

OUR STORY

Candor was created in 1999 to respond to 21st Century customers who need innovative solutions for complex technologies.

As an inventor, the president of Candor, Yogen Patel, foresaw that standard manufacturing methods possessed limitations to achieving this goal. By inventing and developing certain technological advancements, Candor put itself in the forefront as a solution provider.

This process has replaced many undesirable products and chemistries from fabrication processes found in the majority of PCB shops around the world.

We can genuinely claim to be an “environment-friendlier” company since having completely eliminated the use of formaldehyde, EDTA, dry-film resists, ammonia based etch chemistry and tin-lead resists and their associated by-products.

OUR FUTURE

Candor is committed to giving supportive and accomodating service to our customers.

We strive to push the envelope of PCB manufacturing using advanced technology. We achieve this by always going back to our simplified process mantra. Our R&D helps both our customers and the industry toward a brighter future.

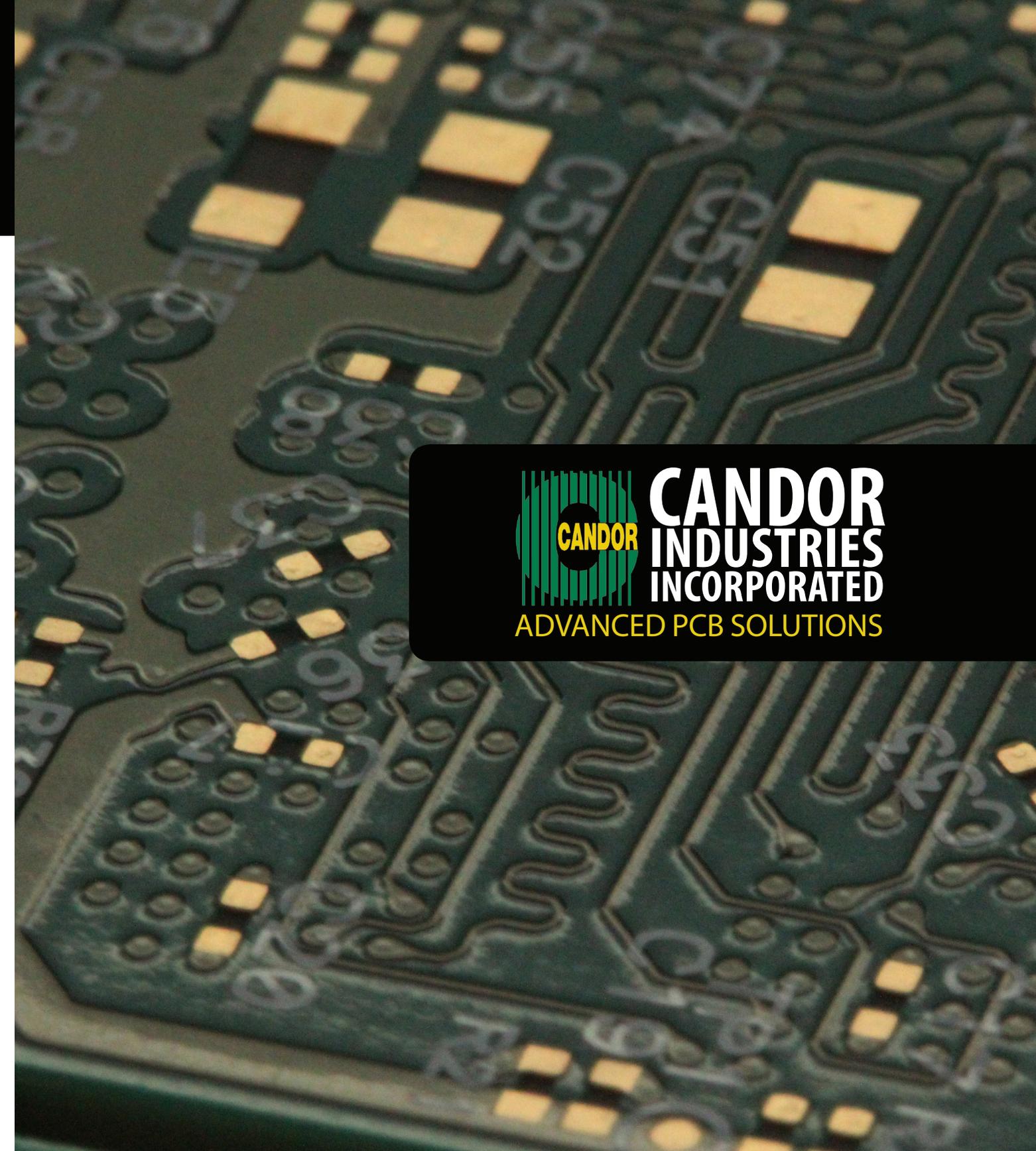


CONTACT US
CANDOR INDUSTRIES INC.
9-125 MARTIN ROSS AVENUE
TORONTO, ONTARIO, CANADA.
M3J 2L9

 **PHONE:** (416) 736-6306
 **FAX:** (416) 736 9387

.....

 **SUPPORT:**
SUPPORT@CANDORIND.COM
SALES:
SALES@CANDORIND.COM



CANDOR
INDUSTRIES
INCORPORATED
ADVANCED PCB SOLUTIONS

WHY WORK WITH CANDOR?

