I-Connect007

OCTOBER 2019

PCB007 MAGAZINE

The Changing Landscape

PCB007 MAGAZINE

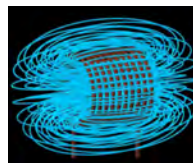
The Changing Landscape of the PCB Industry

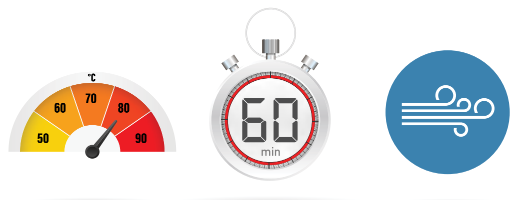
For printed circuit fabricators, the terrain continues to change. While automation is a key part of our transformation, human expertise and problem-solving remain central to the process. This issue considers the current, ever-changing landscape of our industry, from global economics to emerging technologies and processes that may change how the manufacturing floor operates.

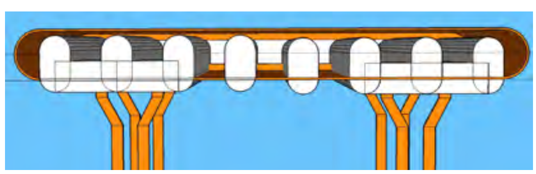
FEATURE ARTICLE:

FEATURE COLUMN:

ARTICLES:







PCB007 MAGAZINE

COLUMNS:

SHORTS:

HIGHLIGHTS:

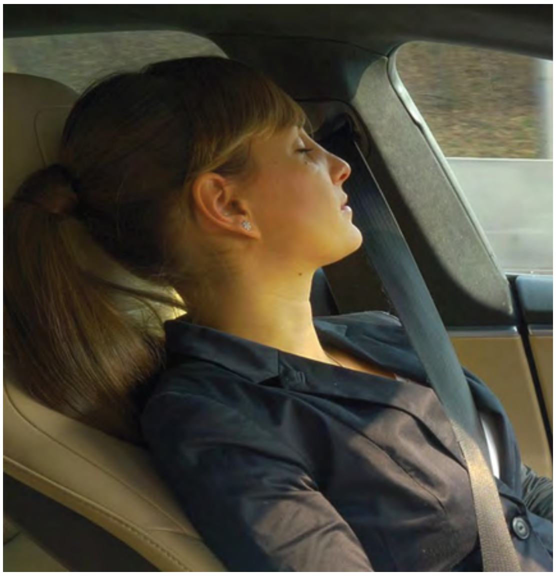
DEPARTMENTS:

Asleep at the Wheel?

Nolan's Notes

by Nolan Johnson, I-CONNECT007

We've all heard the news stories about drivers taking "full" advantage of the conveniences offered by the current crop of semi-autonomous vehicles. We've seen the photos and the videos of drivers reading or, even worse, sleeping while their car drives itself in freeway traffic. On the one hand, it's shocking that people would put so much trust in the system to keep them safe—some people close to me can't even sleep in a vehicle as a passenger, let alone as the driver.



But on the other hand, I suppose we really shouldn't be surprised. Human ingenuity is driven by laziness, and much of the new, emerging technologies enable that kind of creativity. Legend has it that Benjamin Franklin—American publisher, inventor, and statesman—once credited his inventions on a tendency to be "industriously lazy." And who among us hasn't, at some point in their life, extended the function of some purpose-built item or tool to solve a new, unanticipated problem? Surely, you've used a screwdriver as a chisel, for example. Or maybe you're one of those folks who has folded a post-it note folded into a catch basin and attached it below where you drill a hole in the wall to catch all of the wall dust? Mac users, you do know how to open a crown bottle cap with your wall plug, don't you? Give people a tool—and an unfulfilled, unrelated need—and they'll figure out a way to fill their need with what they have.

Yes, resourcefulness is at the center of the art of being "industriously lazy." But let's not forget that current self-driving features are intended to serve as driver assistance, not driver replacement. Ultimately, it is the human operators who are expected and required to stay alert and on call to take over driving responsibilities whenever necessary. Still, I can hear some of you muttering, "If, when using driver assist, I have to stay as alert as if I were driving in the first place, then what is the value to that technology?" And that's a valid point.

Consider, then, the "call my car" feature on some current autonomous vehicles. The concept is simple: beep your car, and the autonomous driving features activate, maneuvering your car to your location to pick you up. Think of this as a built-in valet for your car; it seems quite convenient. And it's easier to wrap one's head around than letting your car drive itself down the freeway at life-ending speeds.

But there's a risk even in valet mode: pedestrians. Imagine if every car in a mall parking lot could start moving by themselves without any operator present. The presence of a person in a vehicle provides the cue that we look for to assess the likelihood that a car near us could start moving. Take away that critical cue, and we could have a world where any car could start moving at any time. Parking lots could turn into a motion-based minefield. The liability is huge. In this world, who would send their child into the parking lot to retrieve something from the car under those circumstances? Accidents are inevitable, and who will stop the unattended car when it makes a mistake?

There will be solutions to these new safety concerns. As humans, we are good at innovating things; we just don't know how those changes will play out. And this is true in parking lots as well as fabrication floors. For printed circuit fabricators, the terrain continues to change under our feet. I'm almost hesitant to list them once again: ever-smaller fabrication dimensions; new yield expectations; environmental extreme survivability; high-speed performance, requiring new materials; chip and component packaging changes; advances in 3D printing that now include the ability to print passive components into the substrate itself; and other additive processes that enable those small fabrication dimensions. At this moment, our list has come full circle—an upward technology spiral.

This issue considers the current, ever-changing landscape of our industry—a landscape being reshaped by "industrious laziness." New application spaces will continue to drive our industry to improve our capabilities, and changes to our market landscape will only continue. We're automating manual processes with the intent to make our results better. But, like autonomous vehicle enhancements, human awareness and supervision are required. The skillset for the operator may change, but if we leave everything to the machines, eventually, something bad is bound to happen.

As we survey the moving, changing landscape, we begin with John Mitchell's column "Opportunities for Learning Abound at IPC." Then, we bring you a perspective from China; Yonglin Gong files this report, "Chinese Review: The 2018 NTI-100 Top Global PCB Fabricators."

Next up is Raymond Goh's column titled "The Laminate Market: What Will the Future Bring." Then, we focus on a new technology that aligns with changing methods in the landscape with a piece from Chemplate Materials' Víctor Lázaro Gallego, discussing induction lamination in "Multilayer Press Technology Using Magnetism to Produce Lamination Heat."

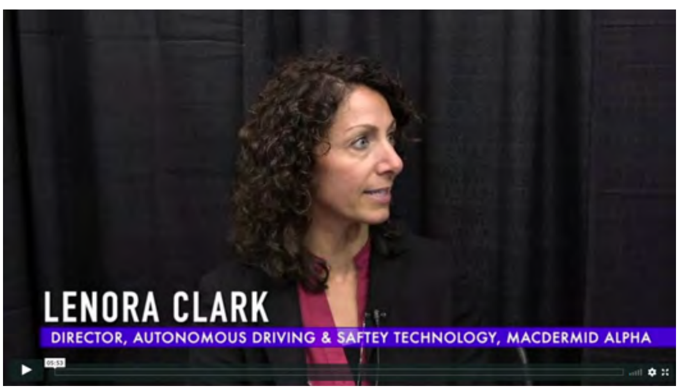
Mike Hill writes Part 2 of his column on "Technology and Reliability Demands Drive Designers and MIL-PRF-31032 Specification." Tara Dunn considers one of the key new market drivers in her column "Additive Electronics Momentum," which is in line with the SMTA Additive Electronics Conference on October 24, 2019. Right behind is Mike Carano with "Via Hole Filling and Plugging, Part 2."

Turning technical, Nikolaus Shubkegel explains solder mask tack dry. And Joan Tourné continues his series on VeCS with the fourth installment on "Tuning Your Signal Performance."

In this changing industry landscape, ask yourself the question: are you asleep at the wheel? **PCB007**

Nolan Johnson is managing editor of *PCB007 Magazine*. Nolan brings 30 years of career experience focused almost entirely on electronics design and manufacturing.To contact Johnson, click here.

RTW SMTAI 2019: Lenora Clark Discusses Company Changes and Her New Role



At the recent SMTA International 2019 Conference and Exhibition in Rosemont, Illinois, Lenora Clark, director of autonomous driving and safety technology at MacDermid Alpha Electronics Solutions (and co-organizer for the upcoming SMTA Additive Electronics Conference), and Nolan Johnson discuss her new role, recent company reorganization, and the synergies customers are seeing.

Click on the image to watch the interview.

Opportunities for Learning Abound at IPC

One World, One Industry

by Dr. John Mitchell, IPC—ASSOCIATION CONNECTING ELECTRONICS INDUSTRIES

It's fall, and students are heading back to school. Will you be joining them? IPC offers many opportunities for continuous learning, enabling you to enrich your career and increase your value to your company.

Courses on IPC's online learning platform, IPC EDGE, are created by subject-matter experts who respond to the requests from members of the electronics industry by providing video presentations, clear text, detailed instructions, interactive activities, and practice quizzes designed to make even the most complex industry topics easy to understand and master.

The Certified Electronics Program Manager (CEPM) program, which is one popular IPC course, is designed to provide the tools and training needed to ensure clients for life. From the big picture to the smallest detail, program management is a must for success. This new, instructor-led online course combines the educational benefits of live instruction and group discussions with the flexibility and cost savings of e-learning.

If your focus is on IPC standards, the Certified Standards Expert (CSE) course might be a better choice. The CSE is an organization's subject-matter professional with a high level of knowledge and understanding of a specific IPC standard or group of standards. The CSE course differs from that of the CIT in that a CIT is expected to train others, whereas a CSE is an internal subject-matter expert. IPC Train-

ing Centers offer training and testing for CSE certification.

Another course that might interest you is electrical discharge (ESD) control for electronics assembly. You are probably aware of the risk of ESD and that it can cost the industry up to $5 billion per year. To address this critical issue for our members, we created ESD control for electronics assembly in collaboration with the EOS/ESD Association. This course covers the causes of ESD, and the steps needed to mitigate its effects when handling, storing, or transporting ESD sensitive components in a manufacturing facility. This is an online, self-paced course with open enrollment throughout the year.

Perhaps you need a fuller understanding of PCB design. IPC recently introduced PCB design fundamentals, a two-part online course featuring a live instructor with more than 25 years of experience in design. Participants will learn to create accurate, IPC-compliant schematics from initial schematic to final layout. Registration for the 2020 courses will begin soon.

If you like to browse before you buy, check out IPC's online video training library. With more than 70 video training programs, the library provides the foundational knowledge required for hand soldering, IPC-A-610 acceptance criteria, rework and repair, components, wire assembly, and more. You can browse all titles, content descriptions, and even view free previews before deciding which course is best meets your professional development needs. The following programs will be added to the library by the end of the year: selective soldering, soldering iron tip maintenance, print reading, and surface-mount technology (SMT) component removal.

These are only a few of the many courses IPC offers as part of our ongoing commitment to continuous learning. For more information on IPC's educational offerings, visit the "knowledge" tab on IPC's website or contact me or David Hernandez, IPC's vice president of education at DavidHernandez@ipc.org. PCB007

Dr. John Mitchell is president and CEO of IPC-Association Connecting Electronics Industries. To read past columns or contact Mitchell, click here.

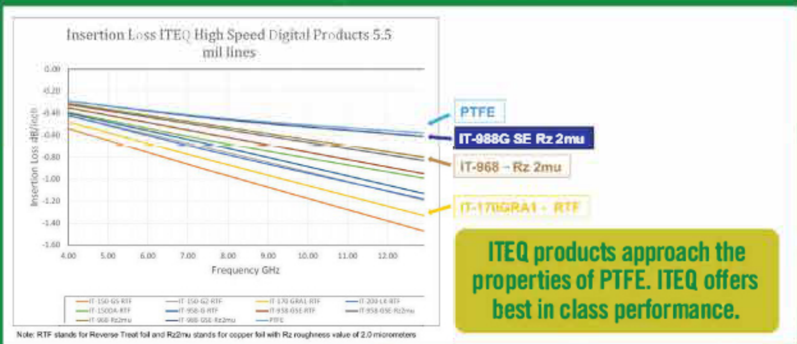
IT-968 / IT-968 SE

High Thermal Reliability
Designed for High Speed Digital Applications
- Telecom, Backplane, Base Station, Server, Storage, Switch

IT-968
Dk @ 10 GHz - 3.85
Df @ 10 GHz - 0.0043

IT-968 SE
Dk @ 10 GHz - 3.30
Df @ 10 GHz - 0.0019

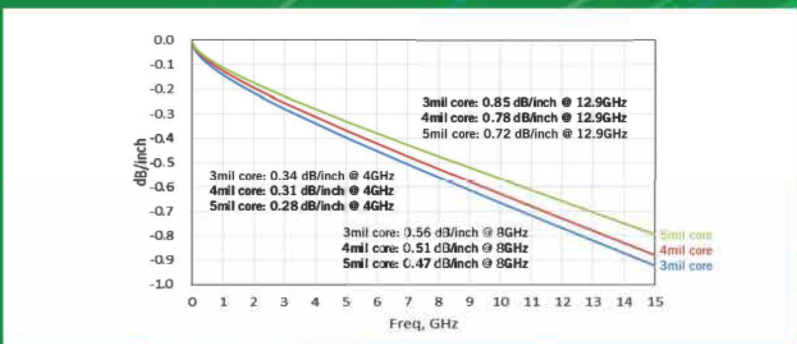
Insertion Loss - Measured



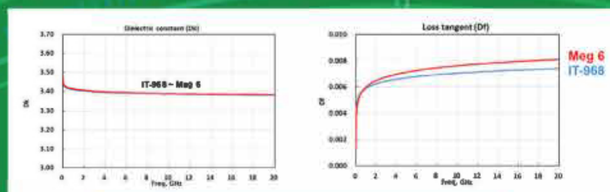
Industry Test Vehicle

32 layers, 0.140" thick
Four 2 oz copper internal layers
0.8 mm pitch, 9.8 mil drills
- Passed 1000 hours CAF, 10 V bias, 50 V
- 1ST - Passed 1000 cycles, 6x 260°C precondition
- No delamination after 8x 260°C reflow after 2 weeks at 35°C/ 85% RH

IT-968 Loss Performance



IT-968 SPP Comparison



GHz	10		12.9		16		20	
Parameter	Df	Dk	Df	Dk	Df	Dk	Df	Dk
IT-968	0.0071	3.39	0.0072	3.39	0.0073	3.38	0.0074	3.38
M6	0.0076	3.39	0.0078	3.39	0.0079	3.38	0.0081	3.38
%	7.95	0.04	8.52	0.04	8.98	0.03	9.45	0.02

Ultra Low Loss IT-968 SE

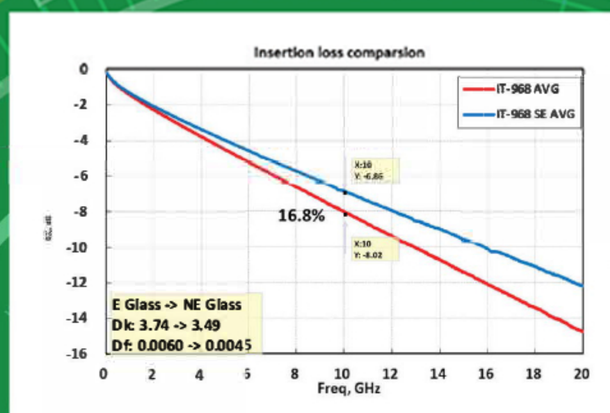
Features
High Tg, low CTE, High Thermal Reliability with Ultra Low Df < .005. Designed for High Speed Applications.

Applications
Server/Storage/Switch (10G/40G/100G), Backplane, Telecom, Base Station

Feature *High Speed , 25Gbs/path Solution*

Property	Test Method	IT-968	IT-968 SE
Tg (°C)	DSC	185	185
T-288 (w/ 1 Oz Cu, min)	TMA	120+	120+
Td-5%(°C)	TGA 5% loss	390+	390+
CTE (%), 50-260°C	TMA	2.2	2.2
Peel strength (lb/inch)	1 oz	6	6
Water Absorption	D-24/23	< 0.1	< 0.1
Dk, 1 GHz 2-10 GHz	IPC TM-650 2.5.5.9 IPC TM-650 2.5.5.13	3.4 3.4-3.7	3.4 3.4-3.7
Df, 1 GHz 2-10 GHz	IPC TM-650 2.5.5.9 IPC TM-650 2.5.5.13	0.0032 0.0036-0.0050	0.0028 0.0031-0.0040

E-Glass vs Low Dk Glass



ITEQ

Chinese Review: The 2018 NTI-100 Top Global PCB Fabricators

Feature by Yonglin Gong
CHINA PRINTED CIRCUIT ASSOCIATION (CPCA)
PRINTED CIRCUIT INFORMATION MAGAZINE

NTI released the list of the world's top PCB manufacturers in 2018, and this information is of great interest to the industry to determine what is going on in the industry and analyze the global situation. In the early days of this project, the list concentrated on the top 100 companies, so it continues to be called NTI-100 for short. Now, it is compiled into Chinese. I will examine the currency exchange rate, company rankings, global distribution of the top-ranked companies, global total PCB output, and the automotive sector.

Currency Exchange Rate

PCBs are produced all over the world, and the currencies are different from country to country. Therefore, the output value is calculated in U.S. dollars so as to unify and normalize the data. For reference, the exchange rate between the U.S. dollar and other currencies is shown in Table 1.

Currency	2014	2015	2016	2017
China yuan (RMB)	6.158	6.284	6.634	6.758
Japanese yen	105.86	121.06	107.84	112.93
Taiwan NTD	31.855	31.777	32.25	30.44
Korean won	1,053.58	1,132.33	1,160.80	1,130.59
Thai baht	32.482	34.253	35.290	33.92
Singapore dollar	1.276	1.375	1.440	1.334
Malaysia ringgit	3.270	4.120	4.100	4.32
Vietnam dong	21,137.07	21,920.68	22,763.00	22,721.03
Philippine peso	44.399	44.520	47.300	50.44
Indonesia rupiah	12,671.31	13,749.27	13,320.00	13,440.00
Canada dollar	1.104	1.279	0.997	1.297
Indian rupee	61.007	64.235	67.800	64.87
Mexican pesos	13.306	15.792	19.05	18.95
Russian ruble	38.512	61.195	57.4	58.31
Swiss franc	0.915	0.962	0.997	0.98
UK pound	0.606	0.655	0.74031	0.81
Euro	0.754	0.902	0.904	0.886

Comments in the table are this author's private view and may not be appropriate.
The author is fully responsible for any error which may exist. (www.x-rates.com)

Table 1: Average exchange rates used to convert local currencies to USD.

Top Company Rankings

PCB manufacturers listed in this ranking have annual sales of more than $100 million. This list of the world's top PCB manufacturers is ranked by their PCB sales in 2018 (Table 2).

