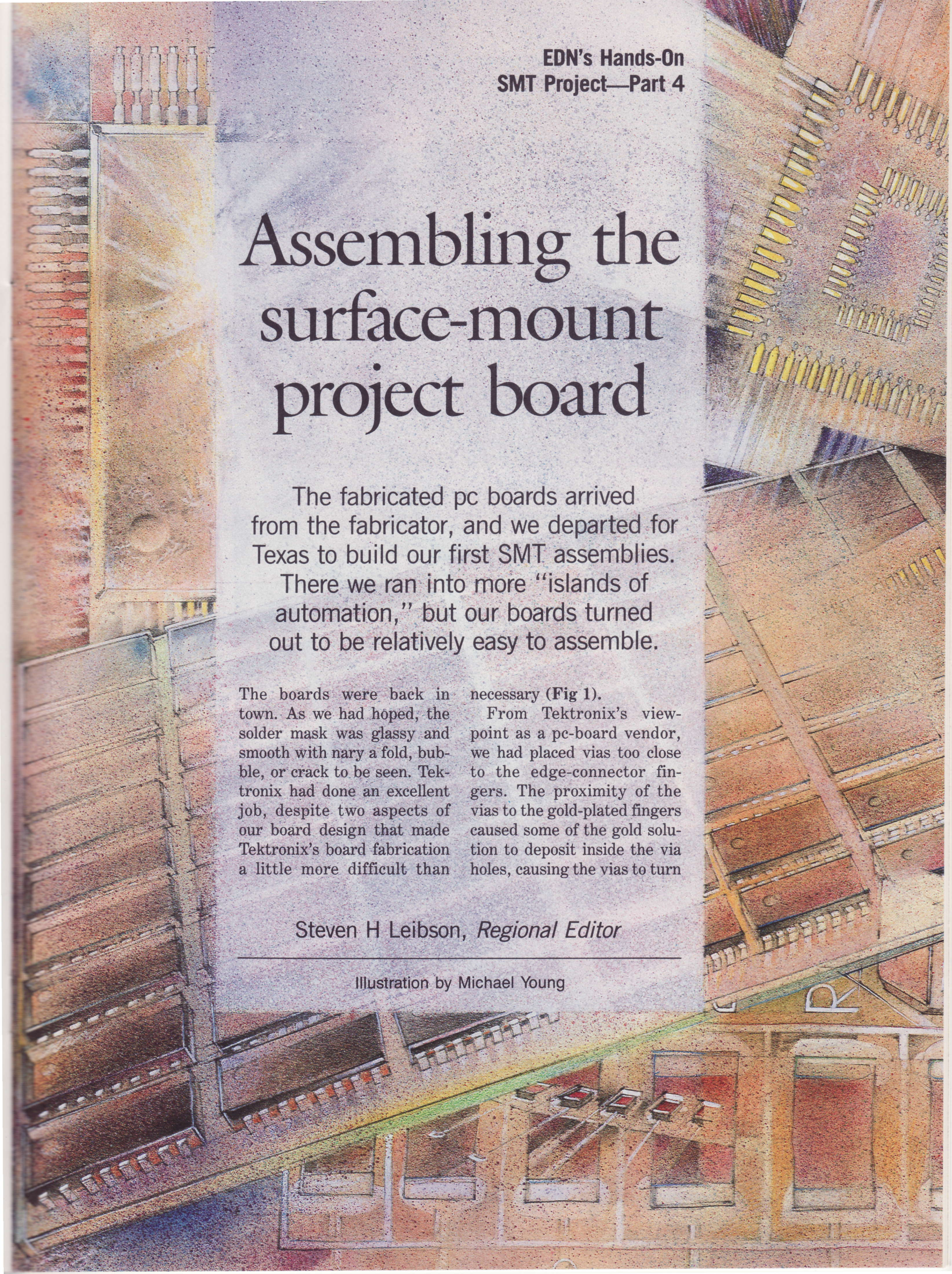


EDN's Hands-On SMT Project—Part 4

Assembling the surface-mount project board

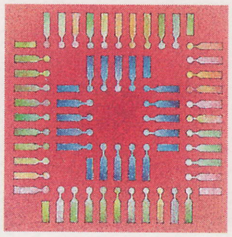
The fabricated pc boards arrived from the fabricator, and we departed for Texas to build our first SMT assemblies. There we ran into more "islands of automation," but our boards turned out to be relatively easy to assemble.

