IPC-2221B APPENDIX A Version 3.0 August 2019

A.1 INTRODUCTION This appendix was developed by the IPC 1-10c Test Coupon and Artwork Generation Task Group and is included in this current document revision as a resource for the design of conformance and qualification coupons. The intent is to provide guidance as to the coupon designs, however if conflicts arise or the information provided is incomplete, the design of the coupon features should be in accordance with the associated product board design requirements.

It is the task group's recommendation that the coupons be designed by the printed board fabricator in order to ensure that the correct drill sizes are used. Additionally, etch and solder mask fabrication compensation shall be applied uniformly to both the coupons and product board after the coupons have been designed.

Solder mask layers are documented for each of the coupons, however they are only to be used if the associated product board design requires solder mask.

Table A.1-1 provides a summary of coupon designs that are described within this appendix.

A.1 Version Changes Changes that were incorporated in this version of Appendix A to IPC-2221B are indicated y gray shading of the relevant subsection(s). Changes to a figure or table are indicated by gray shading of the figure or table header.

Table A.1-1 IPC Coupons

Section	Coupon	Description	Purpose
A.2	AB/R	General purpose AB coupon for through features	Plated hole/via evaluation, feature size and spacing, registration, thermal stress and rework simulation
A.3	A/R	General purpose A coupon for use when B features are not present	Plated hole evaluation, feature size and spacing, registration, thermal stress and rework simulation
A.4	Propagated B	B coupon propagated (blind, buried or filled through) via features	Via evaluation, feature size and spacing, registration, thermal stress and rework simulation
A.5	E	Moisture and insulation resistance coupon	Moisture and insulation resistance
A.6	S	Hole solderability coupon	Hole solderability
A.7	W	Surface mount solderability coupon	Surface mount solderability
A.8	D	General purpose AB daisy-chain via coupon	Plated hole/via thermal stress
A.8.1	Propagated D	Daisy-chain via coupon for propagated structures	Plated hole/via thermal stress
A.9	G	Solder mask coupon	Solder mask adhesion
A.10	H	Surface insulation resistance coupon	Surface insulation resistance

A.11	P	Peel strength coupon	Peel strength and plating adhesion
A.12	Z	Controlled impedance coupon	Controlled impedance
A.13	K	Registration (internal spacing)	Verification of internal spacing between plated holes and copper on internal layers

A.2 AB/R COUPON Coupon AB/R combines the heritage A, B and R coupon design features along with a C feature which represents the smallest via or component hole which has the smallest annular ring. In order to better represent the product board the B1 feature contains internal lands only on layers 2 and n-1, the B2 feature contains internal lands only on internal signal layers and the B3 feature contains internal lands only on internal plane layers. The design also includes features to allow the assessment of minimum conductor and space widths from the product board. To accomplish this, the B4 lands are square and the minimum conductor for each layer is located adjacent to the B4 lands at the minimum spacing for each layer.

Even when there are no B features, the AB/R coupon can still be used as there will just not be any B sized drills. Until the Coupon Generator has been updated to accept designs without B features, the following is suggested to be used for the B: Land size: 1.02 mm [0.040 in] and Via size: 0.51 mm [0.020 in]. Once the coupon has been generated, the manufacturer should remove the B drills from the drill file.

The R feature provides a method to electrically assess 360° registration without the need for microsectioning. Due to the contribution of etch variation with heavier innerlayer foils use of the R features is recommended for design with foil weights of 1 oz. or less. The design parameters for coupon AB/R are shown in Table A.2-1.

Table A.2-1 AB/R Coupon Parameters, mm [in]

Feature	Description	Design Requirements
A	Largest component hole with its smallest associated D+ Round lands on all layers	Drill size **shall** be ≤ 1.07 [0.042] Land size **shall** be ≤ 1.65 [0.065]
B1	Smallest via with its smallest associated D+ Round lands on layers 2 and n-1	Land size **shall** be ≤ 1.02 [0.040] Grid size: 1.27 [0.050]
B2	Smallest via with its smallest associated D+ Round lands on signal layers	
B3	Smallest via with its smallest associated D+ Round lands on plane layers	
B4	Smallest via with its smallest associated D+ Square lands on all layers	
C	Smallest D+ with its smallest associated via or component hole Round lands on all layers	Drill size **shall** be ≤ 1.07 [0.042] Land size **shall** be ≤ 1.65 [0.065]
RA	Registration based on the A feature	Anti-land calculation (minimum of): (Land diameter + 0.0127 [0.0005]) - (2 x annular ring requirement)
RB	Registration based on the B feature	
RC	Registration based on the C feature	
RB1	Registration based on the B feature with a 0.0254 [0.001] allowance	Anti-land calculation: RB anti-land + 0.0508 [0.002]
G	Common connection for “R” measurements	Maximum drill size is 0.51 [0.020] (Square) Land size is 1.02 [0.040]

L	Minimum conductor width (in line with B4 lands)	Minimum conductor for each layer
S	Minimum space width (in line with B4 lands)	Minimum space for each layer
T	Tooling hole	Drill size is 2.00 [0.0787]

Note 1. Thieving may be added to the coupon provided it is in accordance with the associated product board design.
Note 2. The "R" measurements should be used with caution for copper weights greater than 1 oz. or when positive etchback greater than 0.0127 [0.005] is present.
Note 3. For direct plane layer connections B3 and B4 pad size are 1.02 [0.040].
Note 4. B1, B2 and B3 **shall** have lands on every layer for designs using non-functional lands.
Note 5. When the smallest B land exceeds 1.02 [0.040], a modified A/R coupon with a unique designation (e.g., A/R-1) **shall** be used to assess the via feature. Complete documentation of this new coupon and its design requirements will be provided in a subsequent revision to this Appendix.

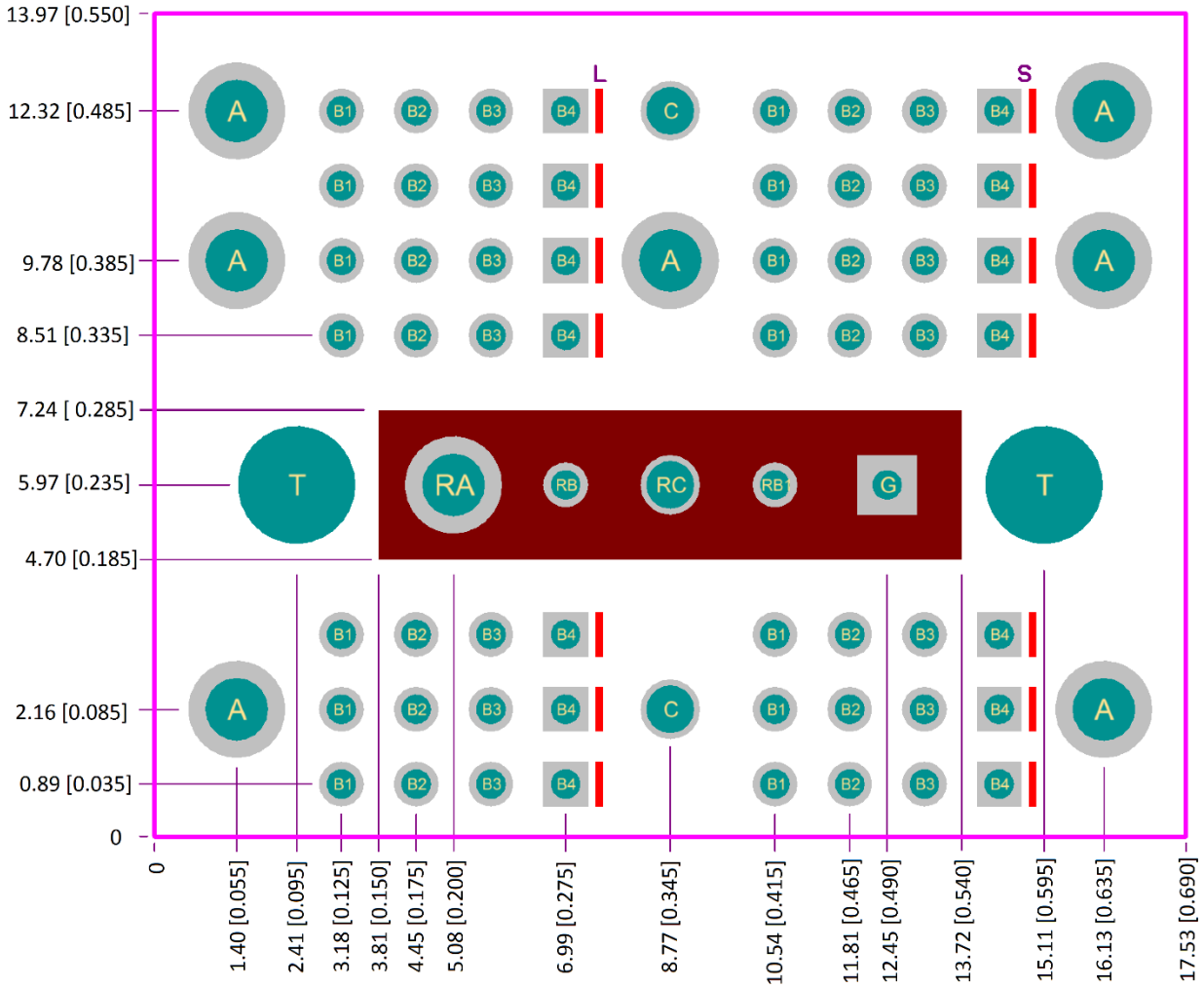


Figure A.2-1 AB/R Coupon Layout, mm [in]