In 2018, the overall PCB market was not as

Rank	Maker	Nationality	2017	2018	Growth
1	ZD Tech*	Taiwan	3,608	3,929	8.9%
2	TTM Technologies	U.S.A.	2,659	2,847	7.1%
3	Nippon Mektron*	Japan	3,283	2,704	-17.6%
4	Unimicron	Taiwan	2,155	2,513	16.6%
5	Tripod	Taiwan	1,519	1,728	13.8%
6	Mflex (DSBJ)*	China	966	1,725	78.6%
7	Compeq	Taiwan	1,789	1,685	-5.8%
8	HannStar	Taiwan	1,314	1,435	9.2%
9	Samsung E-M	S. Korea	1,279	1,348	5.4%
10	KB Chem PCB Group	China	1,040	1,237	18.9%
11	AT&S	Austria	1,175	1,218	3.6%
12	Young Poong Group*	S. Korea	1,746	1,217	-30.3%
13	Fujikura*	Japan	1,138	1,155	1.5%
14	Shennan Circuit	China	860	1,152	40.0%
15	Meiko	Japan	986	1,081	9.6%
16	Ibiden	Japan	1,051	1,054	0.3%
17	Wus Group	Taiwan	880	1,016	15.4%
18	Nanya PCB	Taiwan	883	956	8.3%
19	Simmtech	S. Korea	738	916	24.1%
20	Flexium*	Taiwan	857	888	3.6%
21	Daeduck Group	S. Korea	858	885	3.1%
22	Sumitomo Denko*	Japan	908	820	-9.7%
23	CMK	Japan	789	820	3.9%
24	LG Innotek	S. Korea	789	807	2.3%
25	Kinsus	Taiwan	740	787	6.3%
26	T.P.T.	Taiwan	746	765	2.5%
27	Kinwong	China	665	755	13.5%
28	BH Flex*	S. Korea	628	698	11.1%
29	Gold Circuit	Taiwan	637	683	6.8%
30	Shinko Denki	Japan	708	683	-3.5%
31	Chin Poon	Taiwan	788	670	-15.0%
32	Unitech	Taiwan	602	648	7.6%
33	Murata*	Japan	450	600	33.3%
34	Shenzhen Suntak	China	467	553	18.4%
35	Kyocera PCB	Japan	540	545	9.0%
36	Shenzhen Fast Print	China	497	525	5.6%
37	Career*	Taiwan	429	515	20.0%
38	Victory Giant	China	369	501	35.8%
39	SI Flex*	S. Korea	528	500	-5.3%
40	Isu-Petasys	S. Korea	483	497	2.9%
41	Ellington	China	496	489	-1.4%
42	Dynamic	Taiwan	385	432	12.2%
43	KCE Electronics	Thailand	426	418	-1.9%
44	Kyoden	Japan	398	401	0.7%
45	Wuzhou	China	336	386	14.9%
46	Founder Tech	China	389	383	-1.5%
47	Hitachi Chemical	Japan	381	375	-1.6%
48	CCTC	China	349	372	6.6%
49	APEX	Taiwan	345	370	7.2%
50	Nitto Denko*	Japan	382	362	-5.2%
51	Xiamen Hongxin	China	224	341	40.0%
52	Aoshikan	China	263	338	28.5%
53	Gul Technologies	Singapore	260	338	30.0%
54	Olympic	China	285	330	15.8%
55	DG Shengyi Electronics	China	285	320	12.3%
56	Guangdong Xinda	China	306	300	-2.0%
57	Sanmina	U.S.A.	330	300	-9.1%
58	APCB	Taiwan	290	296	2.1%
59	Bomin	China	266	295	10.9%
60	Redboard	China	283	293	3.5%
61	DAP	S. Korea	269	290	7.8%
62	CEE PCB	China	163	263	61.3%
63	ASE	Taiwan	280	260	-7.1%
64	Shirai Denshi	Japan	259	260	0.0%
65	Sun & Lynn	China	244	260	6.6%
66	Boardtek	Taiwan	227	252	11.0%
67	AbonMax (Palwonn)	Taiwan	230	250	8.7%
68	BYD	China	167	250	49.7%
69	Ichia*	Taiwan	238	240	0.8%
70	MFS*	Singapore	186	238	28.0%
71	Fujitsu	Japan	226	230	1.3%
72	Guangdong Chaohua	China	218	210	-3.7%
73	Kunshan Huaxing	China	202	204	1.0%
74	SDG Precision Tech	China	117	201	71.8%
75	STEMCO	S. Korea	199	200	0.0%
76	Kyosha	Japan	193	191	-1.0%
77	Somacis	Italy	181	187	3.3%
78	Delton	China	155	185	19.3%
79	AKM*	China	170	185	8.8%
80	Onpress	China	177	182	2.8%
81	KSG	Germany	154	182	5.2%
82	Lead-Tech	China	146	181	27.0%
83	Liang Dar	Taiwan	175	180	2.8%
84	Circuitronix	China	175	180	2.8%
85	Ji'An Mankun	China	154	180	3.9%
86	Zhuhai Kingsun PCB	China	124	173	39.4%
87	Shenzhen Sunshine	China	160	171	6.9%
88	Guangzhou Kingshine	China	171	171	0.0%
89	GZ Junya	China	149	169	13.4%
90	Guangzhou GCI	China	148	168	13.5%
91	Daisho Denshi	Japan	164	166	1.2%
92	Würth Elektronik	Germany	158	166	5.1%
93	OKI PCB Group	Japan	150	164	9.3%
94	New Flex*	S. Korea	137	164	19.7%
95	3Win Group	China	150	160	6.7%
96	Huading Group	China	149	154	3.3%
97	Schweizer	Germany	138	148	7.2%
98	Changzhou Haihong	China	144	145	0.0%
99	SZ Jove Enterprise	China	138	144	4.3%
100	Hyunwoo	S. Korea	125	141	12.8%

* Major products are FPC. In USD Millions (N.T. Information Ltd)

Table 2: NTI-100 top 100 global PCB fabricators by revenue (2018).

prosperous as in 2017, but it still remained in good status. The ranking of the world's top PCB manufacturers in 2018 didn't change a lot when compared with 2017, according to Table 2. The obvious change is the decline of Japanese companies. Nippon Mektron, which once ranked No. 1, dropped to No. 2 in 2017; in 2018, it fell further to No. 3. ZD Tech (Taiwan) and TTM (USA) now lead the list, respectively. In 2017, five companies among the top 10 were Japanese companies, while in 2018, only four Japanese companies still remained, and their rankings had fallen as well. Meanwhile, PCB companies in Mainland China show the largest growth, holding three seats of the top 20. One Chinese company, Mflex, has broken into the top 10.

In 2018, the output of the 117 listed companies ($62 billion) accounted for 83% of global output ($74.5 billion). Generally speaking, 20% of the companies in an industry will account for 80% of the sales. Considering that there are approximately 2,400 manufacturers worldwide, 117 manufacturers account for less than 5% of the total number of providers. Therefore, for the PCB industry, 5% of manufacturers are responsible for 80% of production.

REGION	# OF ENTRIES 2017	# OF ENTRIES 2018	TOP MAKER OUTPUT 2017	TOP MAKER OUTPUT 2018	2018/2017 Growth	2018 SHARE
Taiwan	25	26	19,564	20,980	7.23%	33.8%
Japan	21	18	12,149	11,736	-3.4%	18.9%
China	46	49	12,907	15,418	19.33%	24.8%
S. Korea	12	12	7,779	7,663	-1.5%	12.3%
USA	4	4	3,119	3,287	3.1%	5.3%
Europe	4	5	1,806	1,901	5.3%	3.1%
S.E.A.	3	4	962	1,097	14%	1.8%
Total	115	118	58,286	62,062	6.5%	100.0%

In USD Million (N.T. Information Ltd.)

Table 3: Summary of NTI-100 2018 by region, global PCB output.

REGION	2017	2018	YOY	2019F	YOY
America	3,037	3,158	4.0%	3,174	0.5%
Germany	960	994	3.5%	939	-6.0%
Other Europe	1,385	1,257	1.2%	1,270	1.0%
Africa & Middle East	142	143	0%	145	1.4%
WEST TOTAL	5,524	5,552	3.1%	5,528	0.4%
REGION	2017	2018	YOY	2019F	YOY
China	37,200	40,510	8.9%	39,880	-1.5%
Taiwan	7,685	7,780	1.2%	7,690	-1.2%
S. Korea	7,215	7,515	4.1%	7,214	-4.0%
Japan	5,625	5,654	0.5%	5,796	-1.2%
Thailand	2,980	3,132	5.1%	3,160	0.9%
Vietnam	2,620	2,704	3.2%	2,905	3.7%
Other Asia	1,738	1,668	-4.0%	1,700	1.9%
ASIA TOTAL	65,063	68,963	6.0%	68,345	-0.9%
WORLD TOTAL	70,587	74,515	5.6%	73,873	-0.9%

In USD Millions.
Production includes assembly operations, particularly flexible printed circuits. (N.T. Information Ltd.)

Table 4: World PCB production by region.

The Distribution of Top Companies in the World

Table 3 shows the number of companies and output values in different countries/regions that broke into the 2018 rankings. Among the 117 enterprises, 50 are from China, accounting for 42.7% of the total. Of course, the proportion of the total output value of Chinese mainland enterprises is not high, but the growth rate is the highest (19.33%). Southeast Asia is also growing faster. To avoid the impact of international trade disputes, the investment is turning to Southeast Asia. The growth rate of Taiwan is also higher than the average. The world's top PCB manufacturers (117) grew by 6.5% compared to the previous year in 2018, which is higher than the global PCB growth rate of 5.6% (Table 3).

The distribution of PCB output worldwide, according to NTI statistics, is shown in Table 4. NTI sets the premise for statistical data processing. If there is a business acquisition transaction completed in 2018, the seller's revenue will be fully incorporated into the purchaser's revenue reporting. For example, Multek's revenue will be merged into Mflex's. If there is no separate PCB business report for a diversified company, the report will be estimated.

Many PCB manufacturers are now also engaged in assembly. Since the data for bare board and assembly are typically not separated, the statistics also include part of assembly, especially the FPCB production value. This situation may tend to inflate the sales numbers as reported. For this type of manufacturer, the actual bare board revenue may be only 50% of the reported total. Some PCB manufacturers' income includes subcontracting or consignment value, which cannot be removed if it is calculated repeatedly (Table 4).

NTI data has always been higher than the common Prismark data. The process by which Prismark tabulates its data is not clear to me. For example, if you look at the global PCB market output in 2018, according to NTI, the market is about 74.5 billion USD, which is an increase of 5.6% over the previous year. Yet according to Prismark, the market is about 62.4 million USD, which is an increase of 6.0% over the previous year. The gap between the two analyses amounts to more than 10 billion USD. It is not clear which one is correct and/or which one is closer to the truth. In the industry, both of these data are cited, creating a sort of bounding box for the market. For the reference, while NTI and Prismark are different in how they tell the rise and fall of PCB market, the trends identified in both reports agree with each other. Having a grasp on the market trend is often more important in the industry than the actual numbers themselves.

According to NTI, PCB production in North America, Japan, and Europe, though very strong in the past, is now hitting rock-bottom, and that industry growth comes from overseas factories. PCB plants in Europe are investing, not to expand the scale, but rather to focus on technological upgrades to meet the needs of new products. According to the PCB output and the number of employees in Europe, the average annual per capita output in Europe is 115,800 euros (908,000 yuan), which shows that Europe is much more efficient than China. China's PCB production continues to expand and has been more cautious (but not halted) of late due to the beginning of the U.S.-China trade war. I know that China has at least 35–40 new large factories under construction. In a few years, when all of these new factories start to produce PCBs, China's production may account for more than 70% of global production, and Chinese manufacturers will occupy more of the top positions on the NTI-100 list.

Automotive PCB Manufacturers

Automotive has become an important market for PCB, accounting for almost 10% of the total. In the future, the total volume of the automotive market will not change much, even though the content of automotive electronic devices is expected to grow at a consistent pace. Consequently, automotive PCB manufacturers will see better performance. NTI lists the world's 20 largest automotive PCB manufacturers in Table 5, which are among the world's top PCB manufacturers.

RANK	FABRICATOR	NATIONALITY	TOTAL REVENUE	AUTOMOTIVE PCB
1	CMK Corp.	Japan	820	628
2	TTM Technologies	US	2,847	572
3	Nippon Mektron	Japan	2,704	560
4	Meiko Electronics	Japan	1,081	508
5	Chin Poon Industrial	Taiwan	670	470
6	Kingboard Chem GRP	China	1,250	370
7	Tripod Technology	Taiwan	670	470
8	KCE Electronics	Thailand	420	335
9	AT&S	Austria	1,218	275
10	PCB PCB Group	Taiwan	1,435	220
11	Wus Group	Taiwan	1,016	200
12	Unitech PCB	Taiwan	648	190
13	Dynamic Electronics	Taiwan	432	180
14	Unimicron	Taiwan	2,513	170
15	Kinwong	China	755	150
16	Gul Technology	Singapore	338	118
17	Kyoden	Japan	401	116
18	Nanya PCB	Taiwan	956	116
19	CCTC	China	368	114
20	Olympic PCB	China	330	111
	TOP 20 TOTAL		**21,930**	**5,754**

In USD Millions. (N.T. Information Ltd., May 2019)

Table 5: The world's 20 largest automotive PCB fabricators.

Panasonic

Under development

Halogen-free Ultra-low transmission loss Multi-layer circuit board materials

Halogen-free MEGTRON 6

NEW

Laminate R-5375(N)* R-5375(E)
Prepreg R-5370(N)* R-5370(E)
*Low Dk glass cloth type

Features

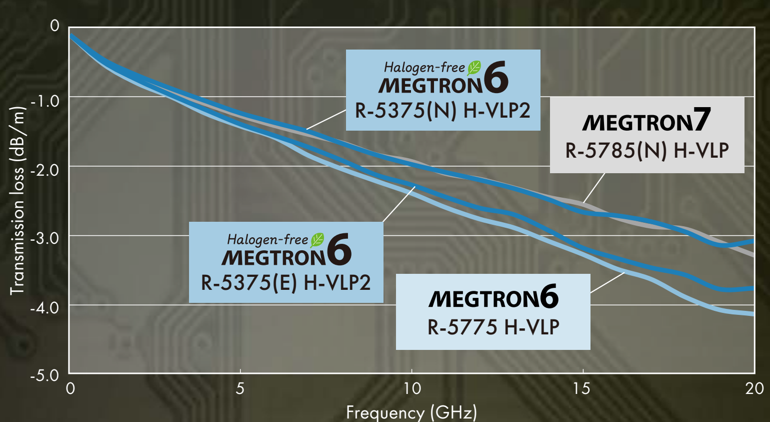
- Excellent HDI and thermal performance **with Halogen free**
- High speed and ultra-low loss material
- Low transmission loss

@10GHz	Dk	Df
R-5375(N)	3.3	0.003
R-5375(E)	3.8	0.005
R-5775	3.8	0.005

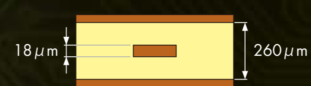
Applications

- ICT infrastructure equipment, High speed networking(High-end server/ router, Optical network, switch), High layer count PCB, etc.

Transmission Loss



- Construction



Line length	200mm , 100mm
Line width	125μm
Impedance	50Ω
Inner Cu treatment	No-surface treatment
Core	0.13mm
Prepreg	#2116 56% x 1ply

Contact us

More information

Current Outlook

2019 is the beginning of rollout and demand for 5G communications. The demand for 5G base stations, routers, servers, antennas, and other equipment will increase rapidly between 2019 and 2020. The installation of 5G communications infrastructure will benefit high-level, multilayer board manufacturers, and PCB manufacturers who can meet such demand will enjoy a spurt of progress. Of course, automotive electronics has become an indispensable market for PCB. The computer market, though it declines a little bit, is still an important part, and smartphones still account for more than 20% of the total PCB, even if they are growing slowly.

Right now, the PCB winners will be those companies who meet the needs of 5G infrastructure, automotive electronics, integrated circuit (IC) substrates, and smartphones. Income growth in other markets will be very limited. **PCB007**

Yonglin Gong is a senior engineer, deputy secretary-general of the China Printed Circuit Association (CPCA), and editor-in-chief of *CPCA Printed Circuit Information Magazine*. Mr. Gong has served the industry for many years. Previously, Mr. Gong was chief engineer of the 20th Shanghai Radio Factory, and he also helped translate Joseph Fjelstad's book, *Flexible Circuit Technology*, into Chinese.

The Future of Chips

The Singapore-MIT Alliance for Research and Technology (SMART), MIT's research enterprise in Singapore, has announced the successful development of a commercially viable way to manufacture integrated silicon III-V chips with high-performance III-V devices inserted into their design.

"By integrating III-V into silicon, we can build upon existing manufacturing capabilities and low-cost volume production techniques of silicon and include the unique optical and electronic functionality of III-V technology," said Eugene Fitzgerald, CEO and director of SMART. "The new chips will be at the heart of future product innovation and power the next generation of communications devices, wearables, and displays."

Kenneth Lee, senior scientific director of the SMART low-energy electronic systems (LEES) research program, adds, "With our new process, we can leverage existing capabilities to manufacture these new integrated silicon III-V chips cost-effectively and accelerate the development and adoption of new technologies that will power economies."

The new technology developed by SMART builds two layers of silicon and III-V devices on separate substrates and integrates them vertically together within a micron, which is 1/50th the diameter of a human hair. The process can use existing 200-mm manufacturing tools, which will allow semiconductor manufacturers to make new use of their current equipment. Today, the cost of investing in new manufacturing technology is in the range of tens of billions of dollars; thus, this new IC platform is highly cost-effective and will result in much lower cost novel circuits and electronic systems.

The patent portfolio has been exclusively licensed by New Silicon Corporation (NSC) Pte. Ltd., a Singapore-based spin-off of SMART. The new integrated silicon III-V chips will be available next year and expected in products by 2021.

(Source: SMART)

MilAero007 Highlights

Operational Excellence: Transform Your Operations With Nadcap ►
Obtaining Nadcap certification can be an overwhelming endeavor for suppliers that don't have proper guidance. In this column, Alfred Macha provides guidelines that companies have used in the past to successfully achieve Nadcap accreditation.

Arlon EMD and U.S. Air Force Academy Establish CRADA ►
Arlon EMD has entered into an agreement with the United States Air Force Academy to collaborate efforts in researching and developing technology platforms for electronic materials, resin systems, and copper-clad laminate products.

Amphenol Invotec to Exhibit at Space Tech Expo Europe ►
Amphenol Invotec will be showcasing its PCBs alongside the extended range of interconnect solutions available from Amphenol for the space sector at Europe's largest B2B space event—the Space Tech Expo in Bremen, Germany on the November 19–21, 2019.

Defense Speak Interpreted: Other Transaction Authority ►
At the end of 2018, DIU had funded 104 contracts with a total value of $354 million and brought in 87 non-traditional DoD vendors, including 43 contracting with DoD for the first time.

From the Hill: The Past 15 Years—Changes to MIL-PRF-31032 Certification, Part 2 ►
Mike Hill provides an overview of the possible related factors to what could have caused the reduction in certified companies, including a decline in the total military market; the cost of certification; and the number of military boards now built to industry standards, to name a few.

American Standard Circuits Recertified to AS9100D & ISO 9001: 2015 ►
American Standard Circuits received their three-year recertification to AS9100D and ISO 9001: 2015 from certifying body Intertek.

BAE Systems Demonstrates First Integration of Unmanned Surface Vessel With Royal Navy Warship ►
At a time of increased threat to international shipping, BAE Systems has demonstrated for the first time how unmanned surface vessels (USVs) can be fully integrated with operational Royal Navy warships to extend their reach beyond the horizon and reduce sailors' exposure to danger.

NASA Supports 'Wild' Ideas to Bring About New Space Tech ►
NASA has a program named NIAC that is dedicated to nurturing visionary ideas that could transform future NASA missions with the creation of breakthroughs—radically better or entirely new aerospace concepts.

Raytheon Unveils Peregrine Advanced Air-to-Air Missile ►
Raytheon Company is developing a new medium-range, air-launched weapon called the Peregrine missile that is half the size and cost of today's air-to-air missiles, yet delivers greater range and effect.

Marshall Aerospace and BAE Systems Collaborate on Next-generation Aerospace Technology ►
BAE Systems and Marshall Aerospace and Defence Group have signed a memorandum of understanding to collaborate on advanced technologies for various next-generation air platforms.


PASSPORT

The Laminate Market: What Will the Future Bring?

The PCB Norsemen

Feature Column by Raymond Goh, ELMATICA

PCBs have been manufactured more or less the same way since we entered the industry in 1972, but the circumstances surrounding the boards have changed. The PCB Norsemen have addressed the copper situation several times in our columns as well as the component crisis affecting the PCB industry. Now, we're experiencing external factors—such as Brexit and the trade restrictions between China and the U.S.—that are affecting the industry and causing delays due to raw material demand and prioritization by huge market players. The PCB market in China is experiencing a slower growth pace since the beginning of the year. Many factors have affected it, including the U.S.-China trade war and its increased tariff and the expectation of a big ramp-up from the automotive or electric vehicle (EV) industry, which did not happen this year.

Take a Step Back and Look at the Figures

We read and hear about factories moving out of China in view of the China and U.S. trade war.

There had been some increase in capacity in neighboring countries in Southeast Asia. However, in the next 3–5 years, China will probably continue to be the largest PCB manufacturing country, as the overall network of the supply chain for the PCB ecosystem that has been built and improved over the last decades will not be gone overnight.

Let's move back one step and take a look at the PCB laminate supplies. There had been significant growth in the global laminate value


EXTREME THERMAL CONDUCTIVE
SOLDER MASK
HIGH HEAT
BY TAIYO
PSR-4000 CC01SE
PSR-4000 HS2W / HS3W
TAIYO
AMERICA
Your Ultimate Solution
Phone [775] 885-9959 • www.taiyo-america.com • info@taiyo-america.com
in

between 2015 and 2016 from 24.3 billion USD to $25.8 billion. However, from 2016–2018, the growth had been unsubstantial with $26.2 billion and $26.5 billion in 2017 and 2018. It has been estimated that 2019 will close at about $26.7 billion.

This year, we cannot ignore the impact of the change in strategy of major players toward the supply chain. The U.S. trade restrictions imposed on Huawei Technologies, one of the world's leading networking and telecommunication manufacturers from China, have drastically affected Huawei's access to U.S. technologies, which includes raw materials. The U.S. trade restriction and warning to its allied countries have forced Huawei's products to face project bans, business contract restrictions, security scrutiny, and even pushback in some countries. Huawei continues to make changes to its strategy to continue doing business.

This year, we cannot ignore the impact of the change in strategy of major players toward the supply chain.

The Race Toward the First 5G Rollout

To wean itself off U.S. suppliers for related IC products, since June, Huawei has moved to redesign its 5G base station IC modules and to cut orders for CCL for use in their active antenna unit (AAU) from a major U.S. supplier while seeking alternative Asian supply sources. China has a huge domestic market and is also competing to be among the first to roll out 5G infrastructure. Huawei is the top provider in China in this aspect and had been ramping up circuit boards for 5G products.

Due to this surge in 5G demand, the PCB supply chain in Asia has been hit with a shortage of high-frequency and low-loss material for 5G. Major laminate suppliers are switching their production lines to produce such material to meet the demand. As a result, fewer FR-4 materials are being produced, which is what we are facing today. The shortage of high-frequency materials has been taken up by the major players. A longer lead time is needed for conventional glass-epoxy laminate FR-4 materials because its production has been reduced.

Copper Foil Shortages: Lessons Learned

After the lessons learned over the past few years about copper foil shortages, laminate manufacturers are quick to implement rules to control the situation today by implementing scheduled production, where a certain model of FR-4 is produced on certain days of the week only. This means there's no flexibility for PCB manufacturers to place an order and receive material with a short lead time. Laminate manufacturers are also defining allocation for PCB manufacturers, where the seller determines how much quantity the buyer can buy to prevent any single PCB manufacturer from taking all that has been produced.

And how do we get through this period? Again, planning, planning, and planning. Spend your time wisely. Plan ahead, make an accurate forecast, and involve your PCB supplier. You know you need to make a reservation in advance for very popular restaurants; why are you not doing the same for your PCB purchasing? We consistently need to remind ourselves that PCBs are not off-the-shelf products; they are customized and take time.

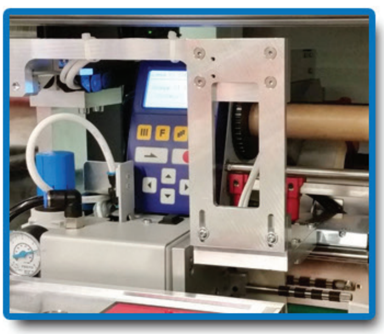
Conclusion

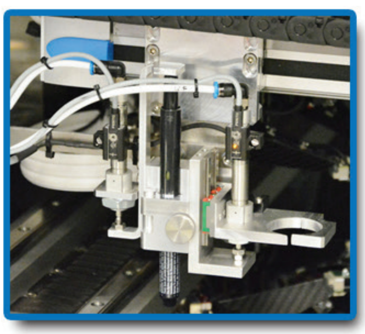
We know the PCB growth from now until next year will most likely be led by 5G products and the automotive segment. I hope this column has directed the spotlight toward the current situation and emphasized our recommendation of a timely disposition of PCBs to avoid unpleasant delays and surprises. **PCB007**

Raymond Goh is COO of Elmatica. To read past columns or contact The PCB Norsemen, click here.