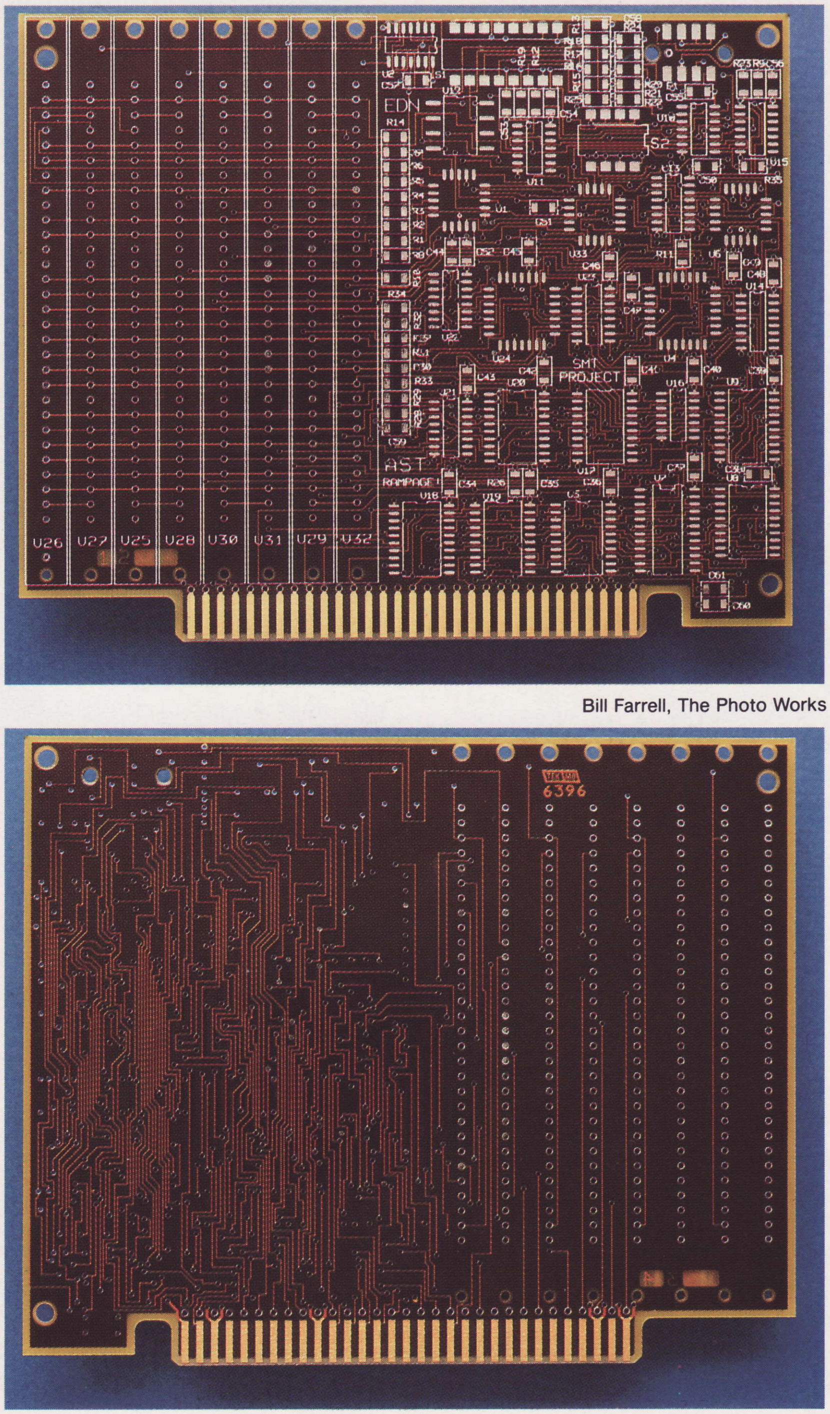
The boards were back in town. As we had hoped, the solder mask was glassy and smooth with nary a fold, bubble, or crack to be seen. Tektronix had done an excellent job, despite two aspects of our board design that made Tektronix's board fabrication a little more difficult than necessary **(Fig 1)**.