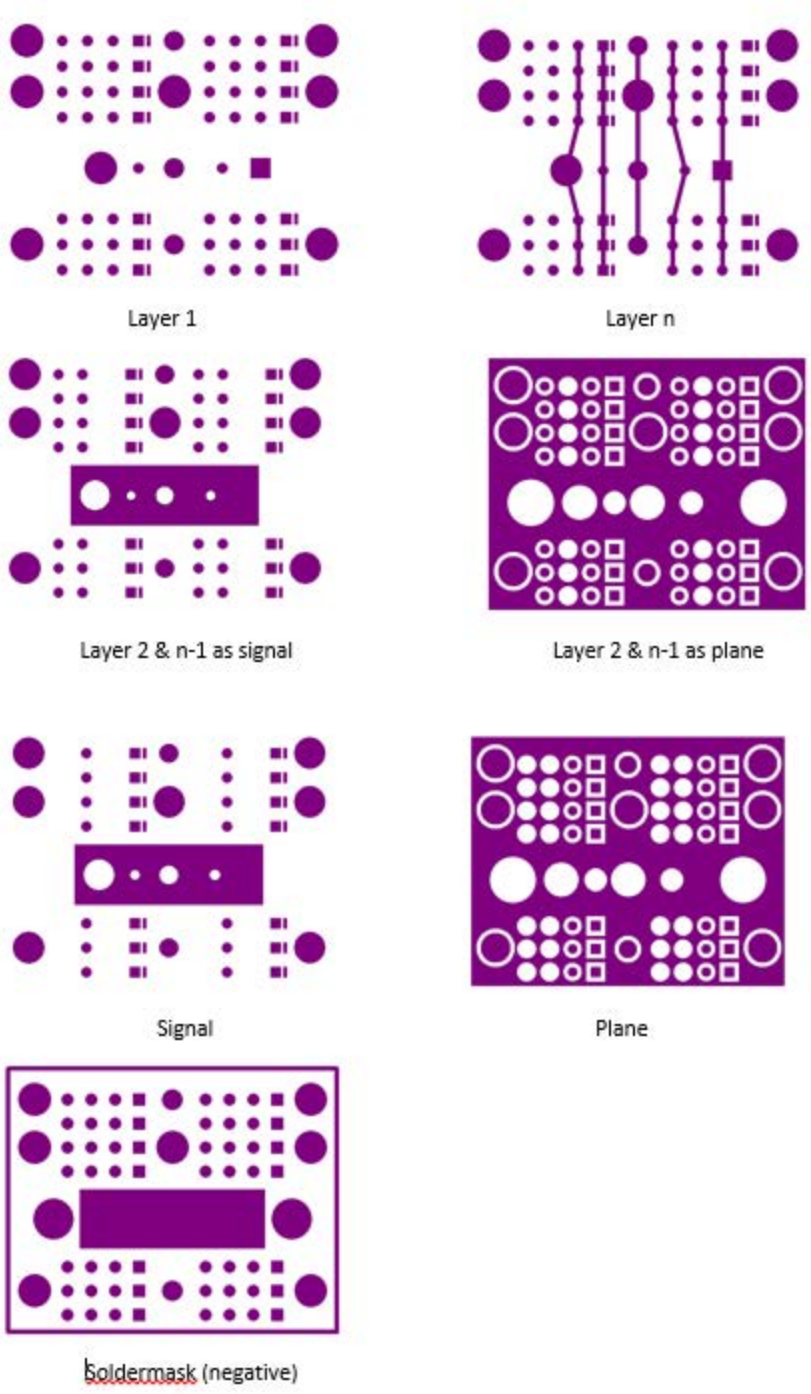


Figure A.2-2 AB/R Coupon Example Layers

A.3 A/R COUPON Coupon A/R follows the same design intent as the AB/R except that it is intended for use when the product board does not contain any vias. The design parameters for coupon A/R are shown in Table A.3-1.

Table A.3-1 A/R Coupon Parameters, mm [in]

Feature	Description	Design Requirements
A1	Largest component hole with its smallest associated D+ Round lands on layers 2 and n-1[4]	Drill size **shall** be ≤ 1.07 [0.042] Land size **shall** be ≤ 1.65 [0.065] Grid: 1.91 [0.075]
A2	Largest component hole with its smallest associated D+ Round lands on signal layers[4]	
A3	Largest component hole with its smallest associated D+ Round lands on plane layers[4]	
A4	Largest component hole with its smallest associated D+ Square lands on all layers	
RA	Registration based on the A feature	Anti-land calculation (minimum of): (Land diameter + 0.0127 [0.0005]) - (2 x annular ring requirement)
RA1	Registration based on the A feature with a 0.0254 [0.001] allowance	Anti-land calculation: RA anti-land + 0.0508 [0.002]
G	Common connection for “R” measurements	Maximum drill size is 0.51 [0.020] Land size is 1.02 [0.040]
L	Minimum conductor width (in line with B4 lands)	Minimum conductor for each layer
S	Minimum space width (in line with B4 lands)	Minimum space for each layer
T	Tooling hole	Drill size is 2.00 [0.0787]

Note 1. Thieving may be added to the coupon provided it is in accordance with the associated product board design.
Note 2. The “R” measurements should be used with caution for copper weights greater than 1 oz. or when positive etchback greater than 0.0127 [0.0005] is present.
Note 3. For direct plane layer connections A3 and A4 pad size are 1.65 [0.065].
Note 4. A1, A2 and A3 shall have lands on every layer for designs using non-functional lands.

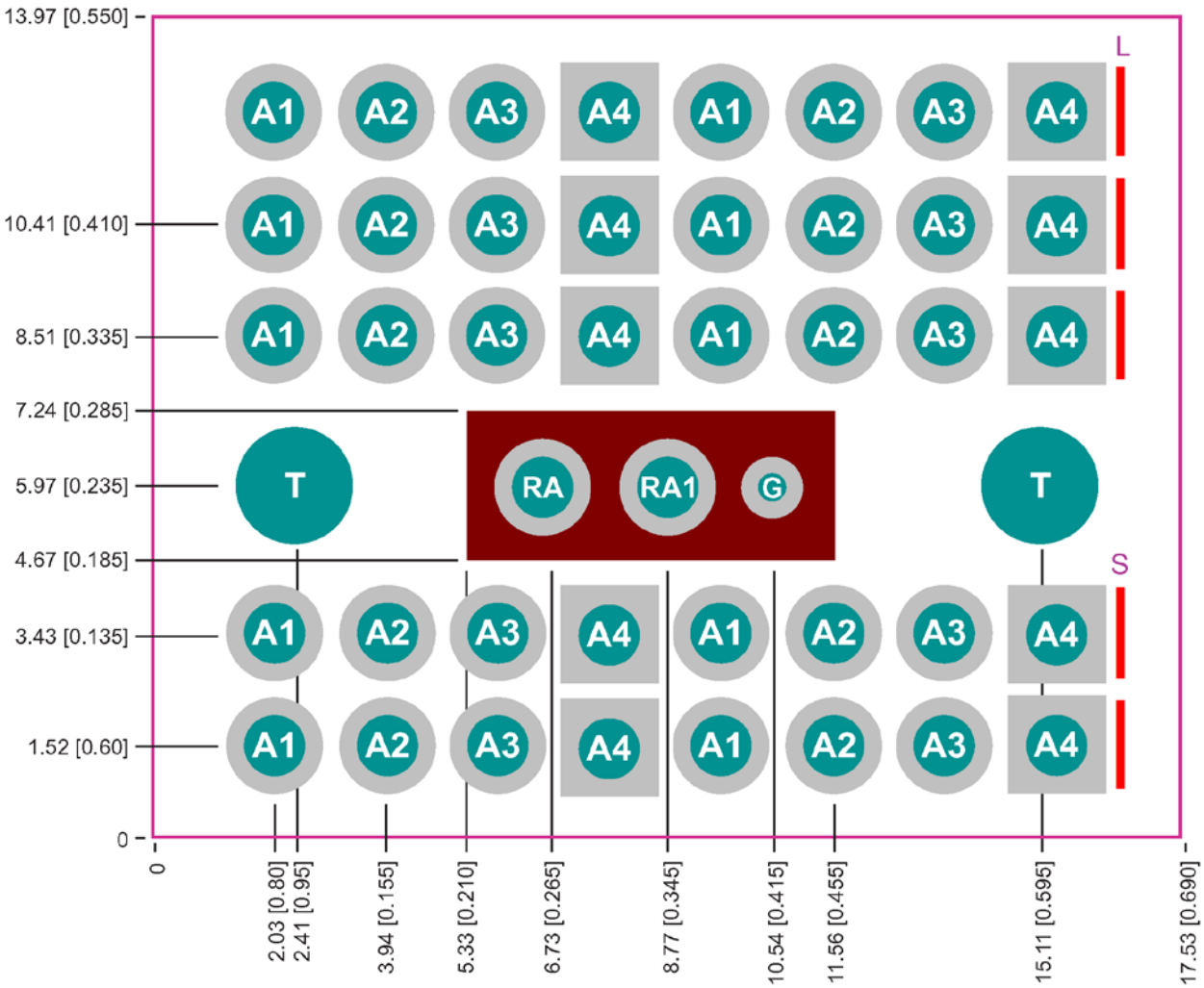


Figure A.3-1 A/R Coupon Layout, mm [in]

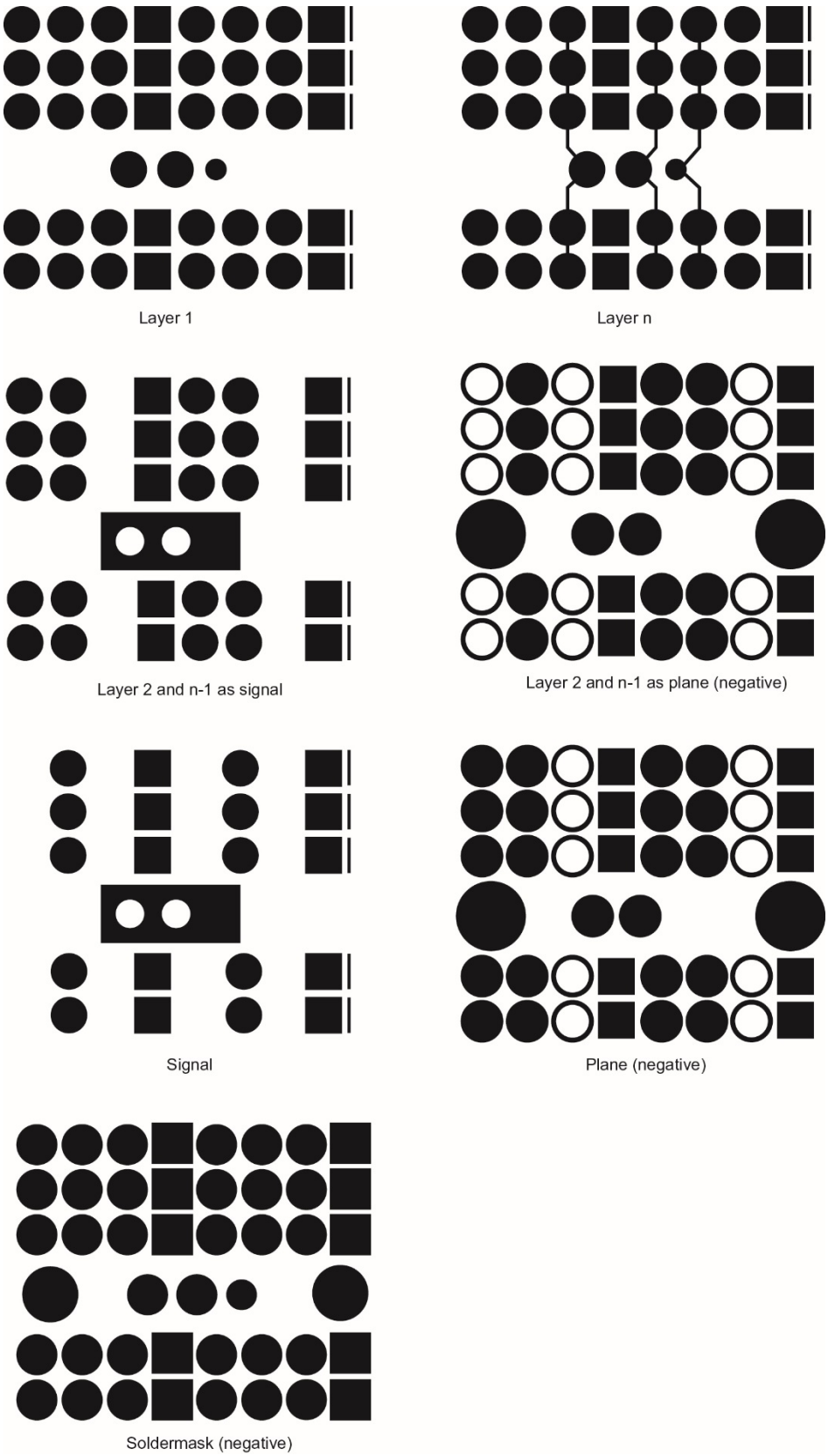


Figure A.3-2 A/R Coupon Example Layers

A.4 Propagated B COUPON Propagated B coupons are used to evaluate other board structures (blind, buried, filled, etc.) that don't meet the criteria for the AB/R coupons. Coupon B follows the same design intent as the AB/R except that it is based on the **structures** used for the propagated D coupon and does not include the R feature. As such, a maximum of two (2) via structures can exist in a B coupon, although these may represent many drilling and plating steps. (See A.8, D Coupon below.)