Unrivaled Test Speed with Full Automation

Introducing the newly designed atg A8a with 8 test probes and a new high speed "lights out" automation for unrivaled throughput.





Highlights:

- Small footprint (6 square meters)
- Dual shuttle pick & place automation
- High accuracy combined with high test speed
- Pen or label marking option

Watch video

Get more info

atg Luther & Maelzer GmbH
Zum Schlag 3 • 97877 Wertheim • Germany
Phone +49-9342-291-0 • Fax +49-9342-395 10
klaus.koziol@cohu.com • www.atg-lm.com

atg®
LUTHER & MAELZER®
A Cohu Company

Multilayer Press Technology Using Magnetism to Produce Lamination Heat

Article by Víctor Lázaro Gallego
CHEMPLATE MATERIALS

Abstract

A revolutionary concept in multilayer press technology has been developed that uses electromagnetic energy to heat the existing stainless-steel separator plates with a never-before dreamed-of accuracy and precision. The heating and cooling systems—embedded within a robust hydraulic press inside a vacuum chamber design—are controlled using a temperature feedback loop that guarantees perfect fidelity between the press recipe and the press result. This unique, electromagnetic induction technology helps make engineer's needs become a reality.

Introduction

The production of the first four-layer multilayer dates to 1960, which means the industry has manufactured multilayer PCBs for more than half of a century. Everyone involved with the PCB industry knows that the technical evolution of multilayer PCBs has been exponential from the first technical requirements to today's. Current PCB requirements bring, collaterally, limits to the industrial-level manufacturing of state-of-the-art technologies.

The technical demands of modern PCBs have revolutionized almost all manufacturing processes from the multiple and complex chemical processes to mechanics, such as drilling, through the photoimaging of the circuit. However, if you look at the lamination process to cure today's sophisticated resin composites, you can see that the evolution of the lamination presses has been slight in comparison. While today's lamination presses are far better in many aspects than presses 59 years ago, press technology still uses the same method to produce the thermal energy (heat) and deliver the thermal energy that the resin composite needs to properly polymerize.

I want to introduce a new approach that uses a different way to generate thermal energy—and a new way to deliver it to the resins—that improves the lamination process for today's materials requirements and new requirements in the future. While initially designed for multilayer PCB manufacturing, this technology can also greatly benefit laminate manufacturers, opening up new avenues of research into

high-temperature composites and/or significantly shortened lamination cycles.

Technology Description

The InduBond lamination press utilizes a unique and novel way to produce the necessary heat to cure the laminates and multilayers—electromagnetic inductance. The standard and well-known press methods use one of the following technologies to generate the necessary thermal energy:

- Heating thermal oil and pumping it through a press platen
- Electrical heaters located right at the press platen
- A steam system through the press platens

All of these well-known systems heat the large thermal mass of the platens. The heat is then transferred to the panels being laminated via conductivity through the resin materials (i.e., the multilayer books to be laminated).

The novelty of this new technology, however, is that the thermal energy (heat) to cure the resin composites is produced directly at each of the stainless-steel separator plates that are between each multilayer panel in the press stacks. This thermal energy is transferred at the same time—with the same temperature magnitude and without any thermal conduction delays—to every panel of the lamination press stack. As the energy is induced very homogenously, the heat distribution has the highest uniformity possible in every position and direction of the press stack (X-, Y-, and Z-axes). Therefore, all of the layers of laminates inside the press reach the same temperature at the same time; there are no thermal transfer delays.

To make this possible, the laws of magnetism are used to induce high electrical currents that are transformed into thermal energy right at the material requiring polymerization. Because the heat is produced only at each stainless-steel separator plate of the stack, the technology can achieve extremely high temperatures and very rapid ramp-up rates with very high energy efficiency.

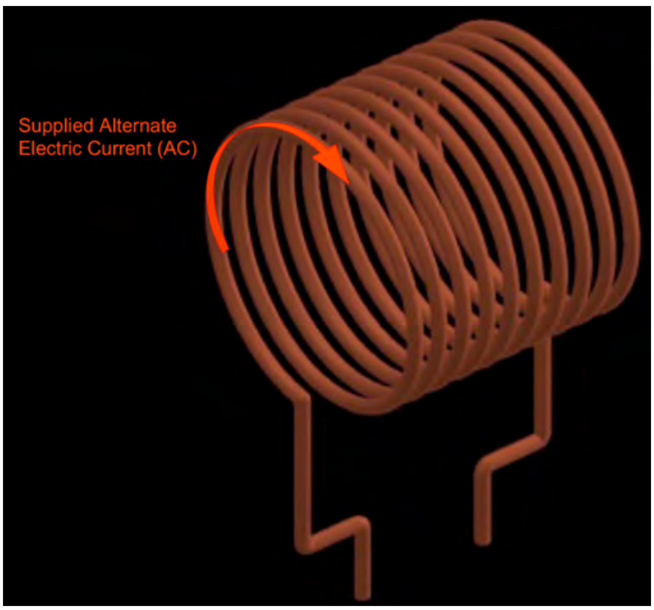


Figure 1: Electric coil.

Principle of Functioning

Magnetism is a very complex branch of physics since it cannot be fully explained by postulates of classical mechanics. For that reason, a very simple and visual explanation is used to introduce the principle of functioning of the InduBond electromagnetic inductance technology.

Electromagnetism is generated when electric currents move with uniform speed through an electrical coil, as defined by the Biot-Savart Law [1] (Figure 1). A magnetic field generated on the inside of the coil and looping around the coil perpendicular to the current in the coil is associated with the electric current in the coil (Figure 2).

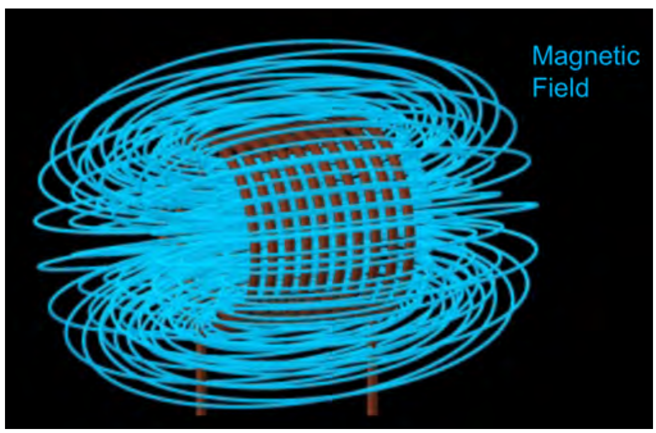


Figure 2: Magnetic field lines produced by the electric coil.

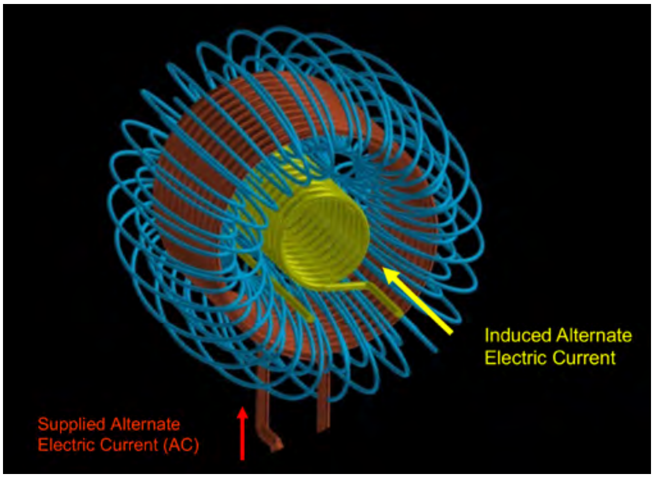


Figure 3: Secondary electric coil aligned to the primary coil.

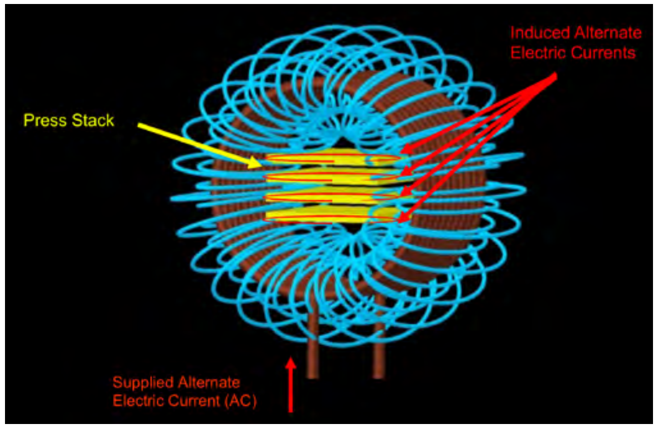


Figure 4: Press stack separator press plates as the secondary coil.

This phenomenon also works in reverse. In the same way a magnetic field is created by an electric current moving in a coil, an electric current can be made to flow in the presence of a magnetic field. In other words, a magnetic field can induce electric currents in a secondary coil if the magnetic field is placed in proper alignment to the primary coil (Figure 3).

Now, if we look at what would be the electrical function of a simple metal plate properly aligned with such a magnetic field, then that metal plate is working as one turn coil in a short circuit for the induced currents. The metal plate previously mentioned refers to the stainless-steel separator plates of the press stack (Figure 4).

The InduBond X-Press Technology has been developed to convert such energy induced on each stainless-steel separator plates (induced electric energy) into thermal energy (heat) by the eddy-current loss theory [2]. The resulting magnetic field generated by the coil is proportional to the main coil geometry seen in Figure 2.

The stack of stainless-steel plates and panels to be heated in the press can also be considered as proportional to the magnetic field because the stackup contains the same size and thickness of separator press plates throughout. Therefore, induced energy is proportional and uniform, generating the same amount of heat on every separator press plate of the stack.

A temperature sensor introduced into a dummy panel in the center of the press stack provides real-time feedback of the temperature of

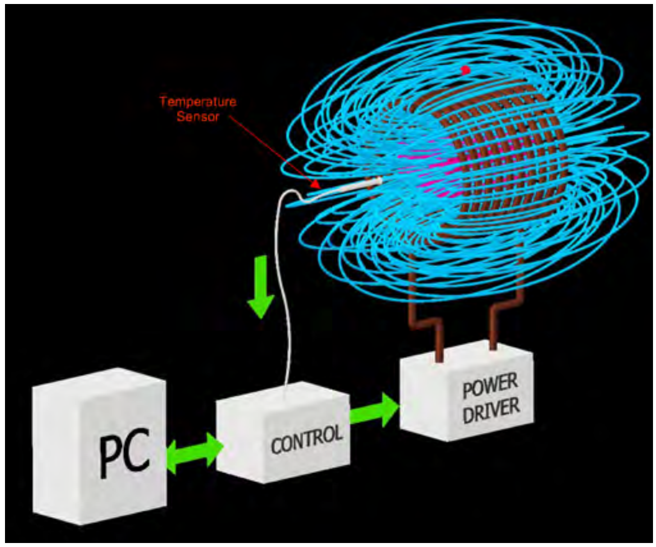


Figure 5: Close-loop control regulation system diagram.

the material. A closed-loop regulating control system uses real-time information provided by the temperature sensor (in concert with other feedback systems of the press) to constantly update the temperature, vacuum, pressure, and power control algorithms. A heating profile can now be programmed on a computer for the control algorithms to track the desired power delivered to the press stack to maintain or control the temperature as desired in real time (Figure 5).

The New Concept of Heat Transfer

This InduBond technology does not heat the big mass of the platens; the platens in this technology are cold and truly thermally isolated from the machine's heavy structure. The magnetic field lines of the magnetic field generated by the coil penetrate the stainless-steel separator plates, inducing high electrical currents that generate heat on each single separator plate of the press stack. That happens without any electrical or physical contact (wireless), thermal oil, or steam—just induction.

The generated heat on each stainless-steel separator plate is transferred immediately to the laminate in contact above and below. This internal and quick thermal transfer makes possible a no-delay energy transfer throughout the entire stack height while the thermal inertia is very low. A dummy panel with exactly the same material and physical properties as the laminate and with embedded temperature sensors is used to track the temperature of the material to be laminated in real time. No parts other than the stainless-steel separator plates are actively heated, which results in high electric efficiency; the heat is created internally on the press stack (Figure 6).

In conventional lamination press technologies where the heavy mass of the platens is heated up to then transfer that thermal energy from the platens to the panels (from outside to inside), an indirect source of energy is needed to heat up those big-mass platens. Conventional technologies typically use thermal oil, electrical heaters, or steam.

For thermal oil systems, the platens become mechanically complex due to the need for channels where the oil must flow to pump the heat into the platens. For electrically heated systems, many ohms of resistance at high power are required to exchange heat via these embedded cartridge heaters into the big-mass platens.

Both methods require huge amounts of power to heat up the platens that are mechanically connected to the main structure, causing high heat conduction losses. Because the heat is created in the external platens, the thermal energy has to travel by conduction from external platens to the material, passing through all separator plates and laminates. Dielectrics are inherently low thermal conductivity materials, which results in thermal conductivity delays (Figure 7).

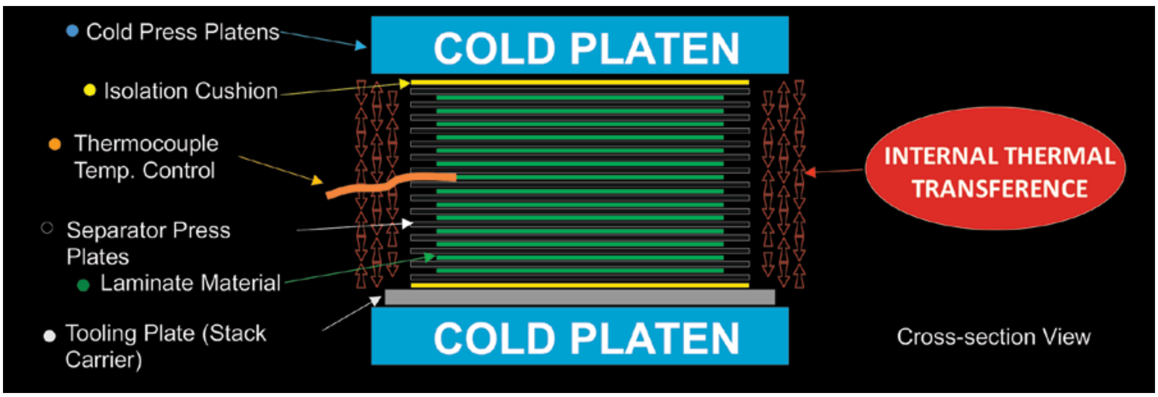


Figure 6: InduBond X-Press schematic cross-section.

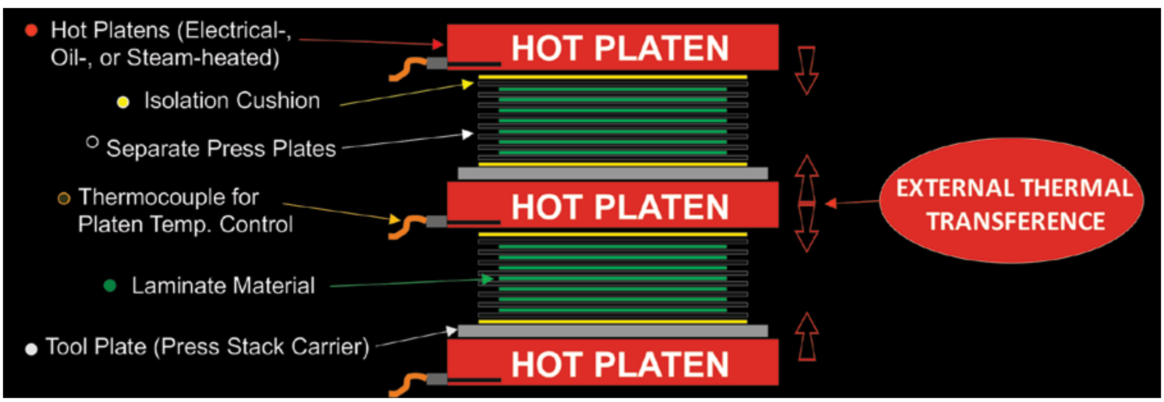


Figure 7: Conventional press technology schematic cross-section.

To make a realistic image of what is explained here, we took thermography on one of the platens from conventional press technology, and another thermography of the new magnetic press technology (Figure 8). The left side shows the hot platen heat distribution where the platen surface and the sides (mechanical links to the press guides) are hot while the center (press stack) is cooler and needs more time to transfer the heat through the press stack materials. The right side shows that platens are at almost ambient room temperature while the center of the press stack is hot.

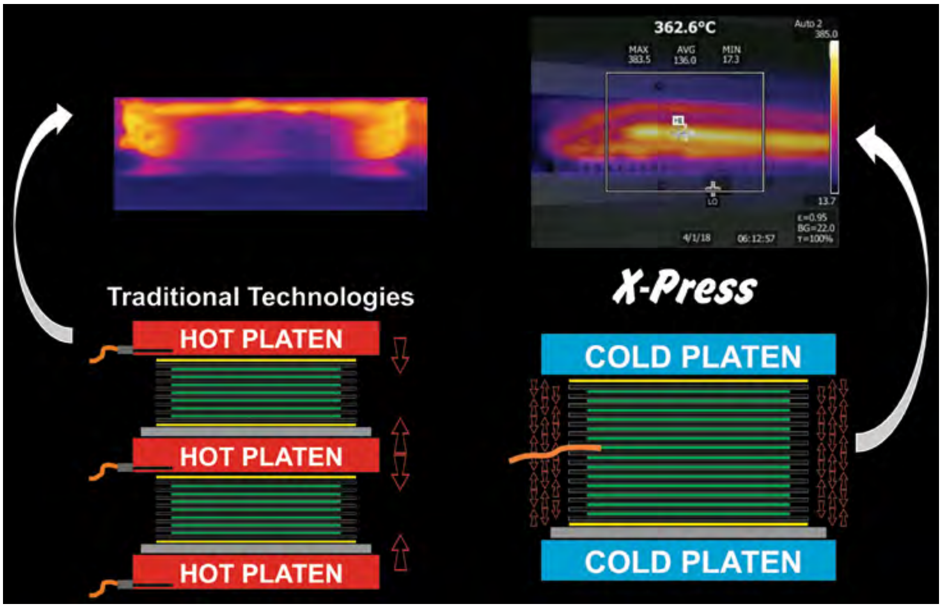


Figure 8: InduBond electromagnetic press technology versus conventional technology.

Energy Efficiency

This technology not only delivers very uniform temperature in the X-, Y-, and Z-axes of the press stack, but there is also another important point to be emphasized. The energy efficiency of this technology is very high compared to conventional technologies. The explanation for this is that the traditional systems heat up large metal masses so that these large masses can transfer the energy to the material to be heated, which has much lower mass by comparison. To give you an idea of this performance, a representative example was prepared that attempts to quantify the amount of energy needed to heat up the platens of a single-opening conventional press as compared to the electromagnetic inductance system (Figure 9).

The necessary energy to heat up a given mass is defined by the equation:

$$Q = \Delta t \cdot \mathrm{Mass} \cdot \lambda$$

Where:
Q = energy (J)
Δt = thermal increase (Kelvin)
Mass = mass to heat up (kg)
λ = specific heat (J/kg·°K)

The traditional or conventional system heats up both M1 platens to transfer the temperature to M2 and M3 through isolation papers (yellow color) to prevent thermal shock. The electromagnetic inductance system heats directly on-

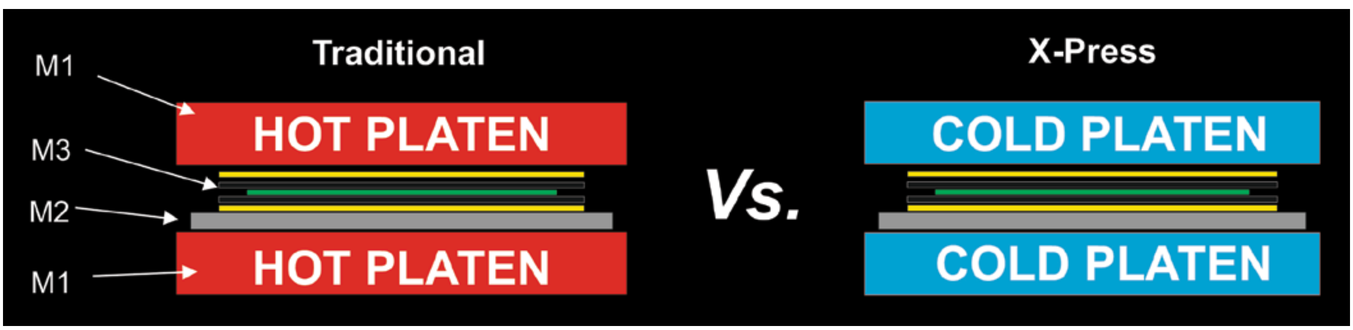


Figure 9: Energy comparison diagram.

to the M3 (the separator plates). The amount of energy needed, under ideal conditions, to heat both systems from ambient temperature 25°C (77°F) to 220°C (430°F) is shown here: M1 (platen mass) is ~70 kg; M3 (mass of stainless-steel separator plates) is ~4 kg; the specific heat of iron is 0.45 KJ/kg·K.

For the traditional technology system:

$$Q = (220°C - 25°C) \times (70\ kg + 4\ kg + 70\ kg + 4\ kg) \times (0.45\ KJ/kg{\cdot}K) = 12{,}987\ KJ$$

For the electromagnetic inductance technology:

$$Q = (220°C - 25°C) \times (4\ kg + 4\ kg) \times (0.45\ KJ/kg{\cdot}K) = 702\ KJ$$

In this example, the traditional method uses 18.5 times more energy than the electromagnetic inductance system. Plus, this example does not consider other large real losses like the energy transferred to other unnecessary large masses in physical contact, such as the links of the platens and guides to the main press structure.