From Tektronix's viewpoint as a pc-board vendor, we had placed vias too close to the edge-connector fingers. The proximity of the vias to the gold-plated fingers caused some of the gold solution to deposit inside the via holes, causing the vias to turn

Steven H Leibson, *Regional Editor*

Illustration by Michael Young





(a)

Bill Farrell, The Photo Works

(b)

Fig 1—Our fabricated pc board exhibits many of the attributes we were taught to incorporate in SMT board designs. Silk-screen component designators on the top of the board **(a)** mark and label the position of each component. We designed every pad pattern used. The bottom of the board **(b)** shows evidence of our manual editing. We put every diagonal trace on the board by hand because Cadnetix's automatic routers only generate vertical and horizontal traces. We believe the diagonal traces plus the manual redistribution we performed to even out the trace spacing enhanced the pc board's manufacturability.

black. Tektronix says its customers don't like this purely cosmetic defect. Although we didn't object to the board's appearance, we could have moved the vias another 20 or 30 mils away from the edge-connector fingers if we had known about this problem earlier.

We also erred by placing two bypass capacitors below the topmost excursion of the edge-connector fingers. This component placement forced Tektronix to take precautions that prevented those component pads from becoming gold plated. Gold plating poisons a surface-mount pad and destroys its solderability, just as a solder mask or silk-screen ink does. Had we known about this problem, we would have omitted the two offending bypass capacitors. Because these two problems were the only complaints Tektronix made about our board, we felt that all of our SMT training and preparation was starting to pay off.

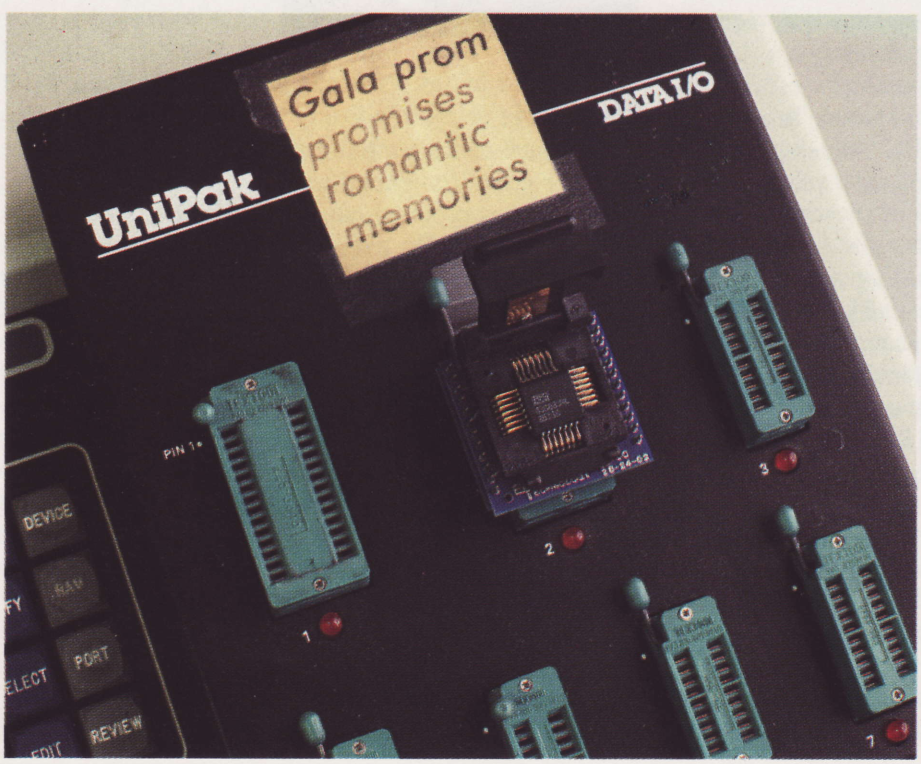
Tektronix also commented that our pc board should be fabricated

Bill Farrell, The Photo Works

This solder screen allowed a screen printer to deposit solder paste on our board's SMT pads but not on the pads for through-hole devices.

The device programmer we used at Cadnetix did not have SMD sockets, so we used three Adapt-A-Socket adapters from Emulation Technology to convert the device programmer's DIP sockets to PLCC sockets.

with only one image per panel because of the small features and tight tolerances of our design. We left the panelization of the board up to the CAD operators at Tektronix, and they built our prototype boards with six images/panel. Apparently, the multiple-image panels created some problems when the manufacturing people tried to hold tolerances for all six boards on each panel through the fabrication process. However, the company overcame these problems because all of the pc boards worked as designed.

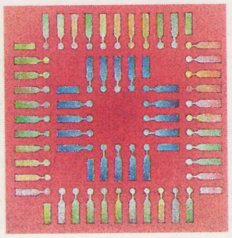


Steven H Leibson

Now that we had the pc board and the parts, we were almost ready to assemble our project board. But before we could travel down to Texas Instruments' SMT Center for the assembly phase of the project, we had to program the PLDs and PROMs. AST provided printouts of the PLD equations and PROM listings, so once again we invited Murphy to join us on our project by typing this information by hand into our IBM PC's word processor. We double checked those transcriptions but still felt uneasy about manually transferring this information.