The first via structure (left half) is referred to as Bx and the second via structure (right half) is referred to as By. Each via structure can be either; through holes (filled or not), blind (stacked or staggered), buried or some combination of blind and buried. Each via structure is then separated into lower and upper sections. The lower section for each via structure is built with the structures that match the D coupon and is intended to be used to evaluate structural integrity of the structures.

The upper section is modified so that it can be used to evaluate registration. This requires any traces used for staggered interconnects be removed and any lands on a single layer be separated by 5 mils (edge to edge). A round .030 in land in the lower left of the coupon denotes the Structural Integrity section. The layer spans of each of the structures is also marked on layer 1.

When structures are stacked, both sections will be the same and only a single microsection is needed. But when staggered structures exist, two microsections need to be evaluated, one from each, upper and lower, of the sections.

The grid is defined as a function of the specified grid from the D coupon. If it is 0.025" or less then the X grid is 0.0375" and the Y grid is 0.025". If the D coupon grid is greater than 0.025" then the X and Y grids are 0.075". The exact X axis positioning of the features in the coupon will vary depending on the pitch. The Y axis (microsection planes) are fixed.

When the grid and feature sizes are small enough (e.g. single or stacked microvias), the design provides seven holes in each of the three microsection planes, each offset in alignment by 0.00846 mm [0.00033 in] to improve the probability of meeting the 10% via diameter requirement during microsectioning.

When spacing allows, additional holes are placed within the arrays to improve coupon copper density, to improve the representability of the coupon from a plating perspective. These holes are located between the microsection planes and are not intended to be evaluated. With the smaller (microvias) pitch, there are an additional two rows of vias between the primary microsection planes. With the larger pitch, an additional set of vias is placed in the center of the area between the holes to be evaluated.

Propagated B coupons are required for each via structure. If multiple vias are included in a via structure, individual B coupons are not needed for each individual via, since they are included in the structure. All drilling and plating operations required by the design shall be represented.

The design parameters for coupon B are shown in Table A.4-1. An example of a B coupon is shown in Figure A.4-1, which depicts stacked and staggered microvia structures.

Table A.4-1 B/R Coupon Parameters, mm [in]

Feature	Description	Design Requirements
BX	Smallest via of type “X” with its smallest associated D+ Round lands on representative layers[2]	Land size **shall** be ≤ 1.65 [0.065] Grid size: X and Y: 1.91 [0.075] or X: 0.95 [0.0375] Y: 0.63 [0.025] Y offset: 0.00846 [0.00033] (Depending on structure)
BY	Smallest via of type “Y” with its smallest associated D+ Round lands on representative layers[2]	
T	Tooling hole	Drill size is 2.00 [0.0787]

Note 1. Internal or external thieving may be added to the coupon provided it is in accordance with the associated product board design.

Note 2. BX and BY have lands on every layer for designs using non-functional lands in the registration section.

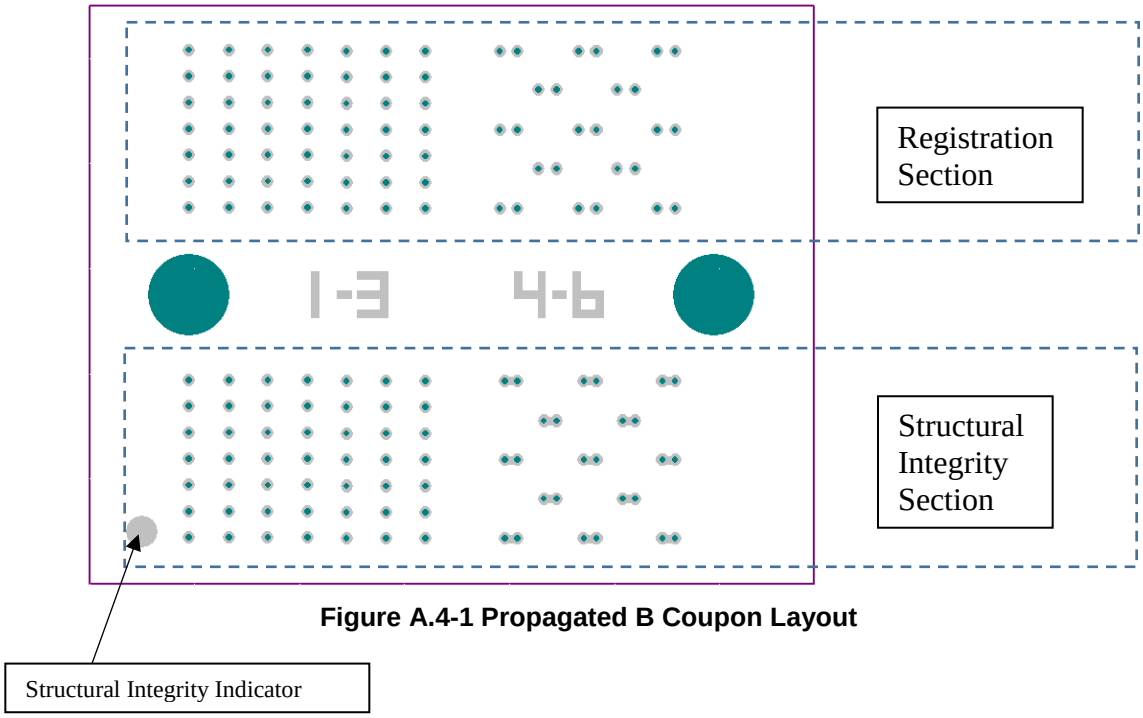


Figure A.4-1 Propagated B Coupon Layout

A.5 E COUPON Coupon E is used to evaluate moisture and insulation resistance of laminated base materials. The coupon is designed to test a maximum of ten layers. For designs with more than 10 layers additional coupons are required and each shall contain the last layer of the preceding coupon (e.g. L1 - 10, L10 - 19, L19 - 28, etc.). The design parameters for coupon E are shown in Table A.5-1.

Table A.5-1 E Coupon Parameters, mm [in]

Feature	Description	Design Requirements
1 - 10	Plated-through test points	Recommended drill size: 1.02 [0.040] Recommended land size: 1.52 [0.060] Grid: 2.54 [0.100]
Electrodes	Parallel electrodes	Width: 0.635 [0.025] Length: 25.40 [1.000] Gap width: 0.635 [0.025] Plane clearance: 0.635 [0.025]

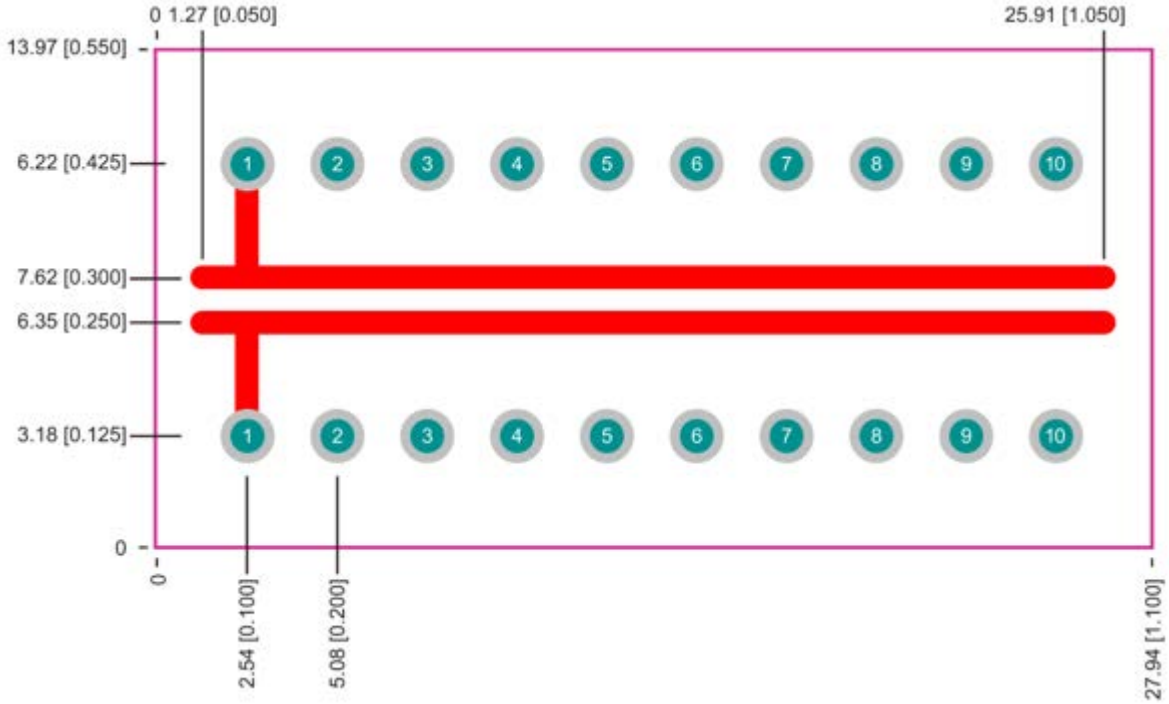


Figure A.5-1 E Coupon Layout, mm [in]