Preparation of the Lamination Press Stack

Because this electromagnetic induction technology indirectly heats up each individual stainless-steel separator press plate—without the large thermal transfer delays of conventional technologies—it is not necessary to make a multi-opening press, although this is possible. This technology allows one to place all panels in a single-opening press.

The lamination press stack is built alternating conventional stainless steel separator plates with the laminate material, as in the well-known standard process. There can be just one panel in the opening or approximately 30 panels in the whole opening capacity (Figures 10a and b). The separator press plates are the same standard stainless-steel separator plates used in a traditional lamination press.

During stackup, the operator places a dummy panel with a minimum of 2-4 thermocouples for the temperature feedback control system in the middle of the stack. Those temperature sensors will monitor the real temperature of the laminate so that the control system can follow the programmed temperature profile (Figure 11).

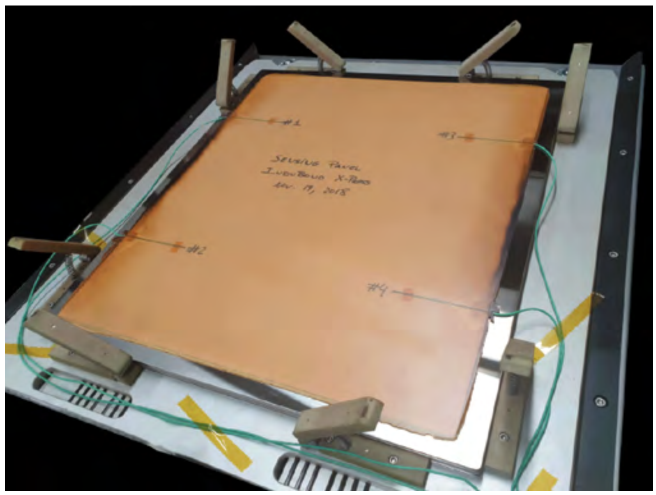


Figure 11: Dummy panel with the four thermocouples embedded.

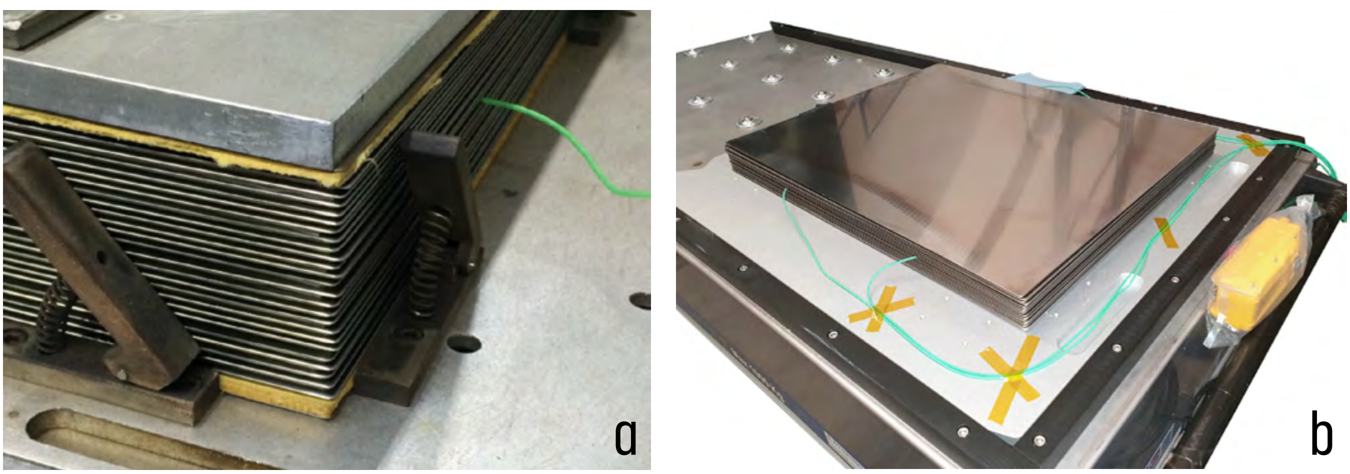


Figure 10a and b: Lamination press stack with conventional stainless-steel separator plates.


Excellon
Manufacturers of precision Micro-Machining Solutions for over 50 Years
COBRA II
Hybrid laser system
689
6-station precision drilling and routing system
HS-1L DS
Single-station, dual spindle high-speed system
136L
High accuracy and speed
154L
Large panel applications
HS-2L
2-station, high speed
Move into the future with the latest innovations in Fabrication Technology from the Industry Leader!
www.excellon.com

Figure 12: InduBond X-Press software screenshot.

Lamination Profile Programming

Like any other modern technology, this technology is managed by a standard computer with dedicated software where the process engineer or operators can set all press parameters. The lamination press profile is normally composed of the combination of three different profiles (magnitudes) in a unique timeline: temperature, pressure, and vacuum (Figure 12).

Regarding the control of this technology, the big difference is that the real temperature of the material in the lamination process is monitored in real time by the control system and used as a feedback for the closed-loop control system. In much simpler words, the process engineer or operator creates a temperature profile following what the laminate or prepreg manufacturer recommends on the datasheet of the material. Once this profile is transferred to the InduBond system, the computer, control, temperature sensors, and power driver work together to make the real temperature of the material track the programmed temperature profile very precisely (Figure 13).

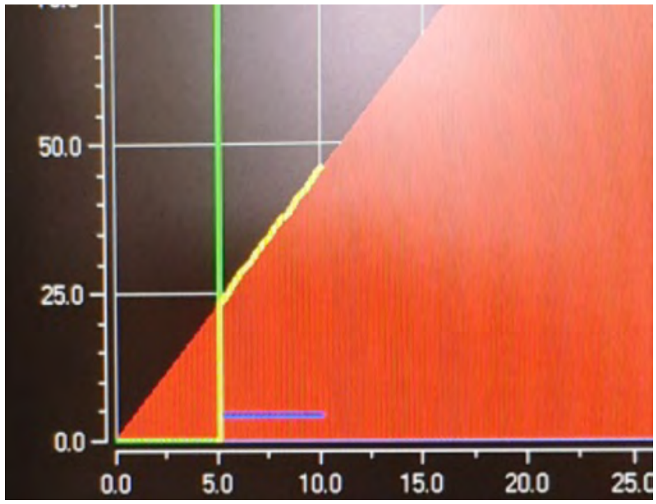


Figure 13: Zoom-in detail of the material temperature tracking the programmed temperature profile.

The software does the same for all other magnitudes, such as vacuum, hydraulic pressure, lamination pressure, and all other process parameters. All of the data during the lamination process is logged in a database and associated with a dedicated work order. The data can be retrieved at any time to generate process parameters reports for traceability purposes. An actual lamination cycle of about three hours can be seen in the reference link; a compressed version (approximately one minute) is available for those interested [3].

Cooling Down the Press Stack

After 20+ years' involvement in multilayer registration and lamination, I've heard multiple discussions about mechanical distortions of the panels after lamination—effects such as twist, camber, or simply uneven distortion between the panels laminated in the same opening with apparently identical process parameters. All of the engineers with whom I have discussed this topic mainly relate these phenomena to the thermal process that the material undergoes during lamination, and all engineers share and agree that the cooling speed and thermal balance of the different panels in the stack play an important role.

Most engineers would also agree that the ideal way to cool down a press stack would be by maintaining the lamination pressure while the materials slowly dissipate the thermal energy naturally through the environment, exchanging the energy with the ambient temperature. However, this process is nonproductive, industrially speaking, for obvious reasons; it takes a very long time and is far too slow.

Analyzing that thermal dynamics statement, and looking into the lamination press stack morphology in such "ideal" cooling conditions in detail while the material is cooling down naturally through the environment, the thermal energy (heat) travels from the center of the material, homogenously, to the edges of the material. This happens because the stainless-steel separator plates have far lower thermal resistance than the laminated multilayers (metal versus glass and resin). This means that the edges of those metal plates in contact with the air of the environment (i.e., room temperature) are exchanging the thermal energy with the environment until, after a long time, the temperature of the whole press stack becomes balanced with the ambient temperature.

I have used finite element calculation software to model this behavior to simulate this thermal dynamic. A link is available that shows a fast-motion video of a single metal plate cooled down from 200°C to room temperature [4]. The blue color is the ambient temperature, and the white color is the separator press plate at 200°C.

In conventional technologies, cooling down occurs in the same way the thermal energy was conducted from the hot platens to the laminate during heat up; the platens are cooled down first, and the thermal energy stored in the materials has to be conducted back from the stack to the platens. This means a significant thermal delay between different panels in the press stack as well as significant variation from panel to panel.

The cooling system developed for this technology emulates the natural way of cooling down previously described but boosted strategically to be able to speed it up for an industrial purpose: cooling down within the limits of the resin manufacturers' requirements. This method of cooling down also minimizes the temperature delays in the height of the press stack.

Once the heating section of the cycle is over, the vacuum chamber is pressurized. The developed cooling system creates a specific loop of airflow with a specific ΔT that crosses through the edges of the press stack separator plates. When the cold air touches the separator stainless steel, energy is exchanged. In other words, the air takes heat from the plates increasing the air temperature while the separator plates become a bit cooler. The air is then pulled by the blowers and conducted to a water/air exchanger that cools down the air again before the blowers push it back to pass through the press stack again. The system recirculates in a closed-loop manner the air inside the chamber.

The system uses the real material temperature monitored by the temperature sensor in the middle of the press stack to regulate the airspeed (blower speed) and the air temperature with the water circuits of the water/air exchanger (Figure 14).

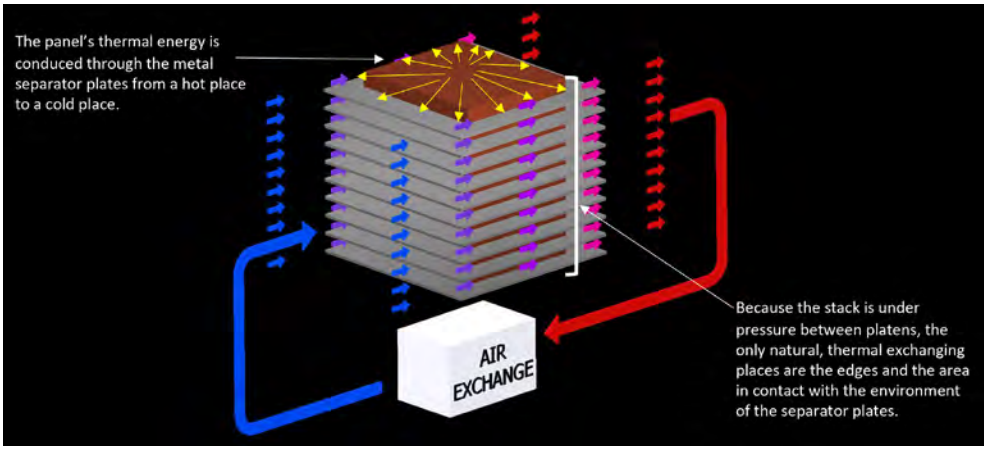


Figure 14: Closed-loop cooling system diagram.

Lamination Results

The InduBond X-Press Technology was developed a few years ago and has been working in production since the day of installation to fully validate the process. Many panels have been laminated and tested by a local customer in Barcelona, Spain. It has taken a long time, but this allowed us to collect a great deal of data from each lamination cycle and from the laboratory analyses of laminate suppliers, such as Nanya, Isola, Panasonic, Rogers, and others. They have all reported good lamination results, as can be seen in the examples shown in Figures 15–18.

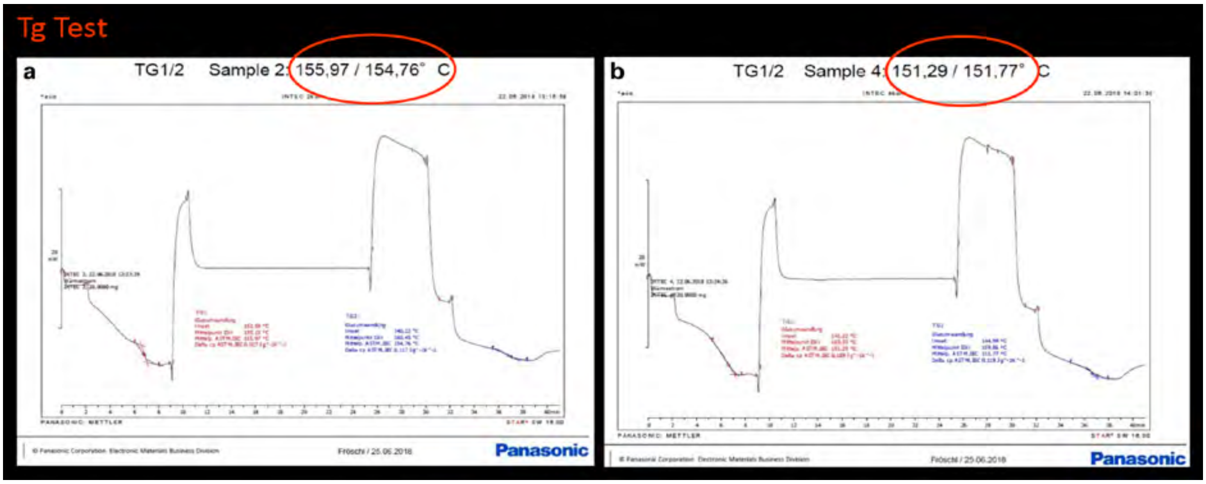


Figure 15a and b: Sample Tg test measurement tests.

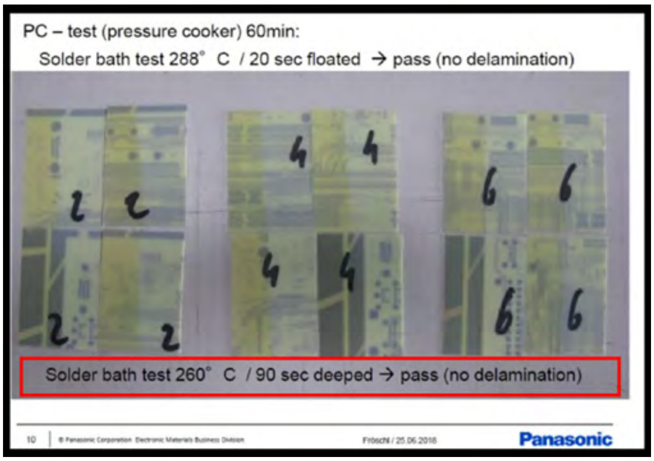


Figure 16: Mechanical stress test.

Figure 17: Thermal stress test.

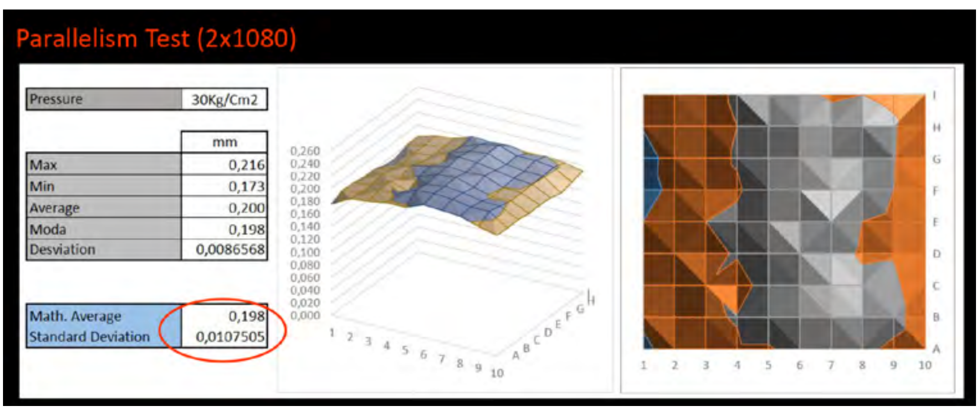


Figure 18: Parallelism measurement.

Summary

Table 1 provides a comparison of the InduBond electromagnetic inductance technology with conventional lamination technologies.

Point	Conventional Technology	Electromagnetic Inductance Technology
Heat Transfer Efficiency	Must heat up the platens (large thermal mass) to transfer temperature from the outside to the inside of the press stacks by thermal conduction. This is true no matter which heating method the standard technologies use (electrical heaters, thermal oil, or steam). As a consequence, the material in contact with the hot platens will have a different temperature profile than the inside of the press stack as there are big heating delays. The large mass of the platens results in a very high thermal inertia that complicates the thermal dynamics to be dealt with.	Induces heat directly into the separator plates, which is a much smaller mass compared to the platens. Plus, they are located right next to the material to be laminated, resulting in low thermal inertia, no thermal delays, and much-improved temperature uniformity along the three axes (X, Y, and Z) of the press stack. Heat is induced equally by the electromagnetic field in every separator plate simultaneously, which ensures controlled, real-time temperature transfer through the entire press stack.
Temperature and Ramp-up Limitations	Standard systems using thermal oil or heaters are faced with big limitations in reaching high temperatures at high heating rates.	Induces the heat directly into the material to be laminated with direct and immediate thermal transfer. And with a much smaller mass to heat up, it is possible to reach higher temperatures (500°C, or 932°F) and very high heating rates (>20°C/min, or 36°F/min).
Accurate Measurement and Control	Temperature control is based on the thermal transfer of energy from outside (big mass platens) to the inside of the press stack through the highly thermally isolating materials (panels). As a result, temperature control in the panels becomes a question of an empirical or experimental process that can lead to dimensional distortions and lamination quality defects in the panels.	Uses a temperature sensor inside of the laminating panels that allows real-time temperature measurement and direct energy delivery (no delays) to the panels. The lamination profile is configured in the software (temperature, pressure, vacuum, and time). The InduBond X-Press will apply the same profiles to each panel of the press stack at the same time—not an empirical or experimental process.
Energy Efficiency	Poor energy efficiency because the system heats up the very large mass of the platens that are mechanically (and thermally) linked to the whole heavy machine structure, resulting in a huge loss of energy to the environment.	High energy efficiency because the energy is delivered right to the materials without heating up the big mass platens; thus, there is no energy leakage.
Footprint and Environmental	Standard systems occupy large spaces. Heaters and their connection and control systems (piping) require space for installation and safety. A large industrial infrastructure accompanies oil heaters, and they require large amounts of oil that must be heated and held in reserve for use on demand.	InduBond X-Press Technology dramatically reduces the floor space requirements and is a clean, "green" technology that requires no oil.

Table 1: Summary of the differences between standard heating methods and the electromagnetic inductance-based system.

Applications and Technology Potential

This new way of producing the energy directly into the materials to cure or polymerize the resins overcomes the limitations of conventional industrial ways of laminating materials. InduBond X-Press Technology shows great potential that can be explored in four different areas:

1. More uniform, repeatable, and controllable lamination of the standard multilayer materials (FR-4, polyimide, CEM-3, etc.). The advantages include high-quality, reliable, and repeatable production; work flexibility; energy and investment savings; and a reduced floor surface.
2. Fast cycle lamination that could speed up prototyping with very short press cycle times. Advantages include high thermal increases (ΔT); high-productivity, high-quality, reliable, and repeatable production; work flexibility; energy and investment savings; and a reduced floor surface.
3. Advanced materials areas, such as high-temperature or high-pressure parameters. Advantages include high thermal increases (ΔT); high temperatures; high-productivity, high-quality, reliable, and repeatable production; work flexibility; energy and investment savings; and a reduced floor surface.

4. R&D of new advanced materials can be now industrialized. Today's scientific community will surely need better features and performance from new advanced materials. Coming technologies will stretch the boundaries of thermal, dielectric, high-frequency, and dimensional performance in ever-more extreme conditions.

Lab results are relatively easily achieved, but those same results must be replicated by industrial production processes that are scaled up in size and productivity to meet real product demands. Our idea is to introduce the InduBond X-Press Technology to the market today because we believe its features can help to design and industrialize tomorrow's materials for the most difficult operating and environmental conditions. New composites can now be cured or fired to very high temperatures, and the cycles can be shortened due to very high-speed rates. With InduBond X-Press Technology, it is possible to treat sintered materials in large dimensions. **PCB007**

References

1. Biot-Savart Law
2. Eddy current loss
3. Compressed video of a lamination cycle
4. Compressed video of a cool-down cycle

Víctor Lázaro Gallego is the R&D and technical director (CTO) at Chemplate Materials S.L. in Barcelona, Spain, an equipment designer and manufacturer for the PCB and FPCB industry that specializes in layer-to-layer registration, bonding, lamination, and automation. He is also the inventor of a few inductive bonding technologies with the trademark InduBond®. Contact Víctor Lázaro at v.lazaro@chemplate.com.

Designing and 3D-printing a Better Brace

Almost exactly one year ago to the day, Anuj Thakkar was in an unfortunate biking accident and left with a broken wrist. It soon became evident that not only would he have to deal with the pain of a broken wrist but would also have to deal with the discomfort of a cloth cast. After a couple of weeks of dealing with the consequences of a sweaty cast that could not get wet during one of the rainiest weeks of the month, Anuj had had enough.

Luckily for Anuj, he was a junior in mechanical engineering at the time, and he became determined to improve his situation. That's when Anuj decided to approach his friend Atanaz Bohlooli, a mechanical engineering and teaching assistant, to ask if she would be interested in collaborating on a project to engineer a wrist brace to alleviate the pain and discomfort. From then on, the two worked together to create a custom-fit, flexible, waterproof wrist brace to be 3D-printed for Anuj to wear in place of his initial cloth cast.

After about three months of design, modification, and testing, the two created a final product. By using a white-light scanner, 3D CAD software, and a Stratasys 3D printer, the wrist brace was durable and ready to be worn. The cast turned out to be a huge success, even garnering doctor approval, which allowed Anuj to wear the wrist brace for the next two months. One fully healed wrist, and a lot of learning later, Atanaz and Anuj continue to pursue engineering projects and push the barriers of conventional engineering.

(Source: Duke University)

Technology and Reliability Demands Drive Designers and MIL-PRF-31032 Specification

From the Hill

by **Mike Hill,** MIL-Q-CONSULTING LLC

The branches of military services continue to counter threats by shifting from boots on the ground to more technical solutions that do not require additional personal. The combat environment requires designs that require faster decisions and more rapid processing, rugged displays to withstand 120°F outside temperatures, extreme cold, blowing sand, and abnormal mechanical forces like the shock and vibration in a helicopter; they have to be smaller, lighter-weight, and extremely reliable.