The Data I/O Model 29 we used at Cadnetix to program our devices accepts the ASCII-hexadecimal PROM files supplied by AST, but the equations for our PLDs had to be compiled into JEDEC-compatible download files before we could transfer them to the programmer. We compiled these equations using an IBM PC and Personal CAD Systems' CUPL compiler. Initially, we attempted to use an old version of CUPL that we'd had for a few years, but AST's PLD listings included statements that only later releases of the CUPL compiler could interpret. A frantic call to Personal CAD Systems netted us the necessary software, and the compilation proceeded without a hitch.

We drove over to Cadnetix with our PROMs, PLDs, the compiled files on a floppy disk, and our DIP-to-PLCC Adapt-A-Sockets from Emulation Technology. After the usual 15 minutes of fooling around with the RS-232C link between the Data I/O programmer and an IBM PC to get the two machines talking, we downloaded each file and burned our parts. Then we were ready for Texas Instruments.



SMT, Texas style with all the fixin's

We arrived at Texas Instruments' SMT Center in Houston on November 3, 1986, a little more than a year after starting the SMT project. Dr Charles Hutchins, the SMT Center's manager, told us to allow three days to perform the assembly, but we allocated four days as a precaution. The job required only two days. One reason the work went quickly was that Jerry Pickert, an intern at the SMT Center, had made some preparations for our arrival. Using our parts list, he had bolted the appropriate parts carriers and feeders in place on the Quad Systems (Horsham, PA) Model 34 placement machine. (That machine is now called the Quad Start.) We loaded our parts onto the machine as soon as we arrived.

Because we were only assembling 10 boards, we didn't use hard tooling to hold the board in position on the placement machine. Instead, we used a carpenter's square for board alignment and fastened both the carpenter's square and our pc boards to the placement machine using transparent, double-sided adhesive tape. We put an additional layer of tape on the top surface of the board to hold components in position during training. (Don't laugh, the tape worked very well.) With everything in position, we started programming the machine with part locations.

Every design engineer should experience programming a place-

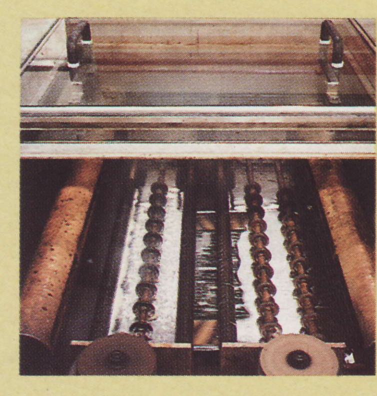
Fabricating SMT pc boards

The Tektronix Printed Circuit Board Division (Tek PCB) became a merchant vendor of pc boards in April 1984. Previously, this division only supplied boards to other Tektronix divisions. We selected Tek PCB to fabricate our SMT project board for several reasons, including the company's familiarity with Cadnetix-format plot tapes.

Tek PCB offered us all of the pc-board manufacturing processes we wanted to use for our design. The 8-mil-trace with 8-mil-space design rules we used posed a simple problem for a company that can currently accommodate 2- or 3-mil conductors and 4- or 6-mil spaces. In addi tion, Tek PCB offered us SMOBC (solder mask over bare copper) and HAL (hot-air-leveled) solder fabrication processes. Both of these processes are ideal for SMT boards.

An SMOBC process applies solder mask to a pc board before plating. Solder mask generally adheres to bare copper better than it does to plated metal. The superior adhesion resists separation during reflow soldering and prevents solder on the surface-mount pads from draining off the pad by running down a trace conductor. Tek PCB applied a dry-film solder mask to our pc board using a double-burn photographic process that prevented the solder mask from encroaching on any of our SMD pads. If solder mask covers a pad, even partially, the solderability of that pad is destroyed. We had no soldering problems attributable to our pc boards.

After applying the solder mask, our boards were tinned in Tek PCB's HAL solder machine. During this process, a slurry of liquid solder covers bare-metal portions of a pc board, and air knives remove the excess molten metal. The HAL process produces an extremely flat surface on the pc board. We measured less than a 1-mil height difference between the tinned pads and the solder mask on our board. This smooth surface encourages the SMDs to sit flat on the board, reducing manufacturing defects from component tilt or movement during the reflow-soldering operation.