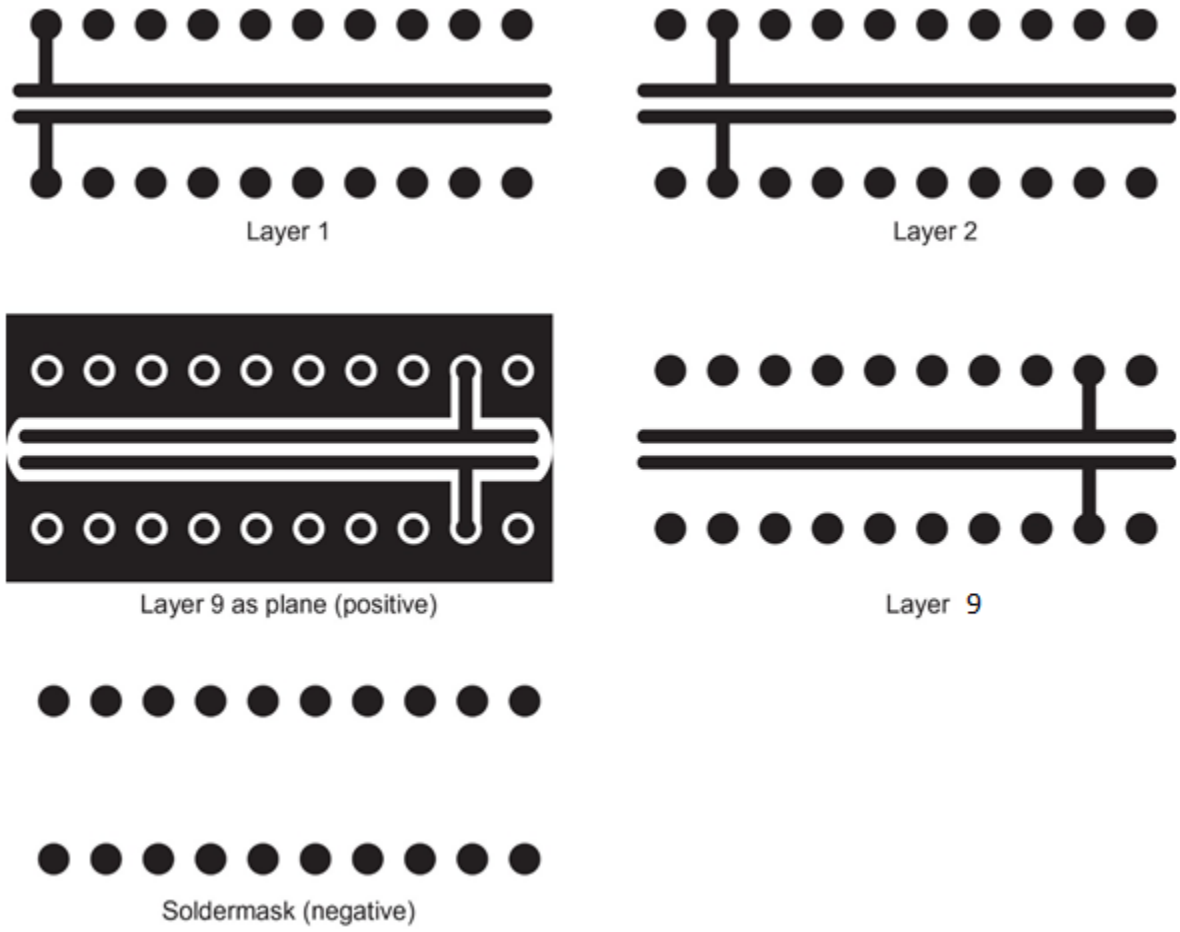


Figure A.5-2 E Coupon

A.6 S COUPON Coupon S is used to evaluate through hole solderability. Representative copper patterns may be placed on internal layers, however, no innerlayer lands are to be included in the design coupon. The design parameters for coupon S are shown in Table A.6-1.

Table A.6-1 S Coupon Parameters, mm [in]

Feature	Description	Design Requirements
S	Plated-through holes (32 each)	Drill size: 0.81 [0.032] Recommended land size: 1.52 [0.060] Grid: Staggered (see Figure A.6-1)

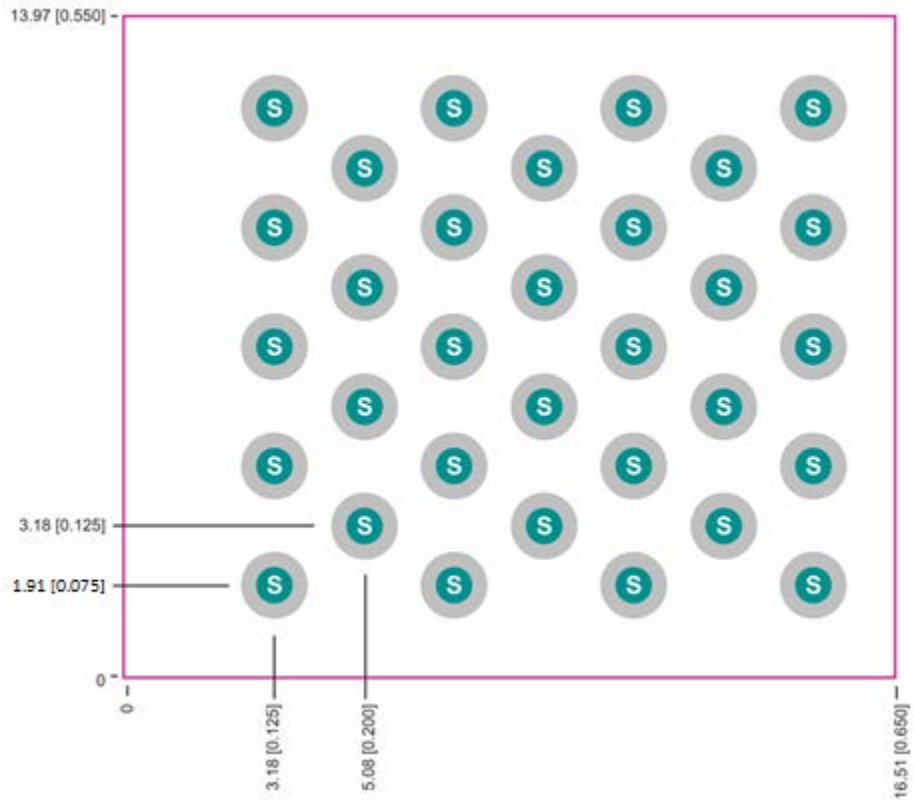


Figure A.6-1 S Coupon Layout, mm [in]

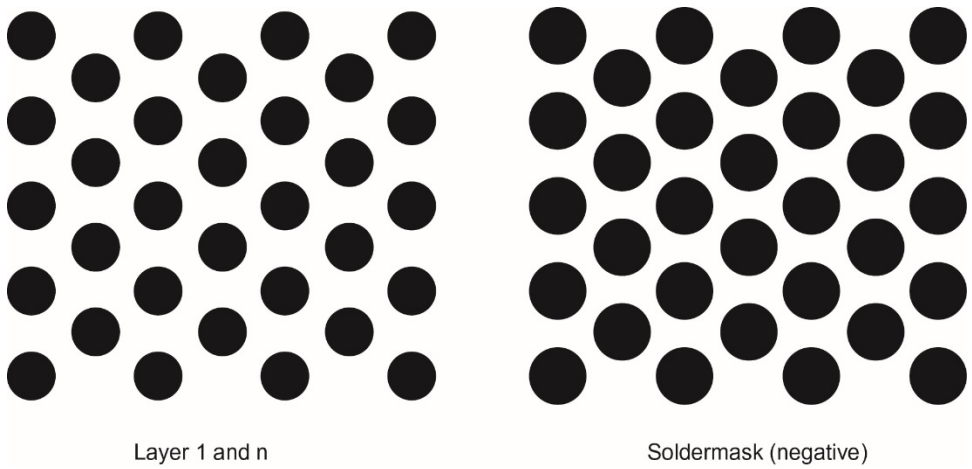


Figure A.6-2 S Coupon Example Layers

A.7 W COUPON Coupon W is a double-sided pattern used to evaluate surface mount land solderability. By intent, no inner layers patterns are included and no solder mask is to be applied to this coupon. Manufacturers should add copper to inner layers as necessary to allow the coupon thickness to match that of the design. The artwork pattern extends outside the rout line by intent to purposely cut through the pattern to facilitate wetting balance testing. The design parameters for coupon W are shown in Table A.7-1.

Table A.7-1 W Coupon Parameters, mm [in]

Feature	Description	Design Requirements
1	Round lands (12 each)	Land size: 1.52 [0.060] Drill Size: 1.15 [0.0453] Pitch: 1.91 [0.075] No solder mask on this coupon
2	Rectangular surface mount lands (12 each)	Land size: 10.16 x 1.52 [0.400 x 0.060] 2 clipped to dimension shown (.64 [0.025] overhang) Pitch: 1.91 [0.075] No solder mask on this coupon
3	Square surface mount lands (4 each)	Land size: 1.52 x 1.52 [0.060 x 0.060] Pitch: 1.91 [0.075]] No solder mask on this coupon
4	Alignment features for solder dip	Land size: 0.635 x 0.254 [0.025 x 0.010] Pitch: 0.508 [0.020] No solder mask on this coupon

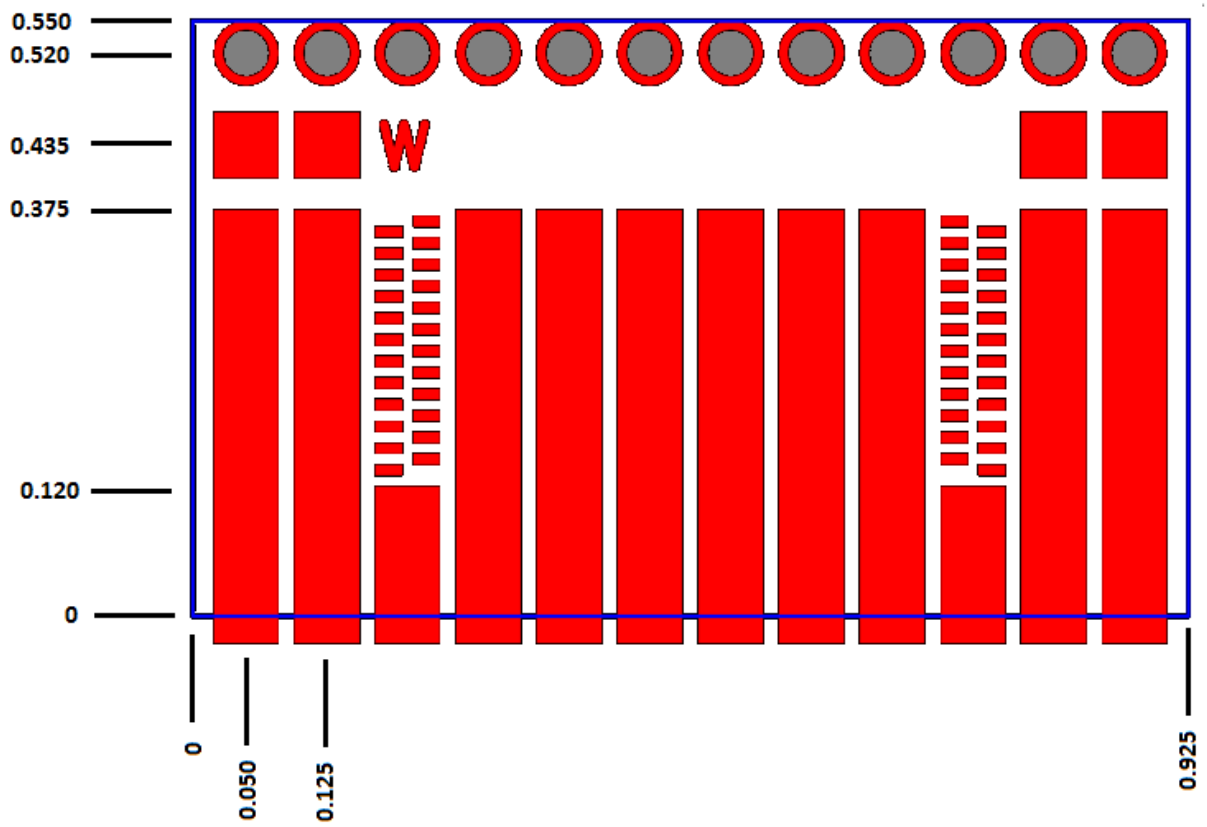
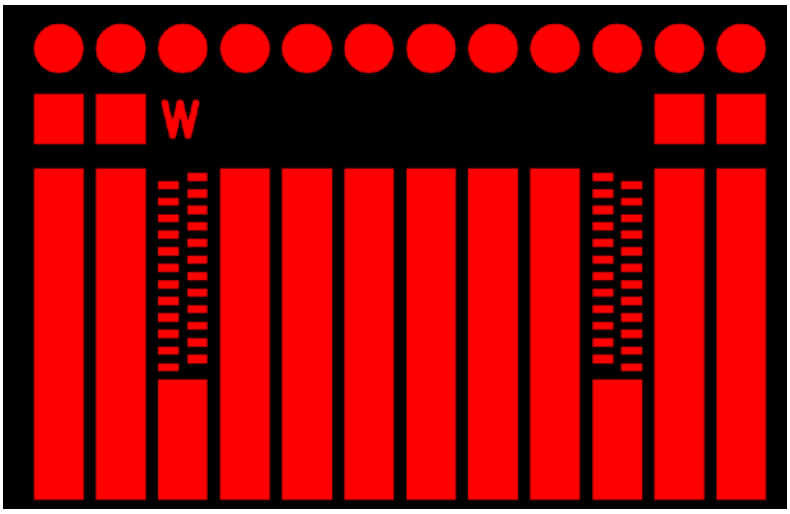