Unmanned vehicles require more AI and associated technology. Electronic uniforms compliment camouflage and drive designs using electronic fabrics. For example, uniforms with cameras on the back of the soldier which project that image on the uniform front make the solder totally invisible (the ultimate in camouflage).

Increased electronic guidance for more projectiles—including bullets, space warfare countermeasures, and cyber threats—push new technology and drive printed circuit features and their associated reliability. With the future demand for more and more military electronics, certification to the PCB MIL-PRF-31032 specification becomes a business decision for many fabricators.

Fluency in the MIL-PRF-31032 language is a key first step to understand the requirements and communicate with the Department of Defense (DoD). This column will define many of these terms and is a must-review before informing the DoD of your intent to certify. Let's

take a look at these unique new words, including expanded definitions, and a quick-glance glossary.

MIL-PRF-31032 Overview Terms

1. MIL-PRF-31032: The military specification for unpopulated PCBs. It is a combination of technical requirements from MIL-P-55110, MIL-P-50884, and IPC-6012; the management systems from ISO: 9001; and specific reporting information about facility and process changes and general summaries of military shipments.

2. Defense Logistics Agency (DLA): The DLA in Columbus, Ohio, reviews the Technical Review Board's (TRB's) semiannual reports, administers web pages that contain the facilities qualification by material type and board complexity, maintains configuration management for military specifications, and audits each certified facility every three to four years.

3. MIL-PRF-31032 Qualifying Activity: The qualifying activity for MIL-PRF-31032 is communicating with the DLA.

4. The Commercial and Government Entity (CAGE) Code: Is obtained by completing a System for Award Management Registration (SAM) and necessary to receive any type of payment from the federal government, and is a five-character ID number used extensively within the federal government assigned by the DoD's DLA. The CAGE Code supports a variety of mechanized systems throughout the government and provides a standardized method of identifying a given legal entity at a specific location. Agencies may also use the code for a facility clearance or pre-award survey. There is no cost to obtain a CAGE Code.

5. Pre-validation: The manufacturer notifies the qualifying activity of its intent to pursue qualification to MIL-PRF-31032, starting a dialogue between the two entities crucial to the success of the effort.

6. Validation: A qualifying activity (the organizational element of the government that grants certifications, ergo DLA) will determine an organization's adequacy and compliance with the requirements listed in MIL-PRF-31032. The determination involves verification of the implemented quality management (QM) plan by a TRB, validation of the effectiveness of the implementation of the QM plan, and demonstrated the capability to produce printed board technology [1].

7. Qualified Manufacturers Listing (QML): Is a detailed listing of the capabilities by material type that the manufacturer demonstrated during qualification testing. Customers using this list can quickly determine which manufacturers are capable of meeting their needs.

8. Specification Sheets: Military specifications that supplement the base document and contain detailed performance requirements for specific printed board technologies. Technologies are classified as different constructions or types of printed boards, such as rigid, flex, multilayer, high-speed, etc. This allows special requirements for different technologies to be addressed as well as rapid implementation of new technologies. This specification also allows the development of custom technology requirements where there is no supporting specification sheet, so customers can push the envelope for new technological advances without waiting for specification changes [1].

9. Quality Management Plan (QMP): The manufacturer must maintain a documented and disciplined quality system that emphasizes process controls, defect prevention, and continuous improvement. Manufacturers are encouraged to develop a QM plan based on their existing process flow and quality system.

10. Self-validation: Self-validation is the manufacturer's means of determining compliance to MIL-PRF-31032 and the QM program. It is an internal audit to MIL-PRF-31032. Self-validation results must be reported to the TRB.

An effective self-validation program reporting to the TRB shows that the manufacturer has taken responsibility for its own system and is key to the reduction in qualifying activity oversight. This must be completed before the qualification activity will begin the certification process. Any discrepancies found must be corrected and implemented before certification can begin.

Certification Process Terms

1. Technical Review Board (TRB): The manufacturer forms a team of in-house experts to make decisions regarding printed board acceptability and certification. This time-saving step reduces costs and lead times by reducing the DoD approval process.

2. Process Flow: A process flow is a sequential list of all processes required to build printed boards from the time the order is taken to the time the product is shipped. This may include flow charts, production travelers, or any other means of documenting the flow. The flow must include any possible processes to be used for QML product, including rework steps, key process monitors, and contract services. Procedure numbers, process procedures, or other references for each step must be identified.

GLOSSARY

- Add-on Qualification: The use of the qualification process to add new technologies to the QML.
- Capability Verification Inspection (CVI): Scheduled periodic (every two years) process of demonstrating the ability to adhere to current qualified technologies.
- Certification: Qualification of products and audited quality systems in the specification.
- Certified Process Flow: The process sequence used for qualified products.
- DLA: The Defense Logistics Agency in Columbus, Ohio, USA.
- Gage Code: A unique number assigned by SAM for the site; must be marked on all MIL-PRF-31032 products.
- Initial Qualification Test Plan: A qualification test plan lists the capabilities that the manufacturer desires to qualify and describes how the manufacturer will demonstrate its ability to meet MIL-PRF-31032.
- Letter of Certification: Sent by the DLA at the conclusion of the validation process.
- Lot Conformance Inspection: In-process inspection of MIL-PRF-31032 products.
- Pre-Validation: The manufacturer notifies the qualifying activity (DLA) of its intent to pursue qualification to MIL-PRF-31032.
- P/PRF: Performance specification.
- Percent Defective Allowance (PDA): Maximum lot percent defective, which will permit the lot to be accepted after final inspection, test, or microsection .
- Periodic Conformance Inspection (PCI): Monthly or annual testing required by technology to sustain qualification status.
- Process Flow: A sequential list/diagram of all processes required to build printed boards from the time the order is taken to the time the product is shipped.
- QM Program: The TRB's administrative procedures for the interpretation of MIL-PRF-31032.
- QM/QMP: Quality management plan (documentation that defines MIL-PRF-31032 systems for the facility).
- QML Listing: QML managed by the DLA.
- QML Certification Brand: "QML" or "Q" is marked on qualified PWBs.
- Qualification Activity: Communicating with the DLA.
- Qualification Testing: Qualification inspection/testing (rework simulation, thermal shock, etc.) must be performed at a lab acceptable to DSCC.
- Revalidation Survey: Periodic audit by DSCC to validate the implementation of the QM program and records.
- Specification Sheets: Military specifications that supplement the base document and contain detailed performance requirements for specific printed board technologies.
- Self-validation: Internal audit to MIL-PRF-31032 done in-house to determine compliance to the QM plan.
- Technical Review Board (TRB): The team that manages all the requirements and maintains certification and keeps the qualifying activity informed on the status of the QMP.
- Test Optimization: Any test or verification may be reduced or eliminated by the TRB.
- Validation: DSCC audit to determine if the QM program is being implemented in accordance with the QM plan.

3. Initial Qualification Test Plan: Qualification test plans must be created when performing an initial qualification. A qualification test plan itself lists the capabilities that the manufacturer desires to qualify and describes how the manufacturer will demonstrate its ability to meet the MIL-PRF-31032 specification sheet requirements for said capabilities.

Ongoing Certification Terms

1. Capability Verification Inspection (CVI): Can be used to validate that the qualified materials and processes continue to conform to the originally qualified capabilities. In this regard, CVI serves as a tool for monitoring the quality and reliability of the manufacturer's technology, capabilities, materials, and processes. The inspection and associated report is performed every two years. The CVI data must reflect the capabilities of the manufacturer as related to the QML listing for each material type.

2. Add-on Qualifications: Allow a manufacturer to more easily expand its QML listing than performing initial qualifications. Add-on qualifications are governed by the QM plan, and the manufacturer's qualification test procedure and are controlled by the TRB. One benefit of add-on qualification is that it allows the TRB to use current jobs to extend their QML listing. It also allows jobs outside the manufacturer's current capability listing to be accepted. To do this, though, careful attention must be paid to the add-on qualification test plan when it is generated. Also, open communication with the customer should be practiced. In the case that the boards do not meet testing requirements, the shipped product cannot be considered to be MIL-PRF-31032 certified.

3. PDA: Percent Defective Allowed (PDA): Is most useful when the nature of the commodity makes it unreasonable to perform destructive testing on finished products, such as printed boards. The idea behind PDA is that if a significant portion of a lot is defective or if a significant portion of representative test vehicles has defects, there is a chance that the remaining lot may have defects that the inspections did not uncover. PDA is a flag for the TRB to look twice at suspect lots before shipment.

4. Periodic Conformance Inspection (PCI): Formerly called Group B, PCI is performed periodically at defined intervals (usually monthly) rather than on each lot. The tests and frequencies listed in the associated specifications are guidelines to a PCI assessment program.

5. Lot Conformance Inspection (LCI): Formerly called Group A and in-process inspection, LCI is the method of verifying and recording printed board compliance on a lot-by-lot basis.

Your foreign-language class for MIL-PRF-31032 is over, so your communication with the DLA can now start without any awkward moments. In my next column, I will summarize step two: pre-certification requirements. **PCB007**

Reference

1. Defense Logistics Agency (DLA), "Certification and Qualification Information for Manufacturers: MIL-PRF-31032," Revision G, August 2013.

Mike Hill is president of MIL-Q-Consulting LLC. He has been in the PWB fabrication industry for over 40 years. During that time, he participated in specification writing for both IPC and the military. Past employers include ViaSystems, Colonial Circuits, and DDi. To read past columns or contact Hill, click here or email Milqconsulting@outlook.com.

Electronics Industry News and Market Highlights

IDTechEx: The Future of Wearables Is Medical (Part 1) ▶

Wearable technology and medical devices have many prominent synergies. Electronic devices on the body are perfectly placed for sensors to collect data relevant to healthcare, with communication options allowing them to be the gateway to digital health data.

Wrist-worn Wearables Maintain a Strong Growth Trajectory in 2Q19 ▶

Shipments of wrist-worn wearables—including smartwatches, basic watches, and wrist bands—reached 34.2 million units, up 28.8% year over year during the second quarter of 2019, according to the International Data Corporation (IDC) Worldwide Quarterly Wearable Device Tracker.

Shipments of Bluetooth Powered IoT Devices to Exceed Smartphones by 2024 ▶

According to ABI Research, by 2024, IoT end markets will represent 31% of total Bluetooth and 27% of Wi-Fi device shipments, up from 13% and 10%, respectively, in 2018.

TrendForce Releases Latest Analysis of the SSD Market ▶

TrendForce data shows that NAND Flash suppliers accounted for about 35% of the total SSD shipments to the channel market in 2018.

Spending on AI Systems in the Middle East & Africa to Top $374M in 2020 ▶

Spending on AI systems in the Middle East and Africa is expected to maintain its strong growth trajectory as businesses continue to invest in projects that utilize the capabilities of AI software and platforms.

Radar Market Size Worth $38B by 2025 ▶

The global radar market size is expected to be valued at $38.01 billion by 2025, registering a CAGR of 3.8% over the forecast period, according to Grand View Research Inc.

Canadian Spending on Customer Experience Technologies to Surpass $18 Billion by 2022 ▶

Canadian spending on customer experience (CX) technologies will total $15.8 billion in 2019, an increase of 5.1% over 2018, according to the Worldwide Semiannual Customer Experience Spending Guide from International Data Corporation (IDC).

Global Biometrics Technologies Market Report 2019-2027 ▶

The global biometrics technologies market accounted for $14.6 billion in 2018 and is expected to grow at a CAGR of 16.2% over the forecast period 2019–2027, to account for $55.42 billion by 2027.

Worldwide Memory Semiconductor Market to Reach $52.057 Billion by 2024 ▶

The Global Memory Semiconductor Market for Consumer Electronics Industry is projected to grow at a CAGR of 0.63% to reach $52.057 billion by 2024, from $49.999 billion in 2019.

Top 10 Trends in Information and Communication Technology Industry for 2020 ▶

TrendForce provides predictions of the information and communication technology industry for 2020, focusing on 10 key themes.


Leverage technology for competitive advantage:
it's what leaders do.
CapStone™ is the culmination of ESI's decades of laser-material interaction expertise and technology leadership in flex PCB processing.
Double your throughput and dramatically decrease your per-panel processing costs.
Adopt CapStone's new technology early...
and stay ahead.
CapStone™

Additive Electronics Momentum

Flex Talk

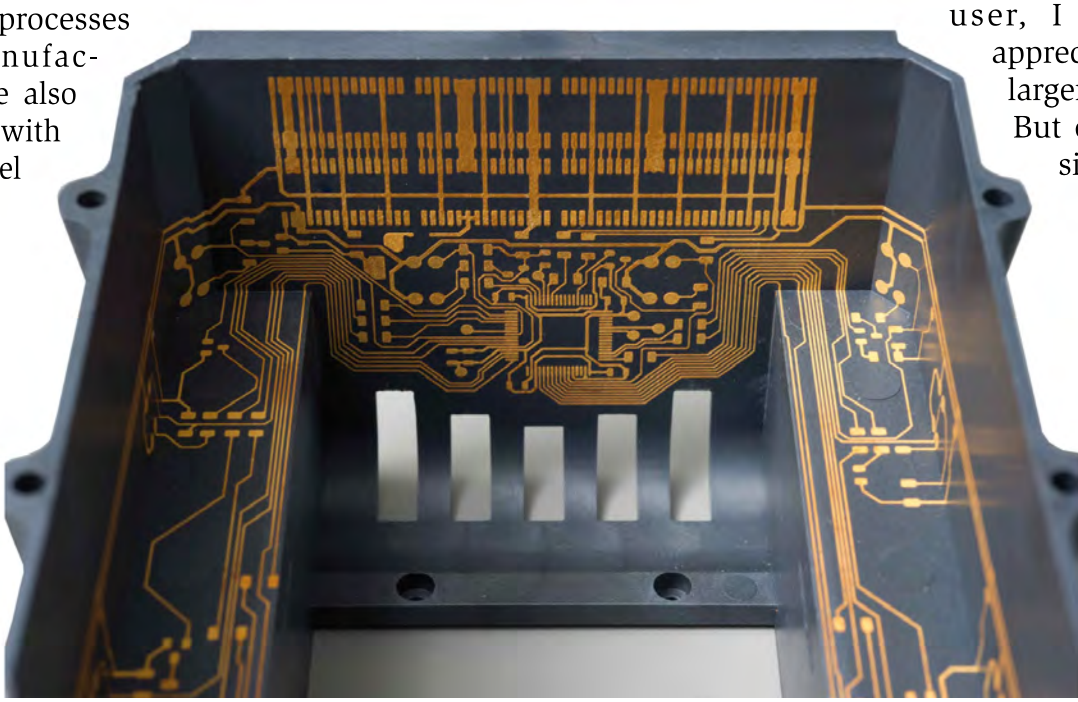
by Tara Dunn, OMNI PCB

There is no denying that additive manufacturing processes have been gaining popularity over the past several years. We have additive manufacturing conferences, journals, and no end to continued research and development. Additive electronics, a segment of this technology, has also been growing in popularity, and there are several alternatives on the market that cater to quick-turn, prototype, and PCB options.

Digging down into the additive electronics market, there is another important segment that is starting to build momentum: the fabrication of feature sizes bridging subtractive-etch processing capabilities and IC-scale technology. Fully additive, semi-additive (SAP), or modified semi-additive processes (mSAP) enable fabricators to create PCBs, on flex or rigid materials, with much finer feature sizes than the traditional subtractive etch processes can manufacture while also working with larger panel sizes than typical IC fabrication. While the concept of additive electronics is not new, it is new technology in terms of PCB manufacturing.

I have been involved with additive electronics for the past several years, and I have seen the discussion of and demand for sub-75-micron feature sizes slowly grow. Conversations, questions, and research about SAP and mSAP increased significantly when it was announced that the mSAP process was used to create the circuitry in the more recent versions of our smartphones. While this process is available in very high volume in some areas of the world, it is still in the early stages of development in other areas.

In smartphone applications, it is easy to see the benefit of 35-micron line and space and the ability to shrink the circuit size to allow space for a larger battery and more sophisticated electronics. As a smartphone user, I know I appreciate that larger battery! But even outside of the



high-volume smartphone market, there is an increased interest in the ability to design with feature sizes below 75 microns and, along with that, there is an increased interest in what options are available in low-volume and development quantities to achieve this. While working with the SMTA to help develop a new conference launching this fall, "Additive Electronics: PCB Scale to IC Scale," it has become clear that the industry is at an inflection point. Increasingly sophisticated electronics in smaller and lighter-weight packaging will continue to drive the need to deliver finer feature sizes, leaving designers with the need to identify potential solutions and provide fabricators with the opportunity to bring in new capabilities to meet this need.

Increasingly sophisticated electronics in smaller and lighter-weight packaging will continue to drive the need to deliver finer feature sizes...

One of the forerunners in this technology advancement is Averatek's A-SAP process. Averatek has created a liquid metal ink (LMI) and semi-additive manufacturing processes that are capable of achieving feature sizes from 75 down to 5 microns. This technology is available for license to PCB facilities and fits well with traditional PCB manufacturing equipment. Currently, the A-SAP process is being piloted in two PCB fabrication facilities and will be commercially available over the next few months.

There are several benefits to the semi-additive process; the most obvious is line width and space. In traditional subtractive etch processing, when feature sizes push below 75 microns, there are significantly decreased yields along with significantly increased cost per part. With additive processing, not only is a 50-micron line width and space now achievable, but feature sizes can also design below 25 microns before pushing fabrication capabilities. These additive technologies open design possibilities in several ways, such as the size. The overall form factor can be shrunk considerably, and this technology is finding use in things like hearing-aid applications, neural probes and catheters.

Mastering complex pinouts is another significant benefit. Routing 0.3-mm-pitch and 0.2-mm-pitch devices can be accomplished without the need for costly multiple lamination cycles and with a reduced layer count when compared to traditional subtractive etch processing. Imagine moving from a 12-layer, 3-lamination stackup to an 8-layer design requiring only one lamination cycle by swapping out four traditional layers for ultra-high density layers and incorporating those with traditional subtractive etched layers.

Not all applications require smaller, lighter-weight packages. This additive technology also lends itself to the ability to add additional electronics into a fixed space. Imagine adding additional electronics and functionality in an automobile.

As this technology develops with our low-volume manufacturers, there will be many more options available for designers challenged with space and weight constraints. At this point, there is a lot of brainstorming and creative thinking being done as to how to best take advantage of these added capabilities. It is clear that over the next several years, this technology will find space alongside both subtractive etch fabrication methods and IC fabrication techniques. **PCB007**

CLICK HERE to attend the Additive Conference on October 24, 2019.

Tara Dunn is the president of Omni PCB, a manufacturer's rep firm specializing in the PCB industry. To read past columns or contact Dunn, click here.

Via Hole Filling and Plugging, Part 2

Trouble in Your Tank

by Michael Carano, RBP CHEMICAL TECHNOLOGY

In my previous column, I presented several options with which to accomplish blind and through-hole via filling. In this edition of "Trouble in Your Tank," I will discuss filling blind vias and through-holes with polymeric pastes.

Via Fill Paste

Often, the phrase "plugging paste" is used to describe the method and material of completely filling blind vias and through-holes. In general, paste filling material selection is at the request of the end user and indicated for a number of reasons. It has been my experience that major OEMs are driving the industry to migrate to the high-Tg/low-CTE plugging paste formulations for high-density applications. In addition, these formulations are of a non-conductive nature that provides a high-quality plugged via that is cost-effective (Figure 1). Limitations abound depending on PCB thickness, via diameter, and paste properties.

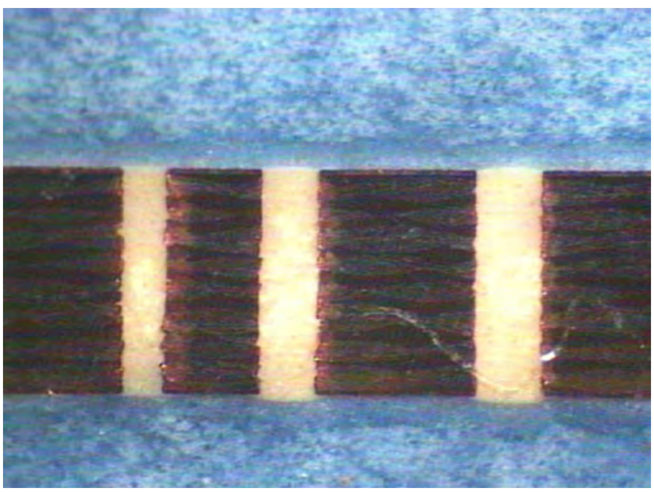


Figure 1: Example of a paste-filled through-hole.