IT ALL STARTS WITH OUR UNIQUE COLLECTION OF PROCESSES:

BENEFITS

Continuous Foil Lamination:

This process uses a roll of copper interlaced between each panel, applying direct heat from the foil to the panels.

Even distribution of dielectric.

Contact Drilling:

Mechanically drilled blind vias by measuring depth at the instant drill bit makes contact with the laminate surface.

Purely mechanical drilling, no laser ablation issues.

Graphite Metalization:

Using conductive colloid solution to electrically charge surface of vias in preparation of plating.

Copper to copper bonding for superior adhesion.

Panel Plating:

Plating is done before imaging, plating throughout the manufacturing panel

There is no photoresist entering the plating tanks and therefore, no contamination from resist. Tensile strength is maintained at 45000 lb/in², elongation is above 30%. No need for expensive pulse plating systems, reducing cost of processing.

Positive Acting Electrolytic Photoresist:

The only company in the world to fully research the positive acting organic liquid electrodeposited photoresist being used at Candor.

This gives a thin, strong coat of photo resist in all crevases, and holes.

CAPABILITY UNLOCKED

Under +-5% impedance tolerance, extremely consistent and precise thickness. Easily laminates specialty materials.

4 mil blind vias, 6 mil buried vias and through holes.

Increased performance against destructive testing, more reliable holes.