Figure A.7-1 W Coupon Layout, mm [in]



Layer 1 and n (no "W" on layer n)
Figure A.7-2 W Coupon Layout

A.8 D COUPON Coupon D is used to evaluate plated hole and via reliability by thermal stress. The coupon is designed to have a sufficient number of plated holes or vias in a chain to obtain a precise ~~precision~~ resistance measurement. The coupon may also be used for propagated via structures as described in Section A.4 by replacing the A and B features with the required propagated via features (see section A.8.1). The design parameters for coupon D are shown in Table A.8-1.

The daisy chain patterns that are used to connect the vias are placed on layers 2 and n-1 for through holes. They are placed on the outermost layers for through holes that are filled. The interconnect conductors that reach the connector mounting locations are located on the lowest numbered layer where daisy chain circuits exist.

Table A.8-1 D Coupon Parameters, mm [in]

Feature	Description	Design Requirements
A	Largest component hole with its smallest associated D+ Round lands on layers 2 and n-1	Drill size **shall** be ≤ 1.07 [0.042] Land size **shall** be ≤ 1.65 [0.065] Grid: 1.91 [0.075] Interconnect conductors: 0.254 [0.010] Interconnect sequence: Layer 2 to n-1
B	Smallest via with its smallest associated D+ Round lands on layers 2 and n-1	Land size **shall** be ≤ 1.02 [0.040] Grid size: 1.27 [0.050] Interconnect conductors: 0.254 [0.010] Interconnect sequence: Layer 2 to n-1
Connector	VH: Four-wire resistance high voltage input IH: Four-wire resistance current source VL: Four-wire resistance low voltage input IL: Four-wire resistance current sink	Finished hole size: 1.02 ± 0.076 [0.040 ± 0.003] Land size: 1.91 [0.075] Grid size: 2.54 [0.100] Interconnect conductors: on the lowest layer number where the daisy chain conductors exist

Note 1. Each of the chains **shall** contain only one unique via structure.

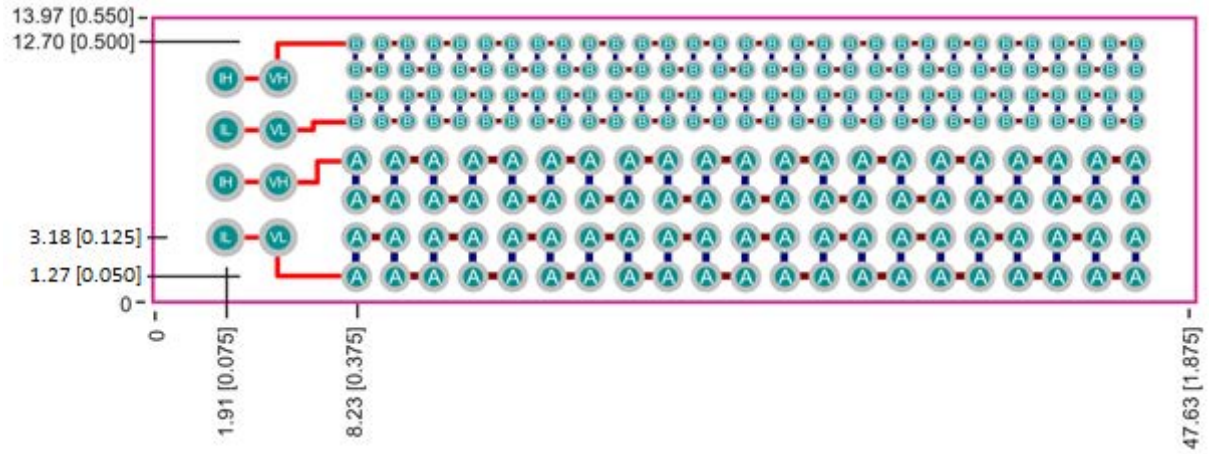


Figure A.8-1 D Coupon Layout with A and B Features, mm [in]

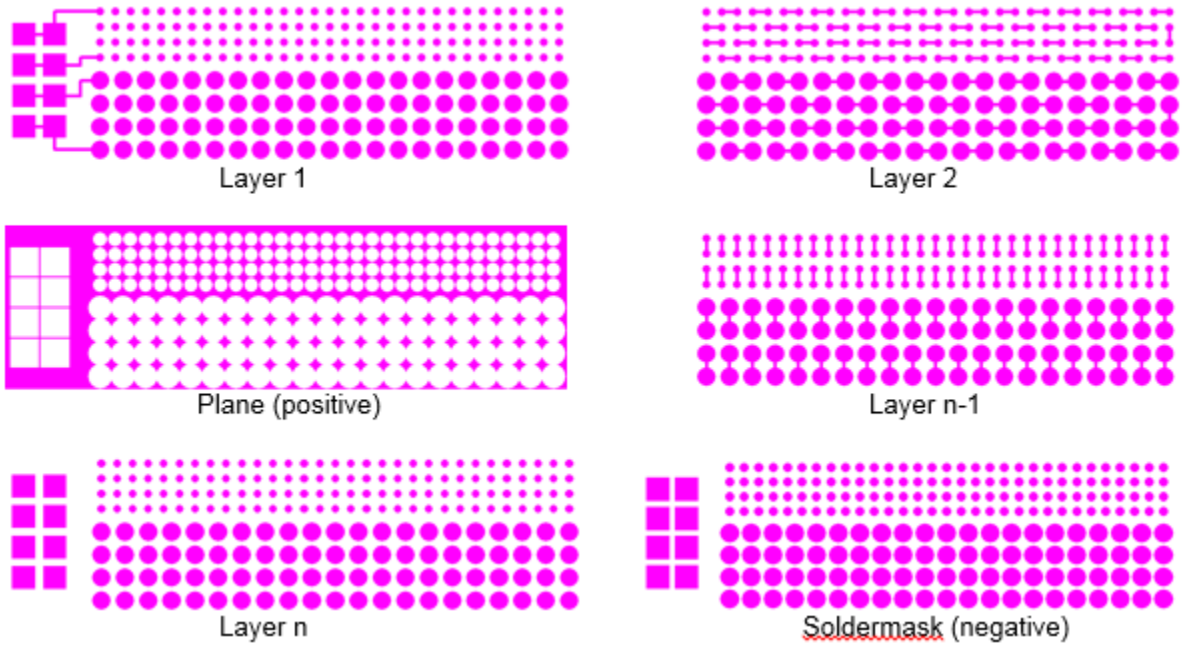


Figure A.8-2 D Coupon Example Layers with A and B (through hole) Features

A.8.1 Propagated D COUPON Propagated D coupons are used to evaluate via reliability by thermal stress. Just as the through hole D coupon, the propagated D coupon is designed to have a sufficient number of structures in a chain to obtain a precise resistance measurement.

The design parameters for propagated D coupons are the same as those for the through hole D coupon and are shown in Table A.8-1.

Propagated D coupons are used to evaluate other board structures (blind, buried, filled, etc.) that don't meet the criteria for the through hole D coupon. The Propagated D Coupon is intended to contain the most complex **structures** used in the board design.

A maximum of two (2) via structures can exist in a D coupon, although these may represent many drilling and plating steps. In some cases, multiple propagated D coupons will be necessary to cover all of the structures in a design.

Table A.8-2 Propagated D Coupon Parameters, mm [in]

Feature	Description	Design Requirements
Net 1 & Net 2	Smallest via used for each portion of the intended structure, with its smallest associated land. Round lands on all start and stop layers for all drills in the structure.	Drill size **shall** be ≤ 1.07 [0.042] Land size **shall** be ≤ 1.65 [0.065] Grid: Variable (≤ 1.91 [0.075]) Interconnect conductors: 0.254 [0.010] Interconnect sequence: Layers at the extremes of the structure
Connector	VH: Four-wire resistance high voltage input IH: Four-wire resistance current source VL: Four-wire resistance low voltage input IL: Four-wire resistance current sink	Finished hole size: 1.02 ± 0.076 [0.040 ± 0.003] Land size: 1.91 [0.075] Grid size: 2.54 [0.100] Interconnect conductors: 0.254 [0.010] on the lowest layer number where the daisy chain

Note 1. Each of the chains **shall** contain only one unique via structure.

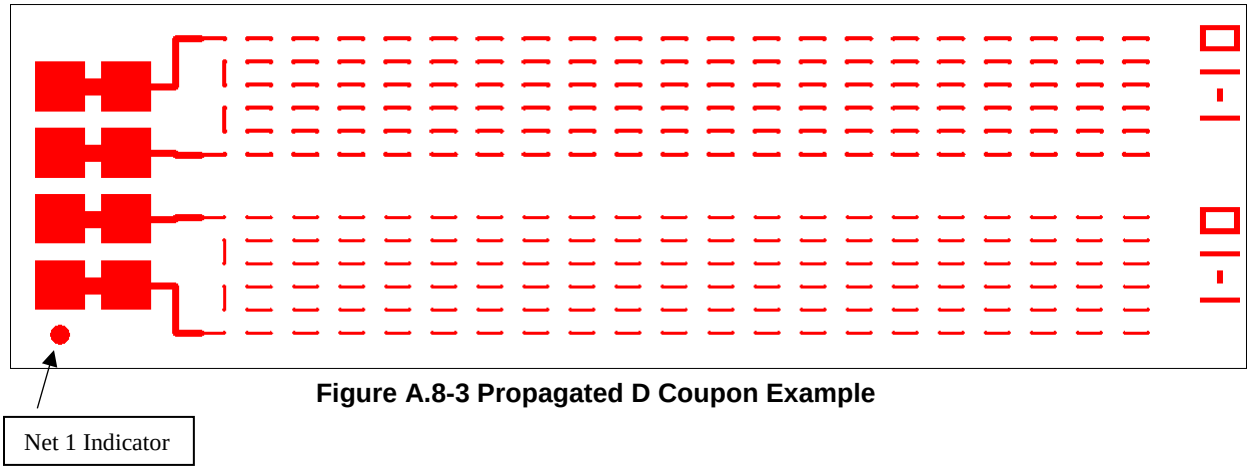


Figure A.8-3 Propagated D Coupon Example

A.8.1.1 Structures An individual drilling/plating sequence was previously considered to be a structure. In many of today's designs, a single net may utilize many different, closely spaced vias. To address this, structures are now defined to include holes drilled and plated at different times when they are closely spaced and used together in the design. Vias must have a single intermediate layer in common in order to be combined together to form a structure.