Properties of Via Fill Materials

What attributes are needed for a high performance via fill material? There are specific requirements for the plugging paste material, such as:

- Good adhesion between copper and paste even under temperature influences
- Good adhesion of copper, dielectrics, or photoresist
- Solvent-free, one-pack system
- No air inclusions in the paste
- Tg > 140°C
- CTE < 40 ppm (below Tg)
- No shrinkage during curing
- Easily planarized

Additionally, the plugging paste material must maintain a reasonable shelf life at room temperatures. Keep in mind that these materials are thermally reactive. It is highly recommended that the fabricator use a 100% solids content of the paste material with the thermally cross-linkable epoxy resin and specially designed ceramic fillers. The ceramic filling material restricts Z-axis expansion when the filled vias are subjected to a thermal load. Interestingly, the coefficient of thermal expansion must remain in the 40–60 ppm range to ensure that via cracks do not occur in the filled via. In addition, it is critical that Z-axis

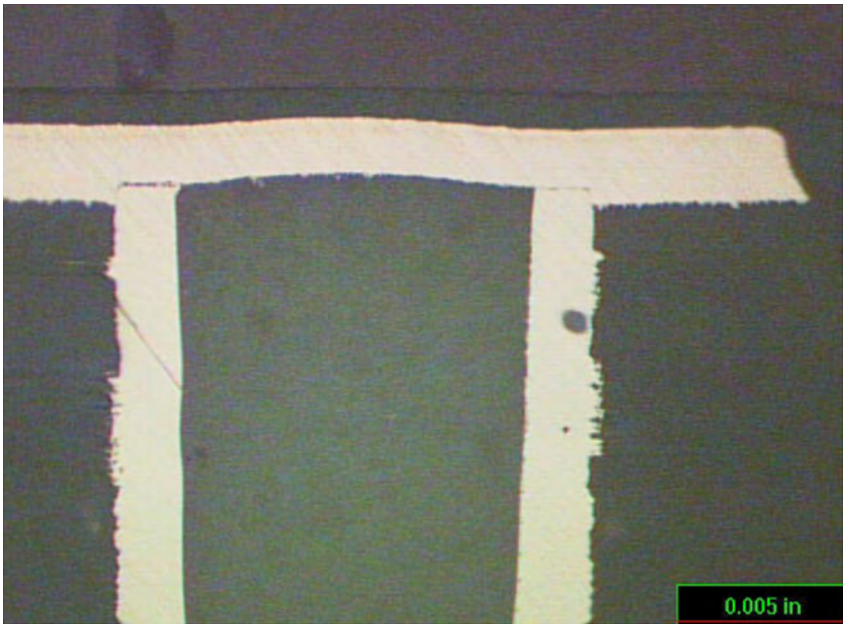


Figure 2: Plated copper separating from filled via due to excessive Z-axis expansion.

throughout various thermal excursions. Z-axis expansion notwithstanding, the second critical thermal characteristic is the glass transition temperature of the cured paste material. Typically, a Tg of 140°C is ideal. However, the Tg can be increased by prolonging the final curing time and increasing curing temperature from 140°C to approximately 175–180°C. It is desired to have the highest possible Tg without impacting the flow and metallization properties [1].

With increased densification leading to higher I/Os, smaller components, higher assembly temperatures and smaller vias, the CTE gains increased importance. Thus, the CTE values of the paste must be minimized to relieve stress that will cause the plug to over-expand, allowing the over-metalized copper deposit to lift [2]. It is critical that to attain long-term stability within the filled via under load conditions, load amplitudes must be minimized as much as possible. This means that the CTE must be as low as possible over the temperature ranges [2].

expansion is minimized to prevent the plated cap from lifting (Figure 2).

As noted previously, a properly formulated plugging paste for via fill must maintain a low CTE at and above 140°C. The ceramic particles that are formulated in the resin system function to restrain Z-axis expansion under thermal loading. The ceramic fillers can be seen in Figure 3 under high magnification of the fully cured polymeric paste.

There is no disputing the fact that the vias must be filled void-free and maintain integrity

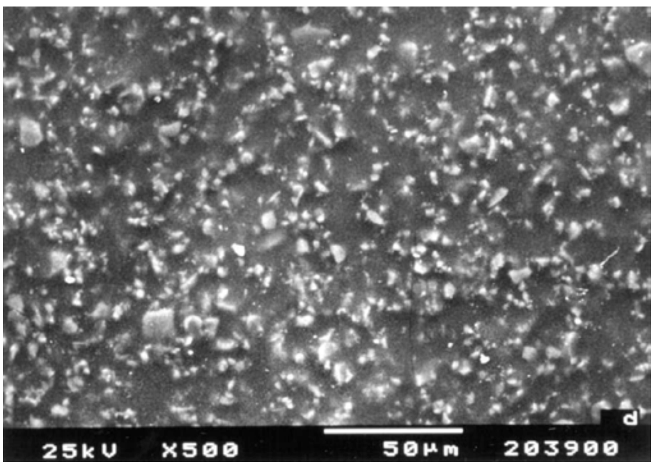


Figure 3: Top view of filled via with plugging paste. Ceramic particles embedded in the matrix are clearly visible.

Conclusion

Regardless of the method of via filling chosen, this is a process that is here to stay. Via-filing technology is a critical aspect of HDI PCB fabrication and the never-ending quest for miniaturization. **PCB007**

References

1. Karsten Andra, "Hole Plugging Technology for Multilayers and HDI Packages," EPC PCB Convention, 1999.
2. Internal communication with Lackwerke Peters.

Michael Carano is VP of technology and business development for RBP Chemical Technology. To read past columns or contact Carano, click here.

Supplier Highlights

DuPont Celebrates Groundbreaking for Circleville Expansion ▸

DuPont Electronics & Imaging recently hosted a groundbreaking ceremony to celebrate progress on the planned $220 million expansion at its Circleville, Ohio, plant.

The Use of Insoluble Anodes in Acid Copper Plating ▸

Soluble anodes have been the staple of the industry for decades, but they require extensive maintenance and generate waste in both copper metal and electrolyte. As plated copper thickness uniformity requirements become more stringent, more and more time will be needed for anode maintenance to ensure the uniformity of the anodic setup.

Brigitflex Invests in New Excellon Drilling/Routing System ▸

Excellon has installed a model 154L vision drill/routing system at Brigitflex Inc. of Elgin, Illinois.

Mitsubishi Electric Develops Metal Corrosion Sensor Designed for Mounting on PCBs ▸

Mitsubishi Electric Corp. has developed what is believed to be the world's first compact metal corrosion sensor small enough to be mounted on printed circuit boards.

iNEMI Publishes Organic PCB and Power Conversion Electronics Chapters From the 2019 Roadmap ▸

The International Electronics Manufacturing Initiative (iNEMI) has announced the publication of the Organic PCB and Power Conversion Electronics chapters of the 2019 roadmap.

Chemcut Representative TriChem Technologies Expands to Illinois ▸

TriChem Technologies will now be representing pre- and post-sales support of Chemcut wet processing equipment in Illinois. TriChem has been representing Chemcut equipment in Ohio, Michigan, Indiana, and Kentucky since 2004.

Nano Dimension Sells DragonFly LDM Additive Manufacturing System to CityU ▸

Nano Dimension Ltd has sold a DragonFly Lights-Out Digital Manufacturing (LDM) system to City University of Hong Kong (CityU), a globally ranked public research university in Kowloon, Hong Kong.

Amphenol Printed Circuits Recognizes Panasonic as a Qualified Vendor for Flexible Laminates ▸

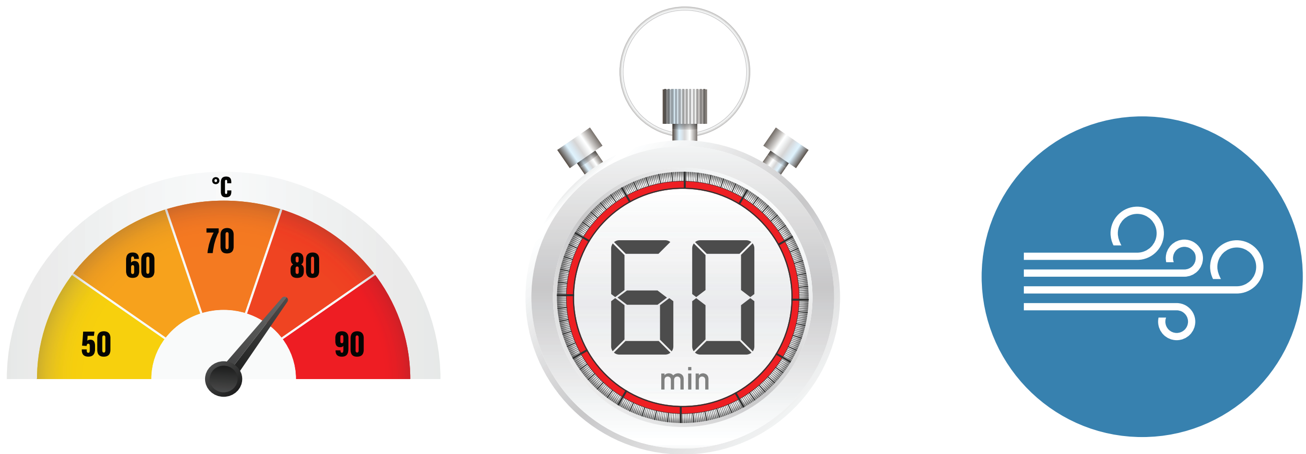
Amphenol Printed Circuits has qualified Panasonic's Felios Flexible Laminates and will now increase usage on new part numbers.

MacDermid Alpha Releases New Acid Copper Metallization Solution ▸

MacDermid Alpha Electronics Solutions has released MacuSpec HT 300, a high throw DC acid copper metallization able to metallize through holes of 15:1 aspect ratio boards at production volumes.

Aismalibar to Speak at Cooling Days Conference ▸

Aismalibar will speak and exhibit at this year's "Cooling Days" Conference in Würzburg, Germany, on October 22 and 23. The conference will focus on how to cool electronics using the latest generation PCBs.



Solder Mask Tack Dry

Article by Nikolaus Schubkegel

Tack drying is a key process step, as it makes the solder mask surface suitable for exposure. Tack drying happens after coating the board surface with a solder mask, but before exposure. If the tack drying process is done correctly, the solder mask surface will not adhere to the base plate or the artwork.

The tack drying process is controlled through three parameters: temperature, dwell time, and airflow.

Practically speaking, in a factory environment, two of the three parameters—temperature and dwell time—are easily controlled. Generally, there is little opportunity to change the airflow. Let's look at temperature and dwell time more in detail.

Temperature

As a general rule, the tack-dry temperature should be as low as possible; in other words, it should only be as high as necessary. If the temperature is too low, the evaporation rate for the solvent will be to slow, and the solder mask will not dry in a reasonable amount of time. If the temperature is too high, however, the dry time certainly will be excellent, but it could create a solder mask lock-in with repercussions by the developing time.

These repercussions manifest themselves in the necessity of longer dwell times in the developer and potential solder mask residues on pads and in holes.

For example, the LPI solder mask for rigid products from Taiyo recommends temperature range for tack dry between 65–80°C (150–176°F) at the surface of the PCB. The elevated temperature leads to a higher vapor pressure of the solvent in the diffusion interface on the immediate solder mask surface. Air circulation then takes the solvent vapor away. Also, as a general rule, increasing the temperature by 20°C will double the evaporating speed of the solvent (valid in the range of 20–100°C).

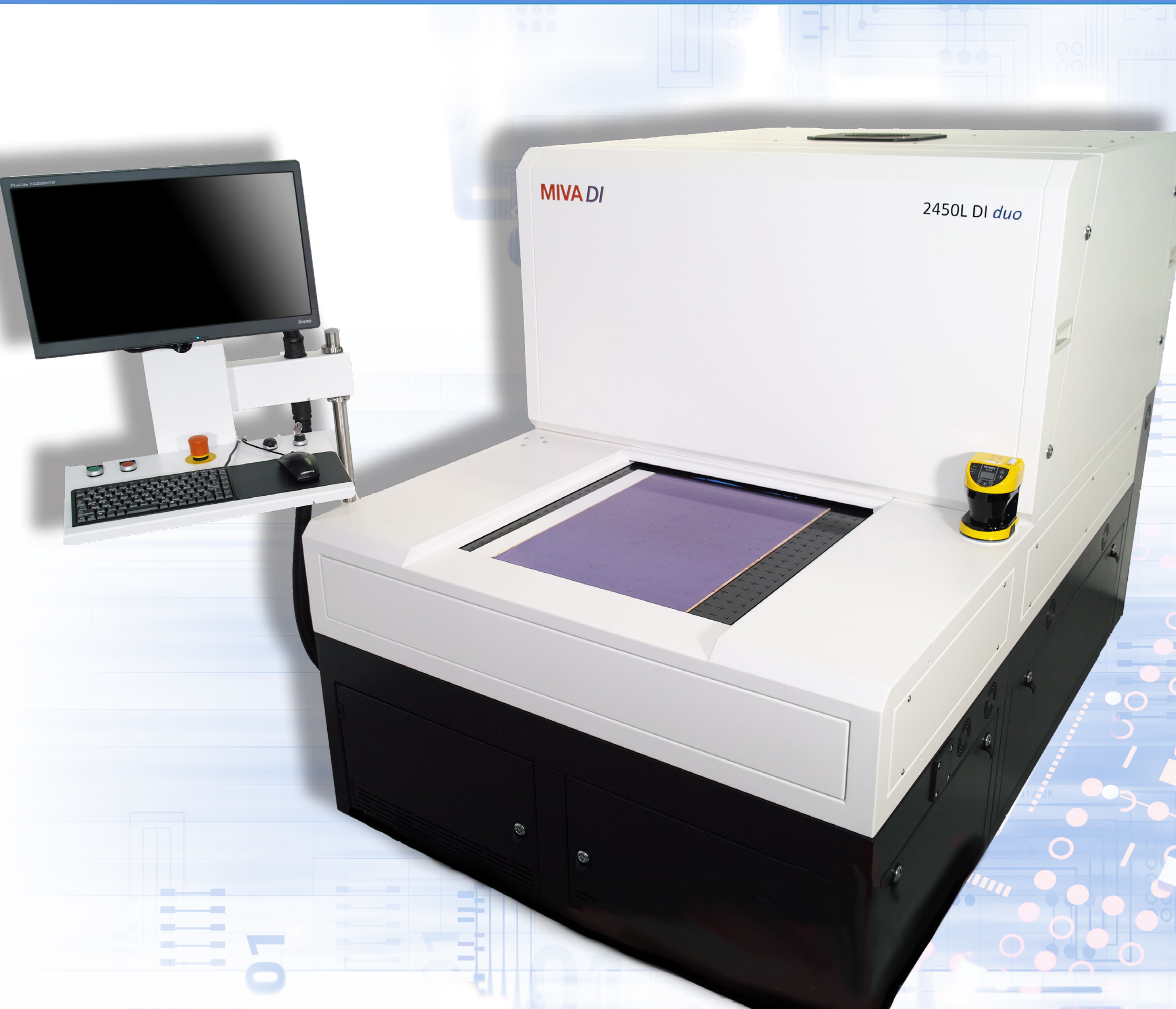
Dwell Time

At any given temperature, the hold time in the tack-dry oven should be as short as possible. In a well-ventilated convection dryer with an air temperature of 80°C, the dwell time should not exceed the recommended hold time

MivaTek Global

2400DT - Dual Tray DI

High Throughput • Automation Ready



Globally Marketed and Serviced

MIVA: LED DIRECT IMAGING

Features

- **LOAD WHILE IMAGING!**
- Quad-wave LED DLP: 360, 375, 390, and 405nm
- Worldwide LED DLP Patent Holder
- Compact Footprint, Low Cost of Operation
- Low Maintenance Costs, No Contract Required
- Panel Sizes: 24" x 30" (up to 30"x60")
- Up to 6 Light Engines, High Throughput
- Automation compatible with third party sources
- Applications: PCB, Glass Emulsion, Chemical Milling, Electroforming, Semi-Conductors
- Production Proven: 44 Machines in North America

MivaTek Global, LLC, 6 Dickinson Drive, Suite 215, Chadds Ford, PA 19317 • Website: www.MivaTek.Global
Contact: Chris Hrusovsky • M (612) 205-6113 • Chris@MivaTek.Global

from the technical datasheet. Typical dwell times are in the range 40–70 minutes, depending on the type of solder mask.

If the air temperature is higher, however, things are completely different. Sometimes, for example, the air temperature can be much higher than the recommended temperature of 80°C. This can happen in tunnel tack dry ovens of a certain design. In such cases, the ideal belt speed (dwell time) must be determined experimentally. The optimal transportation speed (conveyor speed) for a good drying depends on the board size, amount of copper, and panel thickness. In practice, the conveyor speed is set depending on panel thickness. The panel thickness can be measured very easily; it is often stated in the accompanying papers.

Looking at some real-world examples, let's start with these assumptions:

- The PCB is built with FR-4 and copper
  - Copper density: 8,900 kg/m³
  - FR-4 density: 1,850 kg/m³
  - Average density of a PCB: 3,965 kg/m³ (assumption: 30% copper)
- The heat capacities are as follows
  - Copper heat capacity: 385 J/kgK
  - FR-4 heat capacity: 600 J/kgK
  - Average heat capacity of a PCB: 536 J/kgK (assumption: 30% copper)
- We will consider three boards
  - 0.05 mm, 0.5 mm, and 5 mm
  - The size is 508 x 609 mm (approximately 20" x 24")

We will further assume that the airflow is laminar over the PCB surface and flows over both sides: on the top and bottom of the PCB. The heat transfer coefficient depends strongly on the airflow over the PCB surface. Of course, if the airflow is turbulent, the heat transfer coefficient is higher, and drying is better, but that turbulent airflow could cause surface defects to appear on the wet solder mask surface.

At laminar airflow $Re < 10^5$, the calculated heat transfer coefficient is 3 W/m²K at an airspeed of 0.3 m/s and 9 W/m²K at an airspeed of 3m/s. Later, I'll show you how to calculate the heat transfer coefficient. With the given data, it is possible to calculate the time until the board reaches 80°C, for example, if the airflow temperature is 120°C and the temperature of the board at the beginning is 25°C:

$$CV\rho d\theta = \alpha A(\theta_L - \theta)dt$$

Where:

C = specific heat capacity of the PCB, $Jkg^{-1}K^{-1}$

V = volume, A · d, m³

ρ = density, kgm^{-3}

α = heat transfer coefficient, $Wm^{-2}K^{-1}$

θ = temperature of the PCB after time t, K

θ_L = air temperature (i.e., 120°C, 393 K)

θ_0 = temperature of PCB at t = 0, 25°C, 298 K

A = surface, m²

d = board thickness, 0.5 mm/5 mm (0.5; 5 · 10^{-3} m)

t = time, seconds

And:

$$\int_{\theta L}^{\theta} \frac{d\theta}{\theta L - \theta} = \frac{\propto 2A}{CV\rho} \int_{0}^{t} dt$$

Where:

$$\theta = \theta_L - (\theta - \theta_0) e^{-\frac{2\alpha}{C\rho d}t}$$

With the known values, the heating time can be calculated:

α = 3 or 9 / θ = 80°C after time t / θ_L = 120°C / θ_0 = 25°C / C = 328 /

ρ = 3,965 / d = 0.05 · 10^{-3} or 0.5 · 10^{-3} or 5 · 10^{-3} m

Table 1 shows the results of these calculations.

Board Thickness	Airflow	Heat Transfer Coefficient α	Time for PCB to Reach 80°C if Air Temperature is 120°C (in Seconds)
0.05 mm	0.3 m/s	3	3.4 s
0.5 mm	0.3 m/s	3	34 s
5 mm	0.3 m/s	3	347 s
0.05 mm	3 m/s	9	1.2 s
0.5 mm	3 m/s	9	12 s
5 mm	3 m/s	9	114 s

Table 1: The results of calculations to determine the time to heat a room-temperature PCB to 80°C.

The conclusion is that a thin board heats up very quickly while a thick board heats up slowly. Consequently, in a tunnel tack dry, the belt speed needs to be set high for thin boards and low for thick boards. While this is relevant for the processing of boards with different thicknesses, it is also very important for the construction of tunnel ovens.

Airflow

The most important criterion for tack dry is the airflow on the solder mask surface. For a given piece of equipment, there is no possibility to increase the airspeed on the surface of the PCB. It is possible to open or to close ventilation flaps on the machinery, and this will affect the airflow and airspeed on the surface of the PCB. Mainly, adjusting the flaps will simply affect the temperature distribution on the PCB surface, and it is not advisable to change the flap settings.

Even at a long dwell time at an elevated temperature, the solder mask will not dry properly if there is insufficient airflow. Higher airflow results in a higher airspeed on the PCB surface, has a strong influence on the heat transfer coefficient (i.e., the heating-up speed of PCB), and has a strong influence on the evaporating speed of solvent due to a higher difference of partial pressure of solvent in the air and in the diffusion interface on the solder mask surface.

The heat transfer coefficient can be calculated from the Nusselt criteria:

$$N_u = \frac{\alpha l}{\lambda}$$

Which is valid if:

$$R_e < 10^5$$

And:

$$N_u = 0{,}66\ R_e^{0{,}5}$$

Nusselt and Reynolds are dimensionless numbers. Thus, the Reynolds criterion is as follows:

$$R_e = \frac{w l \rho}{\eta}$$

Where:

w = airspeed 0.3 m/s

l = length of the PCB (0.6m)

ρ = air density at 120°C, 1-bar pressure 0.8878 $kg \cdot m^{-3}$

η = dynamic viscosity of air at 120°C, $22{,}792 \cdot 10^{-6}$ Pa.s

λ = heat conductivity of air $32{,}73 \cdot 10^{-3}$ W/m·K

$$R_e = 7{,}12 \cdot 10^3;\ N_u = 0{,}66\ R_e^{0{,}5}$$

$$N_u = 0{,}66 \cdot 84{,}4 = 55{,}6$$

$$\alpha = \frac{Nu\,.\lambda}{l} = 3$$

α = heat transfer coefficient, $Wm^{-2}K^{-1}$

If the airspeed is 3 m/s, then a similar calculation gives an increased heat transfer coefficient of 9.5 Wm-2K-1. The position of the airflow relative to the PCB itself can be important. Figure 1 illustrates the differences in evaporation efficiency. In the first example, boards are parallel to the airflow resulting in both sides of the board receiving nearly equal airflow. In the second case, only one side of one board is surrounded by good airflow, which means that evaporation will be at different rates.

The evaporation speed is different for pure solvents compared to solvent blends and completely different if evaporating a complex solvent resin mixture (as in solder mask). The

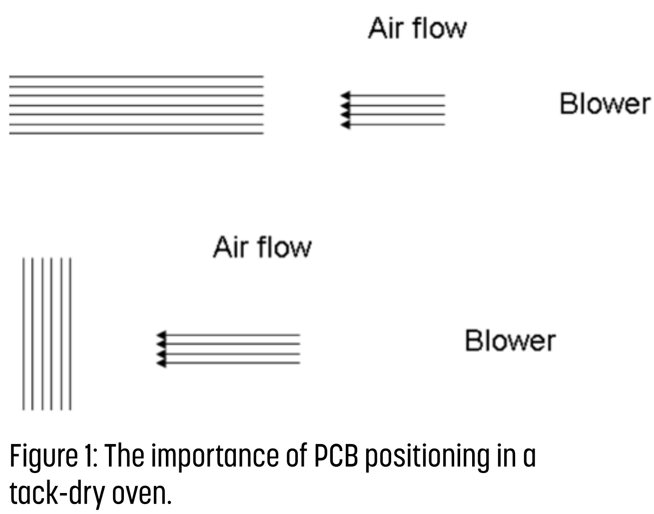


Figure 1: The importance of PCB positioning in a tack-dry oven.