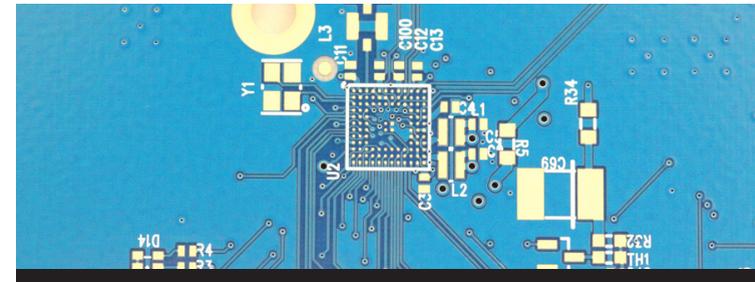
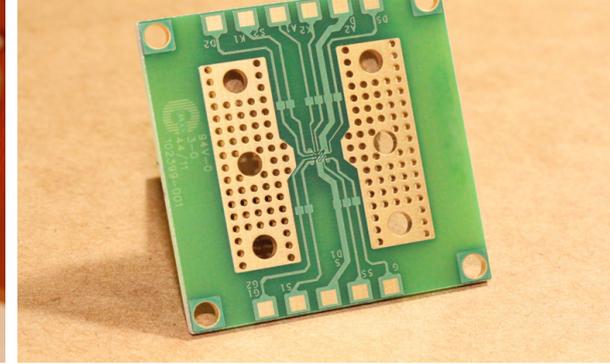
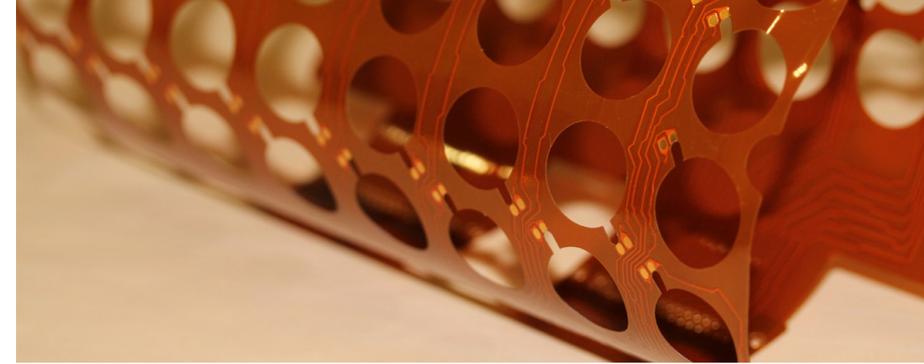
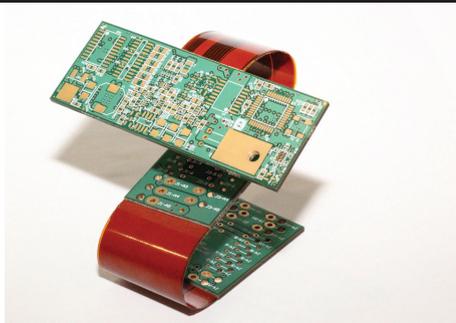
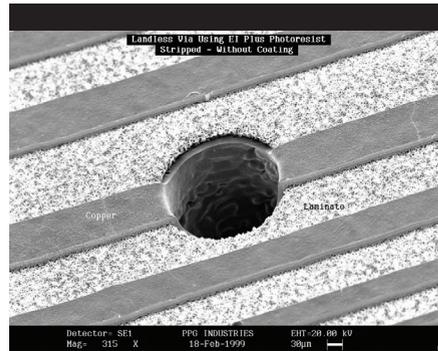
Even plating thickness both in hole and on the surface. Under +-5% impedance tolerances with, extremely accurate and even plating in holes. Dependable prototyping and consistent production.

- Thick, heavy copper boards can be manufactured with ease
- Excellent impedance matching - under +-5% tolerances
- Landless via capabilities
- Fine line capabilities
- Partial Plating

Cupric Etching

Very sharp etching, no overhang or undercut.

Heavy copper boards or ground right next to fine line designs can be etched with ease.



PRINTED CIRCUIT BOARD DESIGN LIMITS

This lists show what you can achieve working with Candor Industries.

Rigid, rigid-flex, and flex circuit boards have the same design limits.

SPECIFICATION	DESIGN LIMIT
Minimum Trace Width	2 Mil
Outer Layer Via Pad Size	Class 3: 2 Mil, and capable up to landless via
Line to Line Spacing	2 Mil
Minimum Through Hole Drilled	Minimum: 6 Mil
Minimum Buried Via Drilled	Minimum: 6 Mil
Minimum Blind Via Drilled	Minimum: 4 Mil
Blind Via Aspect Ratio	8:1 Aspect Ratio
Maximum PCB Thickness	.3"
Number of Layers	40 Layers
Controlled Impedance Tolerance	Under +- 5% Tolerance
Maximum Board Size	19.5" X 22.5"
Copper Thickness/Density	Able to meet customer requirements
Minimum Drill-to- Conductor	6 Mil (edge of hole to track)
PCB Edge to Conductor	2 Mil
Warp (Bow and Twist)	Exceeds IPC-TM-650
Solder Mask Clearance	2 Mil
Solder Mask DAM	2 Mil