Vias that connect multiple layers are identified as a "via structure". These include:

- Vias that extend completely through the board.
- Blind or buried vias that only go partially through the board.
- Filled vias that extend completely through the board are considered to be a separate structure (separate from unfilled vias).

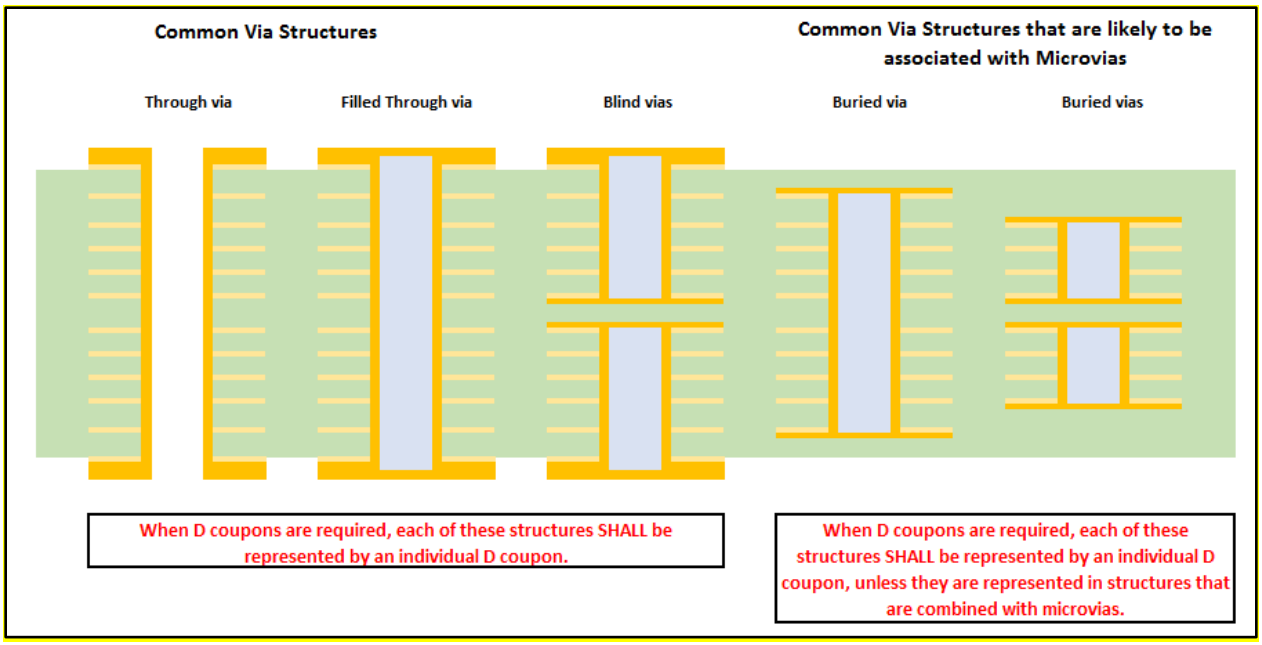


Figure A.8-4 D Common Via Structures

Vias with pad spacing closer than 0.16 mm [0.006 in] (two thermal zones) are considered to be a single structure. Vias with greater pad spacing are considered to be independent structures.

Structure complexity:

- Stacked structures are considered to be more complex than staggered structures.
- "Taller" (those involving more layers) structures are considered to be more complex than shorter structures.

Rules for selection of the most complex structures for D coupons :

- When multiple structures span the same layers, only the most complex are required to be represented by a D coupon.
- All drilling and plating steps **shall** be represented.

Multiple independent structures can be combined into a single net on a D coupon using a pad to pad spacing of 0.16 mm [0.006 in] minimum.

Multiple vias that connect the same layers (e.g. a via from layers 2-3 and a via from 2-5, which both connect layers 2 and 3) **shall** not be considered to be in the same structure.

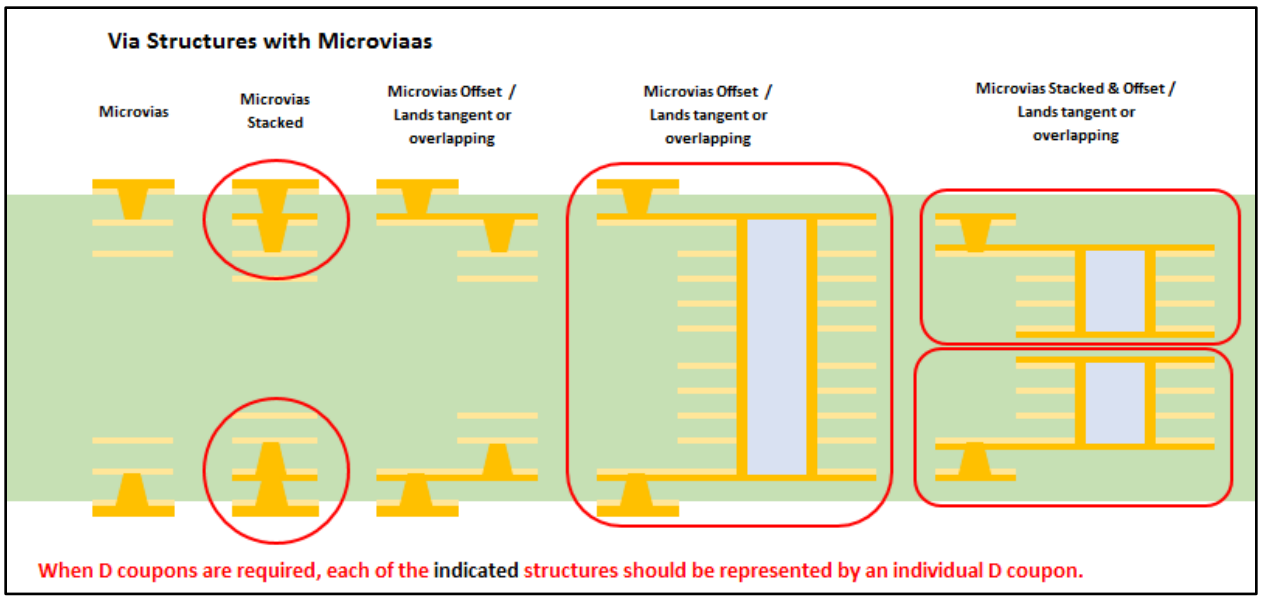


Figure A.8-4 Via Structures with Microvias

A.9 G COUPON Coupon G is used to evaluate solder mask adhesion and is divided into three regions 1) solder mask over copper or surface finish, 2) solder mask over laminate and 3) minimum solder mask web and minimum interconnect conductor width. The surface finish shall represent the product board design and the minimum web spacing and minimum conductor are per the product board. The design parameters for coupon G are shown in Table A.9-1.

Table A.9-1 G Coupon Parameters, mm [in]

Feature	Description	Design Requirements
1	Rectangular lands	Grid size: 1.27 [0.050] Solder mask anti-land calculation: 1.27 [0.050] - minimum solder mask web Land calculation: Solder mask anti-land - (2 x minimum solder mask clip back)
2	Round lands	
C	Minimum conductor	Minimum interconnect conductor associated with the solder mask web features
SMOC	Solder mask over copper or surface finish	Solder mask anti-land size: 0.61 [0.024] Grid size: 0.64 [0.025]
SMOL	Solder mask over laminate	Solder mask anti-land size: 0.61 [0.024] Grid size: 0.64 [0.025]

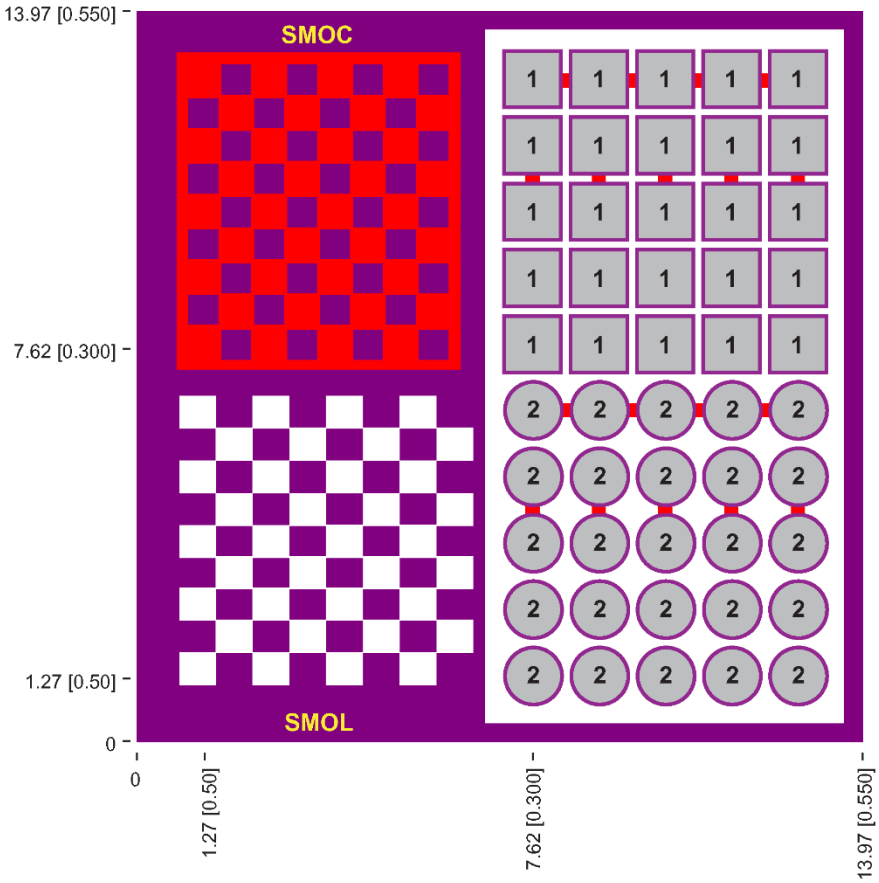


Figure A.9-1 G Coupon Layout, mm [in]