Are you getting to the bottom of your via?
Prevent connection failures in thick PCB substrates with plasma treatment
New VIA™ Series 2.5 cleans deep, narrow vias where other processes can't go
Contact us now to learn more
+1-925-827-1240 / info@nordsonmarch.com
nordsonmarch.com
VIA™ Series 2.5
Plasma System
Nordson
MARCH

composition itself might have an influence on evaporating speed. The solvent concentration itself has an influence on drying speed and evaporating speed of solvents. At the beginning of the evaporation process, evaporation speed is high, then falls towards the end of the drying process. In addition, air humidity has a pronounced influence on the evaporation of esters, ethers, and alcohols. The evaporation of ether, esters and alcohols increases with the square of air humidity. Air humidity has almost no influence on the evaporation of hydrocarbons. As can be seen, the matter is very complex.

The mathematical relationship between the drying rate and airflow can be determined empirically. The drying rate depends on airspeed on the surface of the ink, vapor pressure of solvent, partial solvent pressure in the air, and is given by the following relation:

$$G = Kw^m \Lambda p$$

Where:

K = constant, depending on the airspeed, 0.91 if $Re < 50$; 0,6 if $50 < Re > 104$

w = airspeed m/s

m = constant, 0.4–0.5, depending on the airspeed.; $m = 0{,}385$ if $Re < 50$; $m = 0{,}5$ if $50 < Re < 104$

$\Lambda p = P_v - P_{air}$

P_v = partial pressure of solvent in the diffusion interface at the solder mask surface, vapor pressure of solvent in mm mercury

P_{air} = partial pressure of solvent in the air flow in mm mercury; limited to 0.8 Vol% in air BGV D25, less than the lower explosion limit for safety reasons

G = evaporating speed g/cm2.h

The calculations are not rigorous or exact, but they do illustrate the importance of temperature, airspeed, and hold time to the correct tack dry of solder mask. **PCB007**

References

1. K.F. Pawlow, P.O. Romankow, & A.A. Noskow, *Beispiele und Übungsaufgaben zur chemischen Verfahrenstechnik*, VEB Deutscher Verlag für Grundstoffindustrie, Leipzig, 1972.
2. P. Mischke, *Filmbildung in modernen Lacksystemen*, Hannover Vincentz Network, 2007.
3. E.K. Jülke, "Über die Verdunstung von Lösungsmittel aus Lackschichten," TH Zürich, Promotionsarbeit, 1962.

Nikolaus Schubkegel retired in February 2019. For the past 12 years, Schubkegel worked at Umicore Galvanotechnik GmbH in Germany as a technical service engineer for Taiyo products. Before that, he worked as a process engineer in the solder mask department at the former IBM-PCB plant (later STP) in Albstadt, Germany. Schubkegel obtained an M.Sc. degree in chemical engineering from the Polytechnic Institute in Timisoara.

Vertical Conductive Structures, Part 4: Tuning Your Signal Performance

Article by Joan Tourné
NEXTGIN TECHNOLOGY BV

Editor's Note: Click to read Part 1, Part 2, and Part 3.

The objective of this article is to demonstrate the possibility of using a stitching element as an alternative to the point-to-point connections that are used with traditional via technology. Point-to-point connections have the best performance in terms of signal integrity (one via less in the connection that distorts the signal). A via is mainly a capacitive element that causes signal loss/dispersion.

This study focuses on developing a layer transition element that minimises the loss and dispersion. We will use VeCS-2 technology (blind/hybrid) construction (Figure 1).

The advantage of using a stitching via is that we can use "traditional" orthogonal routing, which is very efficient in signal layer utilization. However, in some instances, orthogonal or Manhattan routing could yield a longer trace length, which could lead to a higher loss due to the dielectric and conductive losses.

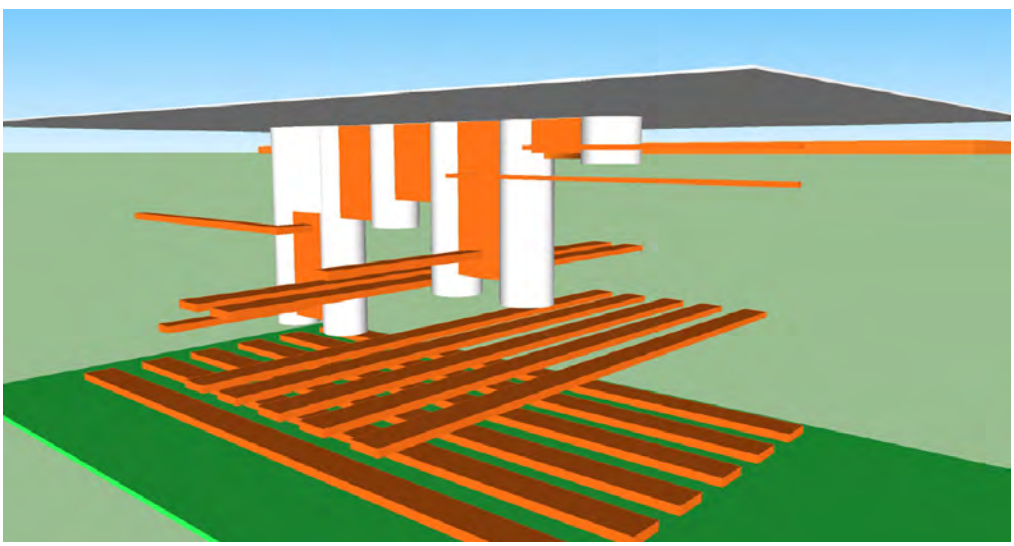


Figure 1: Multi-level VeCS-2 technology construction.

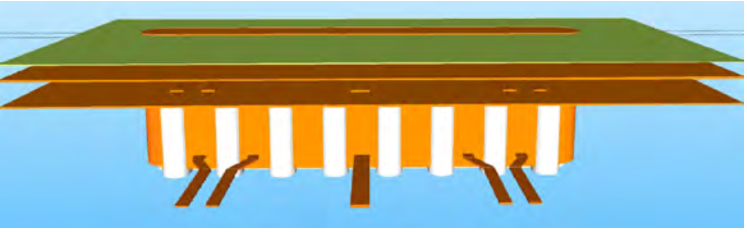


Figure 2: Simulation of the stitching element.

For the simulation of the stitching element, we have set up the element shown in Figure 2: a VeCS-2 (blind) element with eight connections—seven connections on one side and a GND reference on the opposite side of the slot, creating a reference to the seven connections. For visibility purposes, we have not drawn all of the reference layers.

The five connections consist of:

1. A differential pair
2. GND (purpose shielding)
3. A single-ended signal
4. GND (purpose shielding)
5. A differential pair

The GND reference creates controlled impedance for the differential pair. The topside of the differential pair VeCS section is back routed to minimise dispersion. In Figures 3 and 4, we also show the second route features. The dimension of the router bit used for the second route, top back route, and bottom route is 0.3 mm. All of this routing is done in the same process step.

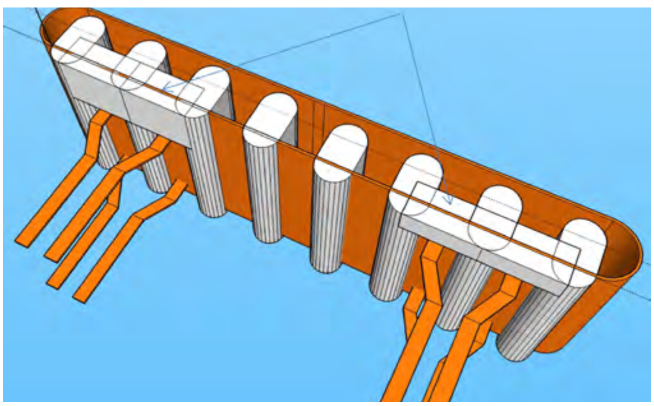


Figure 3: Top back rout.

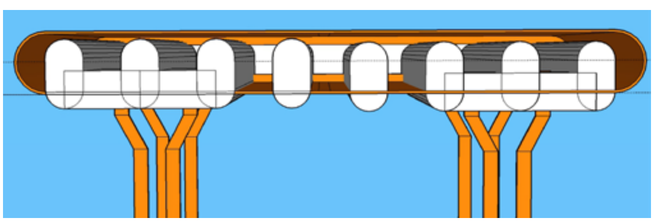


Figure 4: Top view showing the bottom rout (BR).

The element created sets up a buried microstrip for the differential pairs and the single-ended trace. The trace width used in this example is 0.12 mm; depending on the stackup, this can be tuned to the required differential impedance. The vertical trace width is 0.30 mm. This will yield a buried microstrip construction and an impedance of ± 110 ohms (ZOdd). We took into account an Er of 4.0 for the slot filling material.

The stub length is defined by the length below the trace and the part of the slot that is left over after removing the bottom of the slot (Figure 6). We have set a maximum stub length of 0.125 mm + 0.1 mm = 0.225 mm on the bottom side.

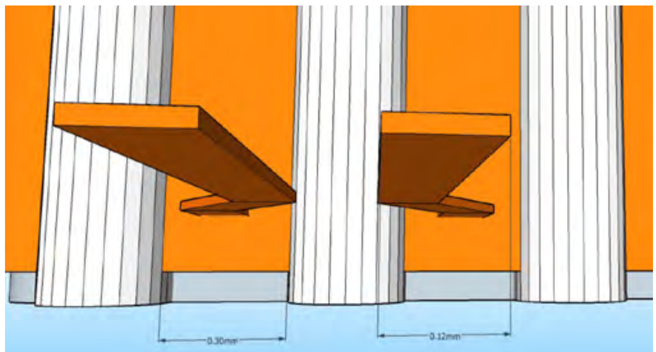


Figure 5: Vertical and horizontal trace width.

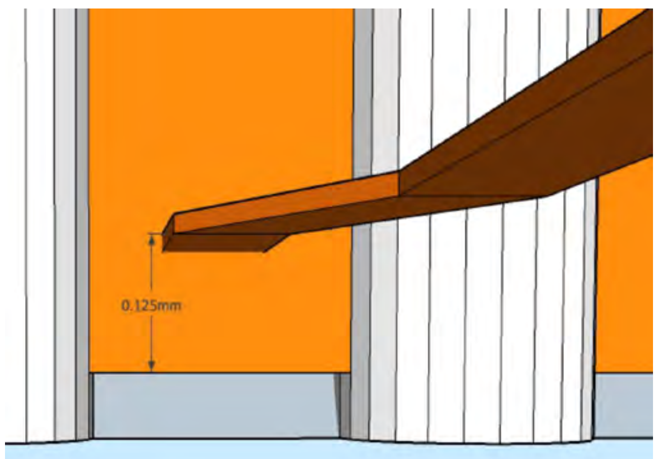


Figure 6: Stub length.

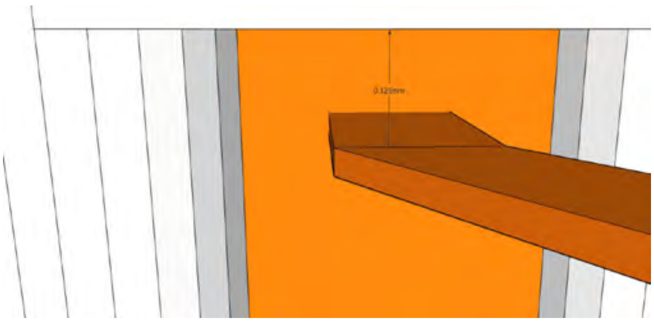


Figure 7: Stub length at the top.

Figure 8: The top side of the overplated slot.

On the topside, we have a maximum stub length of 0.125 mm. Here, we do not have to include the 0.1 mm bottom side of the slot.

The top side of the slot is overplated (after filling the trench with a dielectric), as shown in Figure 8.

The cap plating is optional (Figure 8) on the top of the back-routed VeCS element. If this is applied, then it could influence the impedance and need to be taken into the simulation as a design parameter (Table 1).

The dielectric in the centre of the board (thickness 1.0 mm) can be varied to adjust the vertical trace length to determine the influence on the signal performance. Now, it is used as a filler to save on processing unused inner layers. The filling material used for the slot is a via filling/plugging ink. The properties in the frequency dome are known and can be used in a field solver model to create an accurate model.

Signal Integrity Analysis of the VeCS Stitching Element

Escaping out of a connector or BGA is one complexity factor; there are also options to cre-

Layer	Type	Dielectric	Thickness [um]	Trace width [um]	Spacing [um]	Impedance Zodd [Ω]
1	Outer layer	Cu	50			
	Dielectric	Er 4.0	200			
2	GND	Cu	35			
	Dielectric	Er 4.0	200			
3	Signal	Cu	18	120	120	100
	Dielectric	Er 4.0	200			
4	GND	Cu	35			
	Dielectric	Er 4.0	1000			
5	GND	Cu	35			
	Dielectric	Er 4.0	200			
6	Signal	Cu	18	120	120	100
	Dielectric	Er 4.0	200			
7	GND	Cu	35			
	Dielectric	Er 4.0	200			
8	Outer layer	Cu	50			

Table 1: Stackup.

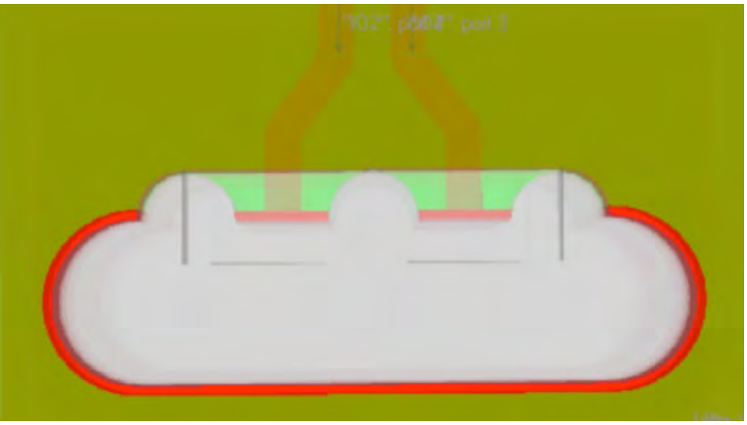


Figure 9: Top view of the VecS element.

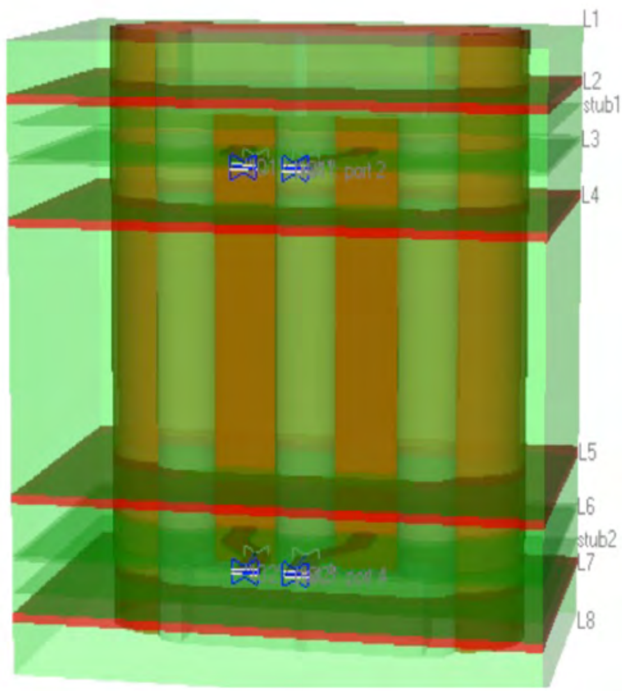


Figure 10: Eight-layer stackup.

ate layer transitions to avoid the point-to-point connections on the same layer. To minimize signal distortion/reflection, point-to-point connections are used in favour of via layer transitions. In this section, we analyse the SI performance of layer-to-layer transitions using VeCS-2 technology.

Enabling the use of layer transition with little reduction in the SI performance of the transmission line will simplify the routing of a lot of designs. This reduces the complexity of the design (e.g., layer reduction, eliminates sequential build-ups, etc.). For the SI performance analysis, we used "Simbeor THz 2017.01" with thanks to Yuriy Schlepnev for translating our VeCS design in the analysis and simulation tool.

From the mechanical VeCS-2 design principles, the following VeCS element was created. We used one differential pair per VeCS, where the rest of the element is shielded in a top view representation (Figure 9).

We created an eight-layer stackup targeting a 95-ohm differential impedance, as shown in the 3D view in Figure 10. When we perform the second route, we clean the bottom section, create the vertical traces, and remove the stub on the top of the VeCS traces. This can all be done in one cycle. Layer 3 and 7 are the signal layers, and the others are reference layers.

The first objective in the analysis was to perform a time-domain simulation and tune the trace width and anti-pad sizes such that we create the smallest distortion possible. We started with a traditional capacitive response as expected form a layer transition (e.g., via through hole) and began to tune the VeCS element such that we got close to a flat line. More modification changed the response from a capacitive (red and blue lines) to an inductive (green line) element, as is shown in Figure 11.

We used the results of the TD simulation to perform the 30 Gb/s bitstream eye-diagram

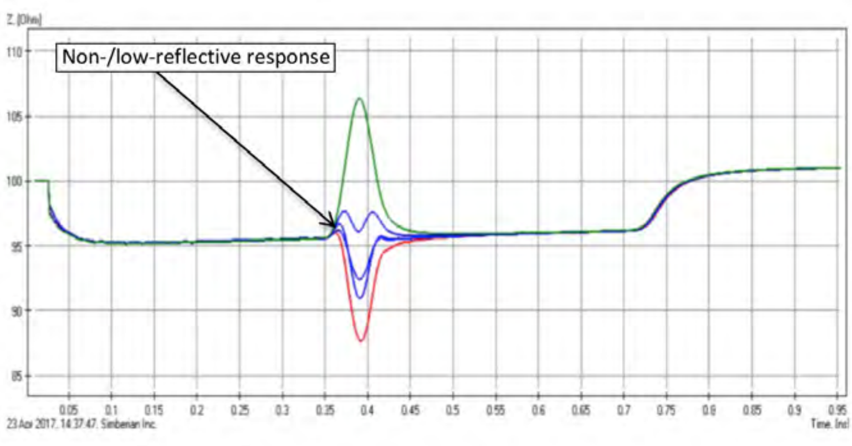


Figure 11: TDR response as a function of the variable vertical trace widths.

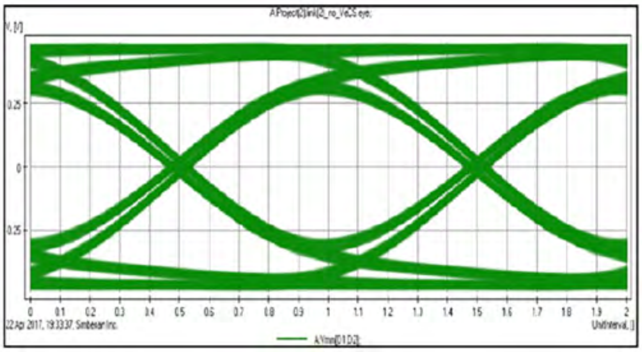


Figure 12: Launch bit pattern.

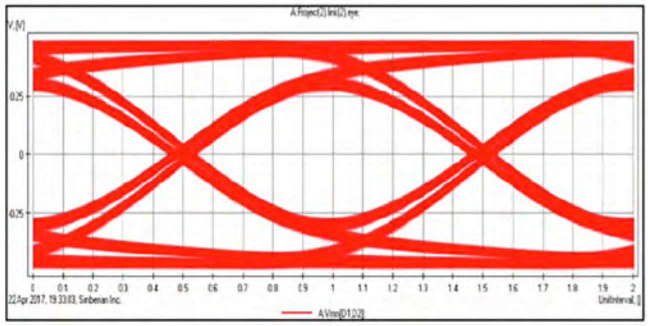


Figure 13: Exit bit pattern.

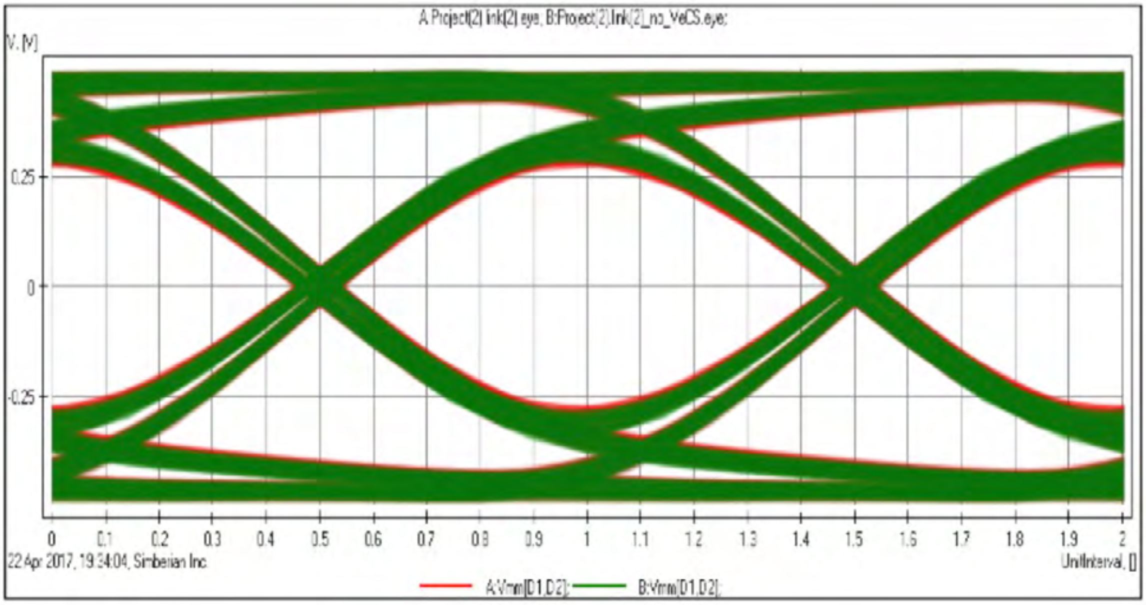


Figure 14: Super-imposed launch and exit eye pattern.

simulation. Figure 12 in green shows the reference transmission line without the tuned VeCS element, and Figure 13 in red the transmission line with a tuned VeCS layer transition (best fit/low-reflection TDR).

