees and PCB/EMS management councils to help direct the future of the industry.
- Participate in problem-solving exchanges on a vast range of industry-specific issues and concerns through IPC's technical e-mail forums.
- Network with other industry professionals at premier industry events such as the international IPC APEX EXPO as well as the regional IPC Midwest Conference & Exhibition.
- Get priority responses to your technical questions from IPC's technical staff.

Shape the Industry

BECOME PART OF a worldwide network of professionals who are actively involved in shaping the future of our industry.

- Participate in developing industry standards your company, customers, competitors and suppliers use. IPC standards are internationally accepted due to affiliation with the American National Standards Institute (ANSI) and International Electrotechnical Commission (IEC).
- Make your voice heard in the regulations that affect your company by taking advantage of IPC-organized environmental and public policy activities.

Train Your People

INCREASE YOUR KNOWLEDGE and facilitate continuous learning among your technical staff.

- Attend any of the 75-plus workshops and international conferences IPC sponsors annually, where the exchange of technical information is unequalled.
- Participate in IPC's voluntary training and certification programs for a cost-effective, industry-recognized way to demonstrate your commitment to quality.
- Purchase IPC's award-winning CD- and DVD-based training materials at a discount, to cost-effectively build the foundation of your company's training program.