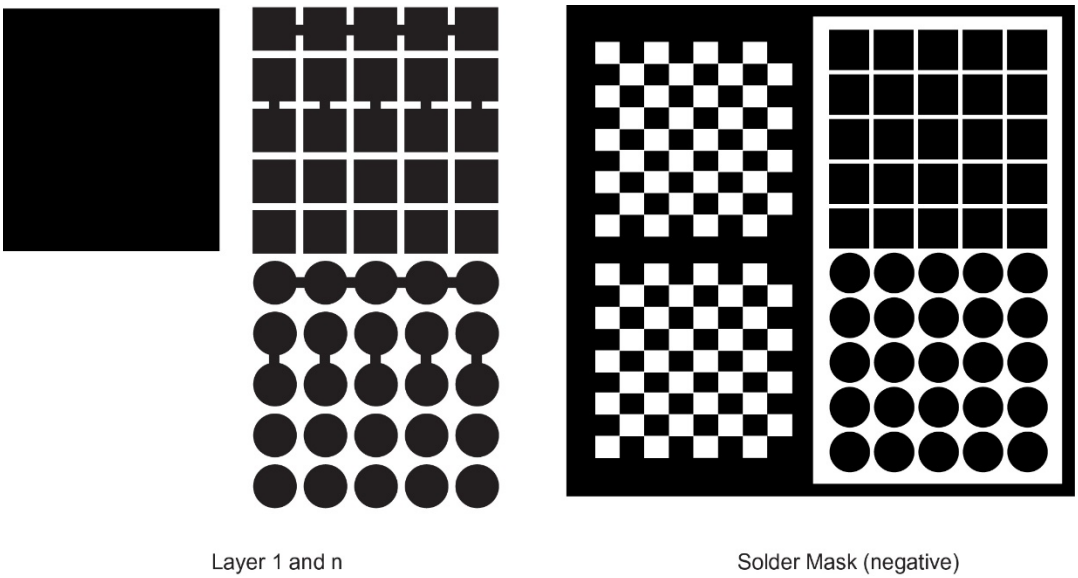


Figure A.9-2 G Coupon Example Layers

A.10 H COUPON Coupon H is used to quantify the effects of process and/or handling residues on surface insulation resistance. The coupon consists of an interstitial comb pattern per panel side. Representative copper patterns may be placed on internal layers. While a solder mask image is

documented, the pertinent performance specification may preclude the use of solder mask on the coupon. The design parameters for coupon H are shown in Table A.10-1.

Table A.10-1 H Coupon Parameters, mm [in]

Feature	Description	Design Requirements
1	Plated-through test points	Recommended drill size: 1.02 [0.040] Recommended land size: 1.52 [0.060]
Electrodes	Parallel electrodes	Width: 0.40 [0.016] Pitch: 0.60 [0.024]

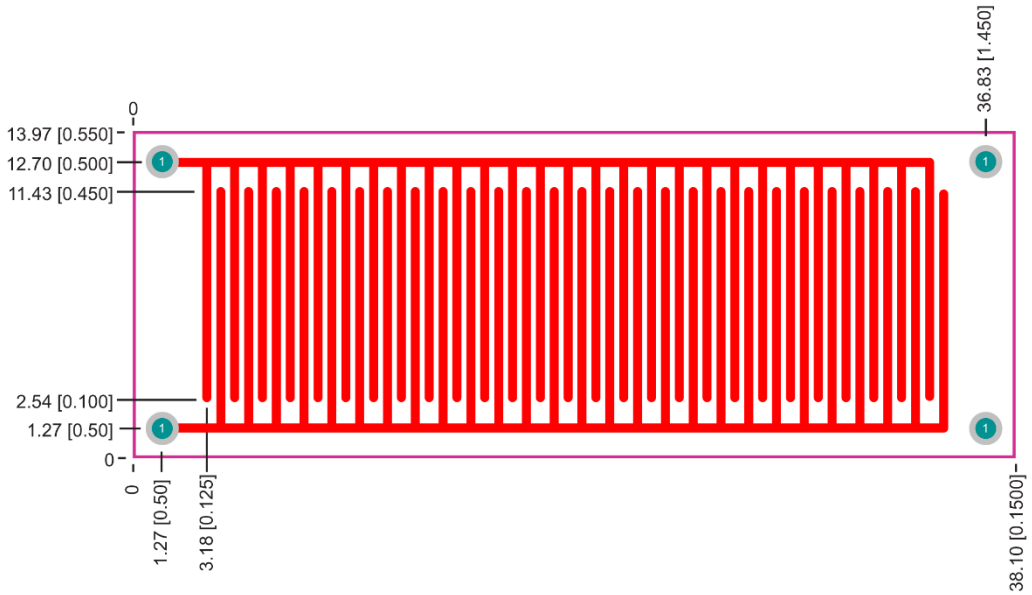


Figure A.10-1 H Coupon Layout, mm [in]

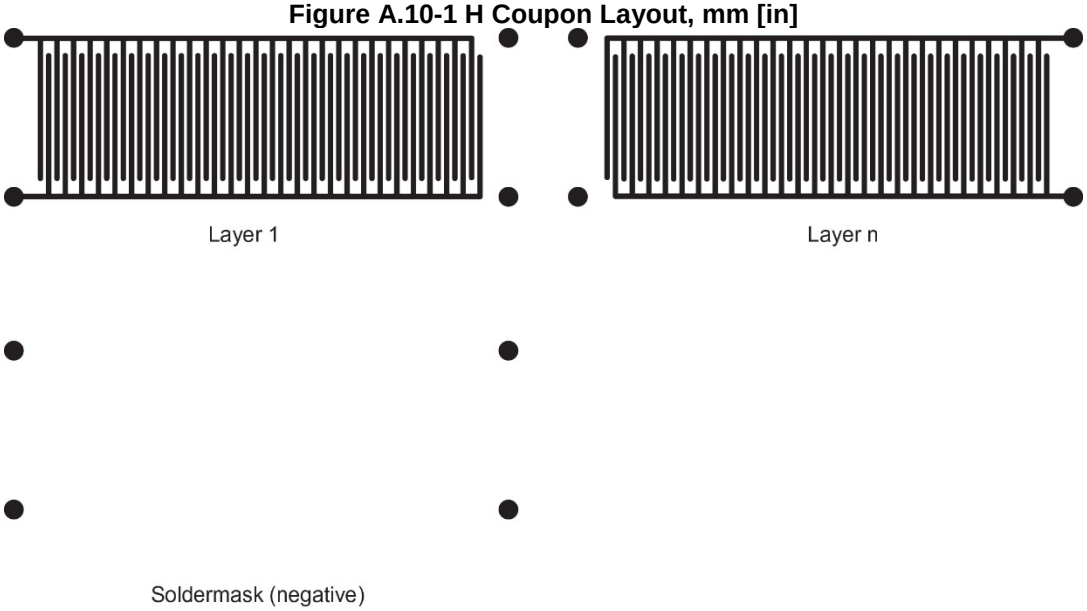


Figure A.10-2 H Coupon Example Layers

A.11 P COUPON Coupon P is used to evaluate the peel strength of metallic foils laminated to the outer layers of a printed board during the foil lamination process and to evaluate plating adhesion. The coupon

consists of a conductor pair per panel side that provides a minimum test length of 25.40 [1.00]. Representative copper patterns may be placed on internal layers. The design parameters for coupon P are shown in Table A.11-1.

Table A.11-1 P Coupon Parameters, mm [in]

Feature	Description	Design Requirements
Peel	Peel conductor	Width: 3.18 [0.125] Minimum length: 31.75 [1.250]
Tab	Peel tab	Width: 5.72 [0.225] Length: 6.99 [0.275]

Note 1. Performance specifications preclude the use of surface finish on the coupon.

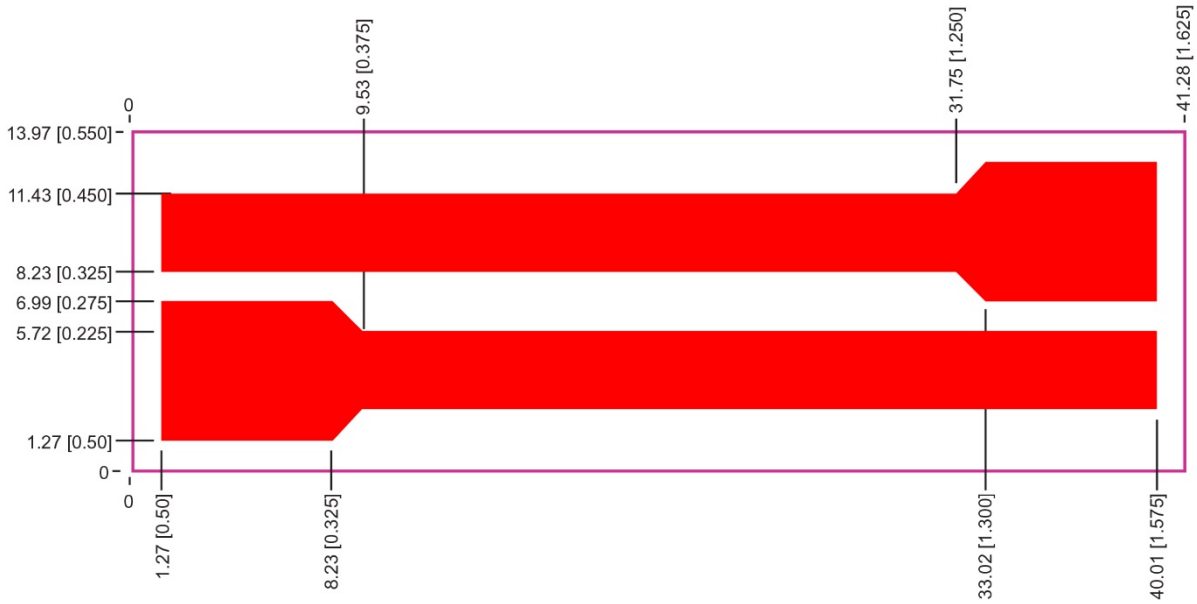
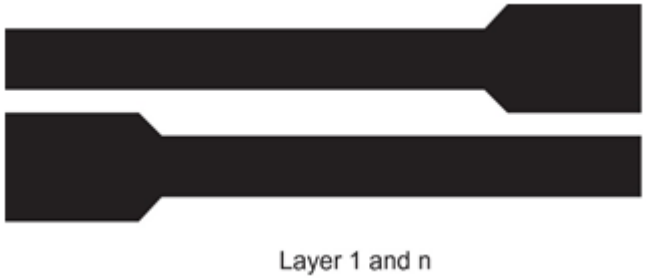


Figure A.11-1 P Coupon Layout, mm [in]



A.12 Z COUPON Coupon Z is used to determine the impedance value of controlled impedance structures of the printed board. Two structures either single-ended and/or differential per layer may be incorporated. The coupon may be designed with up to 24 single-ended structures, 12 differential structures or a combination of the two. The coupon provides for a minimum conductor length of 114.30 [4.50]. The design parameters for coupon Z are shown in Table A.12-1.