In Figure 14, we superimposed the eye-diagram for the transmission line without VeCS (green) on the eye-diagram of the transmission line with the VeCS element (red).

The eye is a bit more closed when using the VeCS; it is a marginal 0.516 V versus 0.539 V for the transmission line without VeCS. Details of the two eye-diagrams are listed in Table 2.

Eye Diagram Parameter	Transmission Line Without VeCS	Transmission Line With VeCS
Eye Level Zero	-0.383742 [V]	-0.377428 [V]
Eye Level One	0.382528 [V]	0.377346 [V]
Eye Level Mean	-0.00182348 [V]	-0.000614657 [V]
Eye Amplitude	0.76627 [V]	0.754774 [V]
Eye Height	0.538959 [V]	0.516313 [V]
Eye Width	0.902439 [UI]	0.897118 [UI]
Eye Opening Factor	0.703354	0.684063
Eye Signal to Noise	6.04823	5.68955
Eye Rise Time (20–80)	0.504472 [UI]	0.512165 [UI]
Eye Fall Time (80–20)	0.505318 [UI]	0.512579 [UI]
Eye Jitter (PP)	0.097561 [UI]	0.102882 [UI]
Eye Jitter (RMS)	0.0250763 [UI]	0.0276838 [UI]

Table 2: Simulation performance data.

Get the spark you need to tune up your sales team.
When an engine doesn't fire on all cylinders, performance is limited. The same goes for your sales team. We provide the spark to get your sales to the next level ...and beyond.
We've helped dozens of companies big and small improve their bottom line.
db MANAGEMENT GROUP
Learn how we can help YOU
207-649-0879
danbbeaulieu@aol.com

Conclusion

By varying the design parameters—such as vertical trace width, antipad, and vertical trace to reference—we can change the impedance and match it with the impedance of the horizontal traces, creating a non-reflective and lower loss transition. We have not addressed all parameters in this short article, but we will do that in the following issues by adding actual measurement data.

In the meantime, more products are built in various programs to determine reliability and match measurements with the simulation models. In Figure 15, a section of a deep VeCS-2 slot demonstrates its capability. This is part of the HDPUG program.

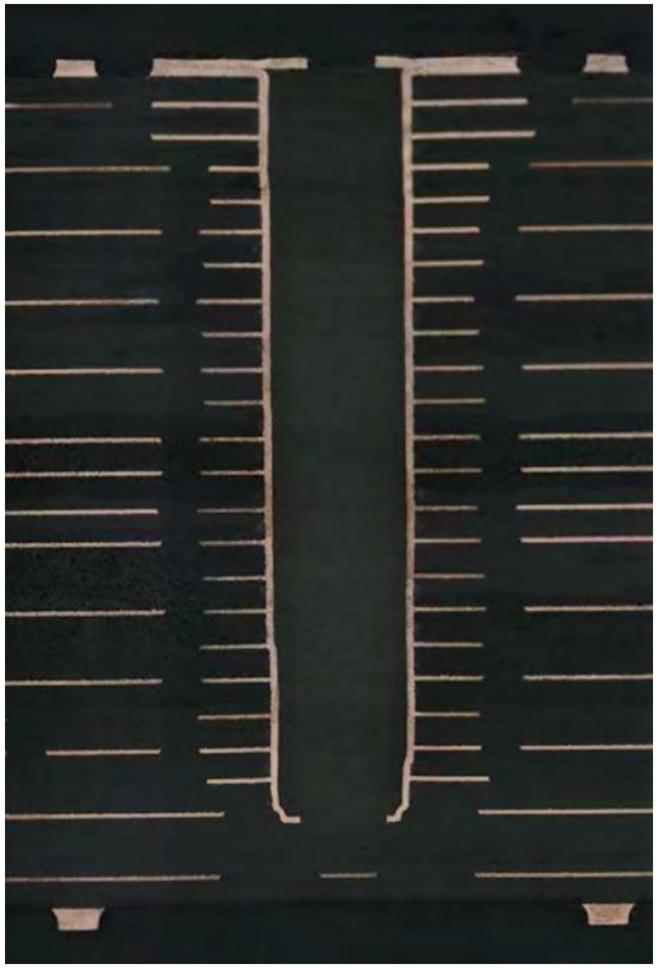


Figure 15: A section of a deep VecS-2 slot.

Acknowledgements

Thanks to Yuriy Shlepnev of Simberean for the simulation work. I also want to thank the WUS PCB China R&D team for all the great development work performed on VeCS. PCB007

Joan Tourné is CEO of NextGIn Technology BV.

RTW SMTAI 2019: EPTAC Continues Its Expansion

Andy Shaughnessy and Brenda Clunie, Vice President of Operations for EPTAC, discuss the company's current growth plans and how its training facilities have expanded across the US and Canada to serve the needs of OEMs who are bringing in new, younger talent. The turnover in tribal knowledge is driving young people to catch up and become empowered.

Clunie said that they are continuously adding instructors and training partners to bring training as near to the customers as they can.

Click on the image to watch the interview.

1 Trouble in Your Tank: Via Hole Filling and Plugging, Part 1 ▶


Michael Carano

High-density interconnect (HDI) demands that vias that do not contain component leads be plugged with either a polymeric paste or electroplated copper. In this column series, Michael Carano talks about the technology drivers for via filling/plugging in the context of HDI.

2 Additive Electronics Conference Set for October 24, 2019 ▶


Tara Dunn

Tara Dunn, president of Omni PCB and I-Connect007 columnist, and Lenora Clark, director of autonomous driving and safety technology at MacDermid Alpha Automotive, discuss the upcoming Additive Electronics Conference in San Jose, California, on October 24, 2019; the impetus and motivation behind the conference; and who can benefit the most from attending.

3 Dana on Data: IPC-2581 Intelligent Bi-directional Data Flow ▶


Dana Korf

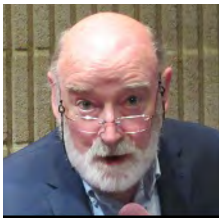
The IPC Consortium is nearing completion of transferring notes on drawings and working with IPC on converting key IPC specifications into attributes that can be automatically loaded into CAD and CAM systems.

4 EPTE Newsletter: PCB Market Trends in Taiwan ▶


Dominique Numakura

PCB monthly shipments for the first three quarters show some positive growth compared to the same periods of the previous year. However, growth was stagnant during October and November and declined significantly in December.

5 The Institute of Circuit Technology Autumn Seminar ▶


Bill Wilkie

On September 19, a multitude of Institute of Circuit Technology (ICT) fellows, members, and associates gathered for the Institute's autumn seminar, which was organised and hosted by ICT Technical Director Bill Wilkie.

6 Standard of Excellence: Great Partnerships Are Forged in Adversity ▶


Anaya Vardya

When you're in the PCB business, the one thing you can count on is that something will go wrong; it always does. Here are six steps to not only help and support your vendor through problematic times but also use the adversity as an opportunity to develop a deeper and more productive relationship.

7 One World, One Industry: Maintaining a Strong Economy Through Workforce Development ▶


John Mitchell

As members of the electronics industry, we work in a quickly evolving technological field illustrated by continual change and advancement. We must have a workforce that is ready to meet these advancements.

8 IPC Designs for Excellence in Conjunction With PCB Carolina ▶


Dale Lee

IPC is offering a day-long technical education seminar on design for excellence (DFX), design for manufacturing (DFM), design for reliability (DFR), and design for assembly (DFA) in conjunction with PCB Carolina on November 12, 2019, in Raleigh, North Carolina, at the McKimmon Conference and Training Center.

9 It's Only Common Sense: Great Customer Service Stories ▶


Dan Beaulieu

We all love wonderful customer service stories. Stories of human kindness and great customer service are everywhere. In a way, customer service is where humanity supersedes business as usual.

10 New IPC Report Shows North American PCB Industry Turnaround ▶

For the first time in five years, domestic PCB production grew in 2018. North America's PCB market also grew last year by nearly 8%, solidifying the turnaround that began in 2017.

Career Opportunities

Sr. PCB Designer—Mentor Xpedition

Freedom CAD is a premier PCB design service bureau with a talented team of 30+ dedicated designers providing complex layouts for our enviable list of high-tech customers. Tired of the commute? This is a work-from-home, full-time position with an opportunity for additional compensation for overtime work at time and a half.

Key Qualifications

- EXPERT knowledge of Xpedition VX 2.x
- Passionate about your PCB design career
- Skilled at HDI technology
- Extensive experience with high-speed digital, RF, and flex and rigid-flex designs
- Experienced with signal integrity design constraints encompassing differential pairs, impedance control, high speed, EMI, and ESD
- Excellent team player who can lead projects and mentor others
- Self-motivated with the ability to work from home with minimal supervision
- Strong communication, interpersonal, analytical, and problem-solving skills
- Other design tool knowledge is considered a plus (Altium, Allegro, PADS)

Primary Responsibilities

- Design project leader
- Lead highly complex layouts while ensuring quality, efficiency, and manufacturability
- Handle multiple tasks and provide work leadership to other designers through the distribution, coordination, and management of the assigned workload
- Ability to create from engineering inputs, board mechanical profiles, board fabrication stackups, detailed board fabrication drawings and packages, assembly drawings, assembly notes, etc.

apply now

Your Dreams, Our Challenge

TECHNICAL SALES ENGINEER

RIGID/FLEX CIRCUITRY OR FLEX CIRCUITRY

AGC NELCO AMERICA, INC. is a global advanced materials company which develops and manufactures high-technology digital and RF/microwave printed circuit materials principally for the telecommunications and internet infrastructure and high-end computing markets.

ESSENTIAL RESPONSIBILITIES

- Training in the area of high-speed materials will be provided; the successful candidate will promote AGC high frequency materials.
- Visit companies that could employ flex circuitry or rigid/flex circuitry to promote AGC solutions/materials.
- Build an OEM customer base for AGC flexible materials; support design work around AGC materials.
- Visit fabricators building high-speed or high temperature rigid/flex circuitry.
- Participate in industry consortium, assist in publication of technical articles.
- Work from a home-based office; relocation not required.
- Domestic US travel required.

QUALIFICATION REQUIREMENTS

- BS degree in a relevant technical discipline.
- Industry expert in either rigid or flex circuitry design.
- Extensive experience in the high-speed cable and connector industry.
- Familiarity with the high-speed cable & connector industry and the customers that are converting designs to high speed flex circuitry.
- The candidate could come from the rigid/flex fabricator base and have familiarity with the manufacture and design of multilayer rigid/flex circuitry.

Please forward your resume to:
HR@AGC-Nelco.com

apply now

Career Opportunities

NW Regional Quality Manager

Westak of Oregon is one of the premier manufacturers of military, aerospace and commercial printed circuit boards. We're an Equal Opportunity Employer, committed to hiring a diverse workforce in the beautiful state of Oregon, an economic force in technology. We offer a competitive salary, including comprehensive benefits and a relocation package for those who qualify.

As our NW Regional Quality Manager, you will be responsible for managing and leading the QA department to support the manufacture of quality product.

TASKS:

- Manage quality engineers and technicians
- Manage the Standard Inspection Procedure & Calibration Program
- Manage our internal/external CA/PA system
- Manage the RMA process
- Manage the ISO Quality System, including the Internal Audit System
- Manage the IPC, military, Bellcore and customer specs
- Measure quality performance
- Employee training, certification, and performance reviews
- Plan & coordinate audits to ensure controls are in place and maintained to continuously improve product yield
- Set QA compliance objectives
- Other duties as assigned

REQUIREMENTS:

- 5 years managerial experience in PCB operation
- Technical degree or equivalent experience
- In-depth understanding of IPC specifications, military specifications and Bellcore requirements
- SPC, SQC and DOE training/experience
- Able to analyze non-conformance of product
- Effective verbal and written skills
- Must be a U.S. person, as defined by ITAR regulations: a U.S. citizen, U.S. permanent resident, political asylee, or refugee
- Able to prioritize and meet deadlines

Send your resume to hr@westak.com.

apply now

OEM Sales Manager

Chicago/Home-Office-Based

Want to advance your career by joining a globally successful and growing world-class CCL manufacturer and help drive that success? We are seeking to hire an OEM sales manager to grow and manage key customer accounts with OEM's and Tier 1 manufacturers in the USA, focusing on Ventec's core market segments: mil/aero, automotive, and medical, offering a full range of high-reliability materials, including polyimide, IMS, and thermal management products.

Skills and abilities required for the role:

- Non-negotiable: Drive and tenacity!

Required:

- 7 to 10 years' experience in the PCB industry in engineering and/or manufacturing
- Detail-oriented approach to tasks
- Ability to manage tasks and set goals independently as well as part of a team
- Knowledge of MS office products

Full product training will be provided.

This is a fantastic opportunity to become part of a successful brand and leading team with excellent benefits.

Please forward your resume to jpattie@ventec-usa.com and mention "Technical Sales Engineer—Chicago" in the subject line.

apply now

Career Opportunities

Career Opportunities

Career Opportunities

TAIYO AMERICA

Assistant Department Manager, Operations, Carson City, NV

This is an entry-level professional management trainee position. Upon completion of a 1-2-year apprenticeship, this position will be elevated to facility/operations manager. Primary functions during training: shadow incumbent staff managers to learn and understand the operations and personnel of the operations department. This position will train and learn, develop, implement, and coordinate strategies related directly to the manufacture of Taiyo products. Additionally, this position will be learning all about the facility, environment, and health and safety functions. Eventually, this position will be responsible for the administration, security and maintenance of the facility and warehouse

Required Experience/Education:

- 4-year college degree in industrial engineering or another similar science discipline combined with work experience in ink or coatings manufacturing
- Ability to read, analyze, and interpret common scientific and technical journals, financial reports, and legal documents
- Ability to respond to inquiries or complaints from customers, regulatory agencies, or members of the business community
- Ability to develop and implement goals, objectives, and strategies
- Ability to effectively present information to top management, public groups, and/or boards of directors
- Ability to apply principles of logical or scientific thinking to a wide range of intellectual and practical problems
- Knowledge of governmental safety, environmental, transportation regulations/laws

Preferred Skills/Experience:

- Bilingual (Japanese/English)
- Toyota Production System (TPS)

Working Conditions:

- Occasional weekend or overtime work

See complete job listing for more information.

apply now

eptac

TRAIN. WORK SMARTER. SUCCEED.

Become a Certified IPC Master Instructor

Opportunities are available in Canada, New England, California, and Chicago. If you love teaching people, choosing the classes and times you want to work, and basically being your own boss, this may be the career for you. EPTAC Corporation is the leading provider of electronics training and IPC certification and we are looking for instructors that have a passion for working with people to develop their skills and knowledge. If you have a background in electronics manufacturing and enthusiasm for education, drop us a line or send us your resume. We would love to chat with you. Ability to travel required. IPC-7711/7721 or IPC-A-620 CIT certification a big plus.

Qualifications and skills

- A love of teaching and enthusiasm to help others learn
- Background in electronics manufacturing
- Soldering and/or electronics/cable assembly experience
- IPC certification a plus, but will certify the right candidate

Benefits

- Ability to operate from home. No required in-office schedule
- Flexible schedule. Control your own schedule
- IRA retirement matching contributions after one year of service
- Training and certifications provided and maintained by EPTAC

apply now

Career Opportunities

INSULECTRO

Technical Account Manager Chicago/Minneapolis

Insulectro, the largest national distributor of printed circuit board materials, is seeking a talented sales superstar for a Technical Account Manager role based out of either our Chicago or Minneapolis office. This role will focus on maintaining the existing customer base and developing new business within the assigned territory in both the printed circuit board and printed electronics industries. We are looking for the perfect fit of education, experience, and attitude that matches our company culture and enhances the service level to our customers.

Qualifications:

- A self-motivated business professional who is driven to succeed with a minimum of 3 years outside sales experience in the PCB or PE industry
- Proven sales/business development record
- Excellent communication and interpersonal skills
- OEM and electronic assembly experience is a plus

We offer:

- Competitive salary and commission plan with a comprehensive benefits package
- A fun, high-energy company with an entrepreneurial spirit
- A great group of people to work with!

apply now

CML

Analyst Programmer, Hong Kong

We believe in caring about our people because they are our greatest asset. CML works with multicultural stakeholders daily to achieve more and bring them the best solutions. That's why we continuously invest in optimizing our culture and focus on providing our team with opportunities to develop their skills (e.g., through professional coaching to achieve their highest potential).

The analyst programmer will assist the IT and ERP manager in Hong Kong to support the company's BI systems, ERP systems, and other related IT-landscape applications.

In addition, this post will participate in system development projects and provide support including, but not limited to, user requirement collection and analysis, user training, system documentation, system support and maintenance, enhancement, and programming.

- Develop and enhance related IT systems and applications
- Prepare functional specifications
- Transfer the relevant business and interface processes into IT systems and other applications to get a maximum automation degree and prepare all required business reports
- Conduct function testing and prepare documentation
- Manage help desk/hotline service

CML is a leading provider of printed circuit boards. We develop tailor-made sourcing and manufacturing solutions for our customers worldwide with strong partnerships and reliable connections.

apply now

Career Opportunities

Career Opportunities

Career Opportunities

Career Opportunities

ZENTECH

Zentech Manufacturing: Hiring Multiple Positions

Are you looking to excel in your career and grow professionally in a thriving business? Zentech, established in Baltimore, Maryland, in 1998, has proven to be one of the premier electronics contract manufacturers in the U.S.

Zentech is rapidly growing and seeking to add Manufacturing Engineers, Program Managers, and Sr. Test Technicians. Offering an excellent benefit package including health/dental insurance and an employer-matched 401k program, Zentech holds the ultimate set of certifications relating to the manufacture of mission-critical printed circuit card assemblies, including: ISO:9001, AS9100, DD2345, and ISO 13485.

Zentech is an IPC Trusted Source QML and ITAR registered. U.S. citizens only need apply.

Please email resume below.

apply now

BLACKFOX
Premier Training & Certification

IPC Master Instructor

This position is responsible for IPC and skill-based instruction and certification at the training center as well as training events as assigned by company's sales/operations VP. This position may be part-time, full-time, and/or an independent contractor, depending upon the demand and the individual's situation. Must have the ability to work with little or no supervision and make appropriate and professional decisions. Candidate must have the ability to collaborate with the client managers to continually enhance the training program. Position is responsible for validating the program value and its overall success. Candidate will be trained/certified and recognized by IPC as a Master Instructor. Position requires the input and management of the training records. Will require some travel to client's facilities and other training centers.

For more information, click below.

apply now

Events Calendar

AltiumLive 2019: Annual PCB Design Summit ▶
October 21–23, 2019
Frankfurt, Germany

Medical Design & Manufacturing (MD&M) Minneapolis ▶
October 23–24, 2019
Minneapolis, Minnesota, USA

TPCA SHOW 2019 ▶
October 23–25, 2019
Taipei, Taiwan

China Electronics Fair ▶
October 30–November 1, 2019
Shanghai, China

IPC Electronics Materials Forum ▶
November 5–7, 2019
Minneapolis, Minnesota, USA

International Test Conference ▶
November 12–14, 2019
Washington D.C., USA

productronica 2019 ▶
November 12–15, 2019
Munich, Germany

Electric & Hybrid Aerospace Technology Symposium ▶
November 13–14, 2019
Cologne, Germany

2019 International Electronics Circuit Exhibition (Shenzhen) ▶
December 4–6, 2019
Shenzhen, China

IPC APEX EXPO 2020 ▶
February 1–6, 2020
San Diego, California, USA

Additional Event Calendars

IPC

SMTA Surface Mount Technology Association

iNEMI

pcb 007

Coming Soon to *PCB007 Magazine:*

NOVEMBER: From My Point of View
Sometimes, the best view into an industry or a community is through individual voices. In this issue, we talk to members of our business community, gathering and sharing their voices and perspectives.

DECEMBER: What You Need to Learn
No matter your age or experience level, to move technology forward, we all need to be continuous learners. In this issue, we highlight the highest impact topics to further your expertise.

PUBLISHER: **BARRY MATTIES**
barry@iconnect007.com

SALES MANAGER: **BARB HOCKADAY**
(916) 608-0660; barb@iconnect007.com

SALES: **ANGELA ALEXANDER**
(408) 489-8389; angela@iconnect007.com

MARKETING SERVICES: **TOBEY MARSICOVETERE**
(916) 266-9160; tobey@iconnect007.com

MANAGING EDITOR: **NOLAN JOHNSON**
(503) 597-8037; nolan@iconnect007.com

CONTRIBUTING EDITOR: **PATRICIA GOLDMAN**
(724) 299-8633; patty@iconnect007.com

CONTRIBUTING TECHNICAL EDITOR: **DAN FEINBERG**
baer@iconnect007.com

TECHNICAL EDITOR: **PETE STARKEY**
+44 (0) 1455 293333; pete@iconnect007.com

ASSOCIATE EDITOR: **KIERSTEN ROHDE**
kiersten@iconnect007.com

CONTRIBUTING TECHNICAL EDITOR: **HAPPY HOLDEN**
(616) 741-9213; happy@iconnect007.com

PRODUCTION MANAGER: **SHELLY STEIN**
shelly@iconnect007.com

MAGAZINE LAYOUT: **RON MEOGROSSI**

AD DESIGN: **SHELLY STEIN, MIKE RADOGNA, TOBEY MARSICOVETERE**

INNOVATIVE TECHNOLOGY: **BRYSON MATTIES**

COVER: **SHELLY STEIN**

COVER IMAGE: **ADOBESTOCK©HELIVIDEO**

PCB007 MAGAZINE

PCB007 MAGAZINE®
is published by BR Publishing, Inc.,
942 Windemere Dr. NW, Salem, OR 97304

October 2019, Volume 9, Number 10
PCB007 MAGAZINE is published monthly,
by BR Publishing, Inc.

ADVERTISER INDEX