Table A.12-1 Z Coupon Parameters, mm [in]

Feature	Description	Design Requirements
TP	Plated-through test points (48 each)	Recommended drill size: 1.02 [0.040] Recommended land size: 1.52 [0.060] Grid size: 2.54 [0.100]
Conductors	Single-ended (2 test points): • Microstrip • Stripline Differential (4 test points): • Edge-coupled microstrip • Edge-coupled stripline • Broadside coupled stripline	Width(s): Per product board requirements Minimum length(s): 114.30 [4.50] Differential spacing: Per product board requirements Corners: Radiuses or 45 degree turns **Note**: Care should be taken to ensure constant separation of conductors on differential corners.
Planes	Reference planes	Edge of coupon clip-back: 0.254 [0.010] Spacing to edge of test-pad: 0.508 [0.020]

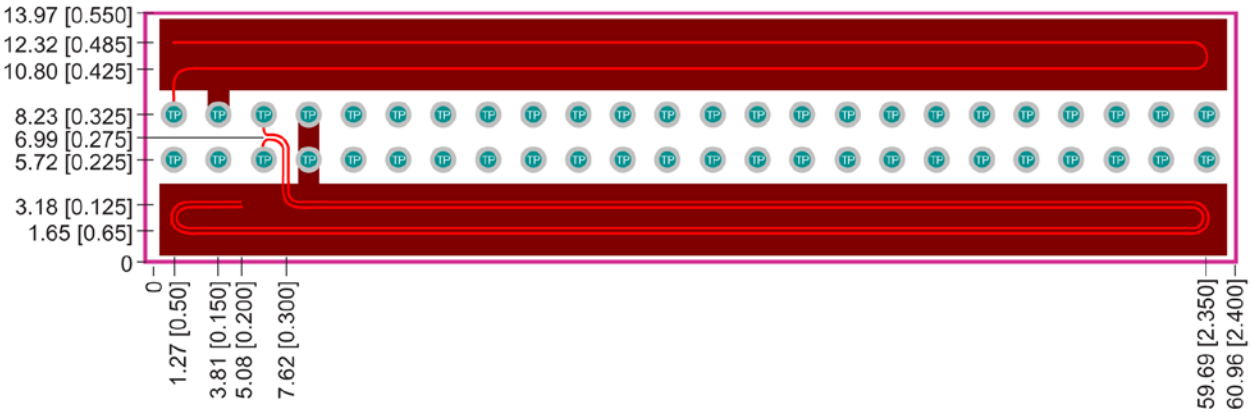


Figure A.12-1 Z Coupon Layout (Microstrip and edge-coupled microstrip), mm [in]

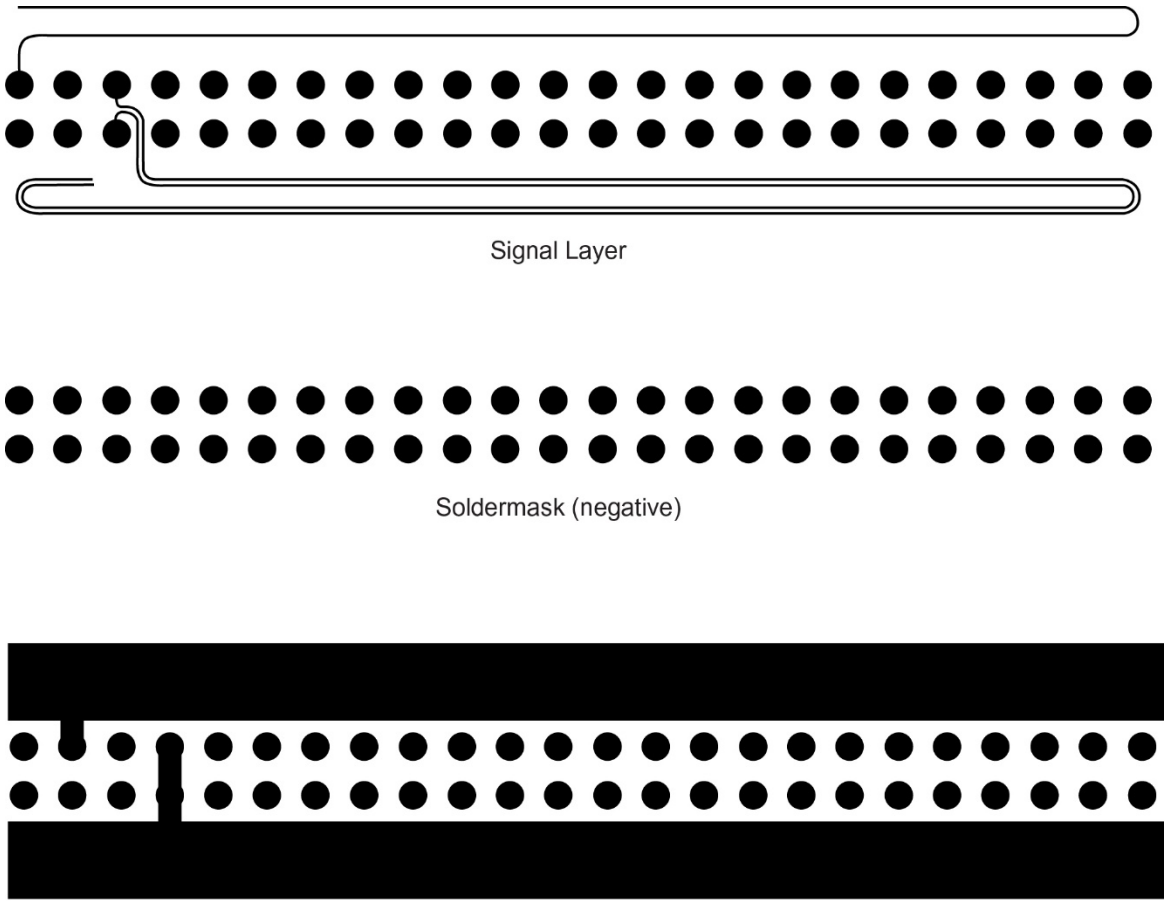


Figure A.12-2 Z Coupon Example Layers

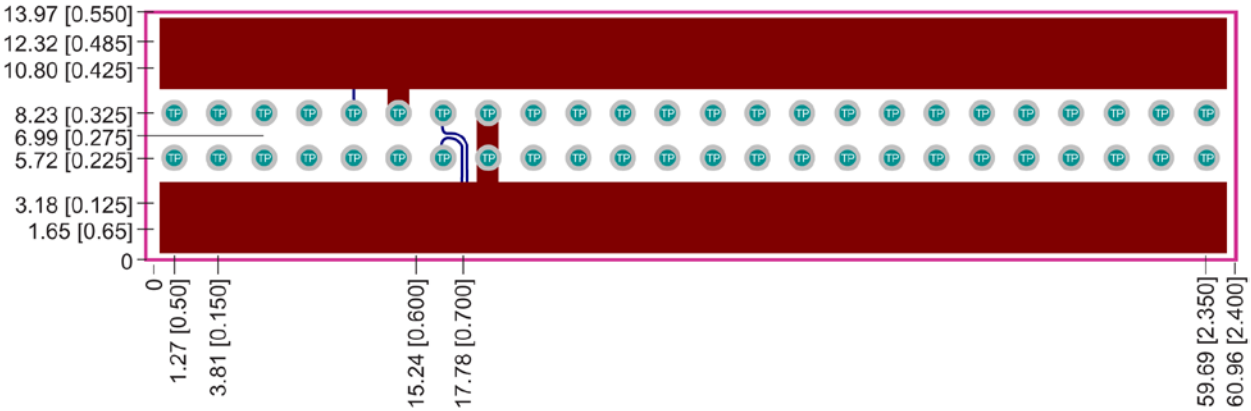
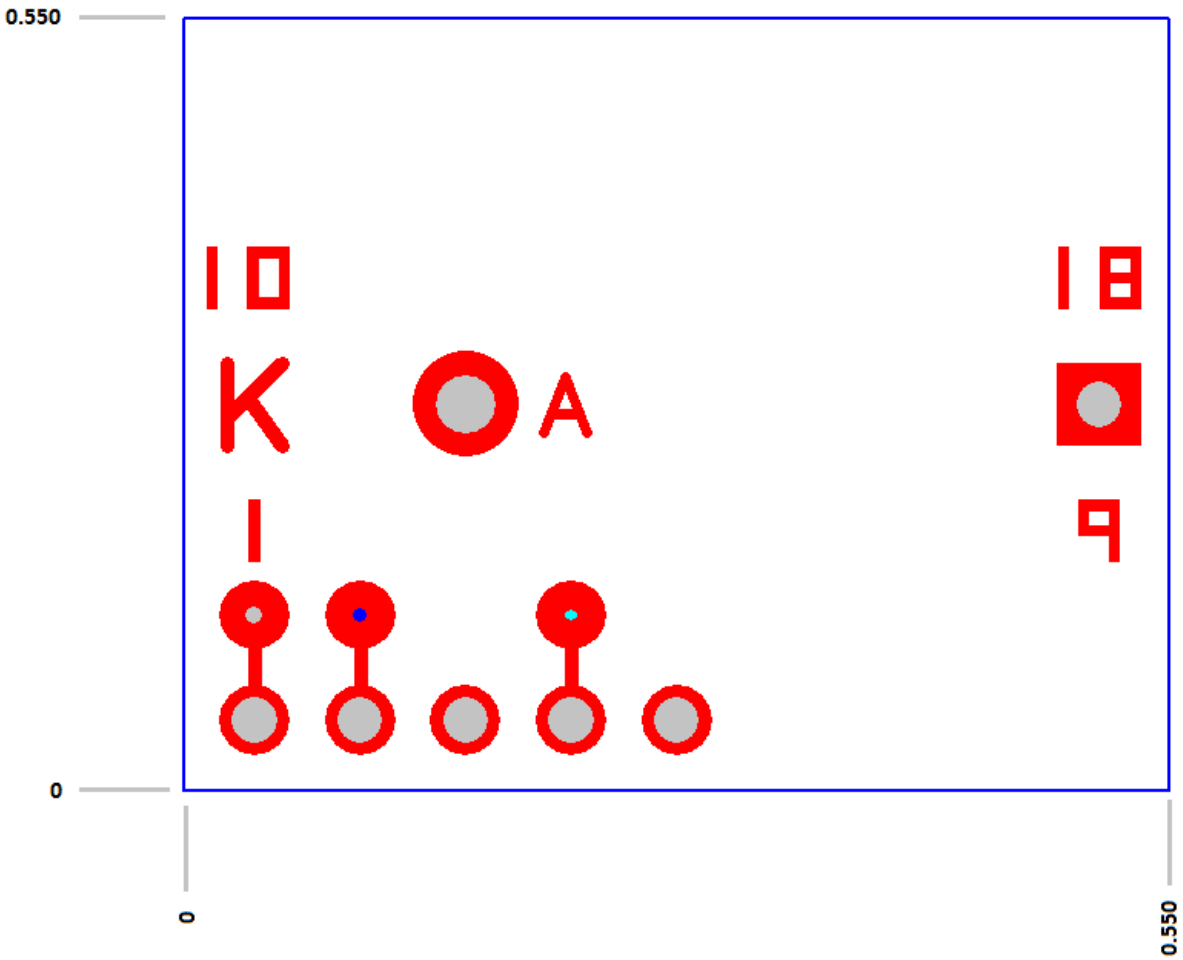


Figure A.12-3 Z Coupon Layout (Microstrip and edge-coupled microstrip using alternative test points), mm [in]

A.13 K COUPON Coupon K is used to verify that minimum spacing requirements exist between plated holes and copper on internal layers. The coupon is tested electrically using an ohm meter. A short between any of the round via test points and the square common test point indicates a failure.



Structures represented:

- Component hole A
- Unfilled through via B1
- Filled through via B2
- Buried via B3
- Blind via from layer 1 B4
- Blind via from layer n B5

- **Figure A.13-1 K Coupon Layout, mm [in]**