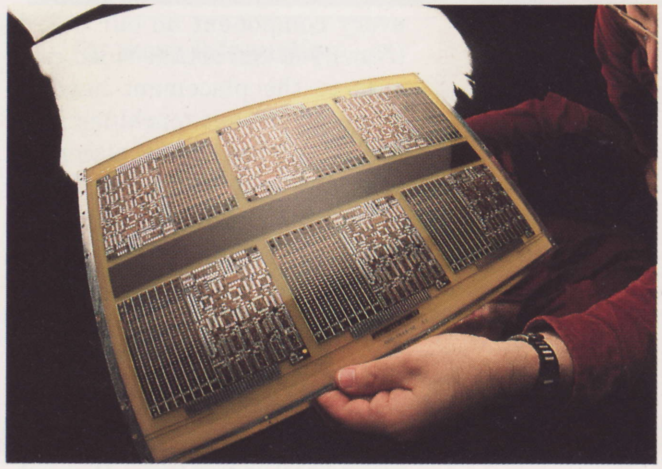


The hot-air-leveled (HAL) soldering process Tektronix used to coat our fabricated pc boards with solder is ideal for SMT assembly. Air knives remove the excess solder and leave a uniform surface that encourages SMDs to stay put during reflow soldering.

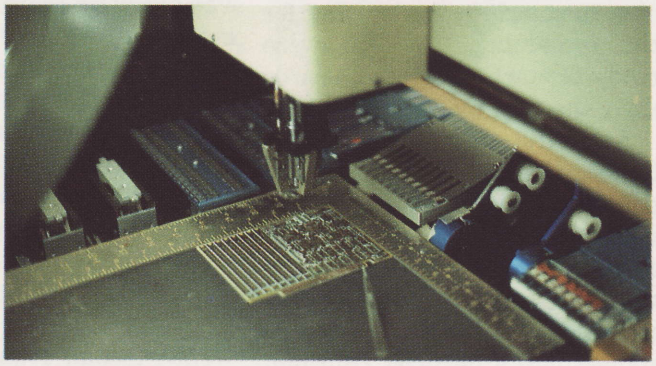
Although our design did not require them, Tek PCB can also fabricate a pc board that has blind or buried vias. Such vias do not bore completely through a pc board and are especially useful for SMT designs because they can leave one surface of the board relatively free of vias. That uncrowded surface can accommodate many more 1-layer SMD pads.

ment machine by hand at least once. It gives you a much greater appreciation for the people who do such tedious work regularly. You must program two positions for each part placed on the pc board. The first position tells the placement machine where to obtain a part from a feeder or part carrier, and the other position describes where the machine is to place the part. Each site is located in a 5-dimensional space: X, Y, Z, rotation, and placement-head number. The placement-head number specifies which of four vacuum chucks the machine should use to pick up the SMD.

Our board contains more than 100 SMDs, and we had to show the placement machine where to pick up and where to put each one. Of course, if the placement machine could have used the component location information directly from our CAD tapes, we would have avoided this training. But once again we stepped in and effected the transfer of information between one computer-based tool and the next by hand.


Bill Farrell, The Photo Works

Although our prototype boards were fabricated six to a panel, Tektronix believes it would achieve optimal manufacturing yields by fabricating only one board per panel.


Steven H Leibson

A carpenter's square served as the only tooling we needed for pc-board alignment on the placement machine. We covered both sides of the board with transparent, double-sided adhesive tape while training the machine to place components. The tape on the bottom surface held the board in position on the machine, and the tape on the top surface held the components in place during training.

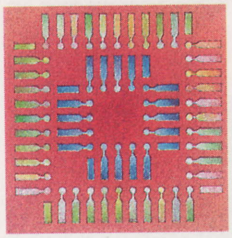
Ergonomics 101 for assembly tools

The nonergonomic design of the placement machine's control panel complicated our task. The control panel is a small, handheld box about the size of a portable DMM, and it's attached to the placement machine by a coiled cord. You move the machine's placement head using buttons (which resemble a computer keyboard's cursor keys) on the control panel's nontactile, membrane keyboard. We found the arrangement of the placement-head control buttons to be nonintuitive, and we often started the placement head moving in the wrong direction by pressing the wrong button. We would have preferred to use a joystick control for this operation.

You need many sample parts to program the placement machine, and we experienced component shortages because we lost several SMDs during the training session. Passive SMDs frequently jumped out of the vibratory feeders and irretrievably dropped into mounting holes on the placement machine's work surface. (The SMT lab provided dummy ICs—with no silicon inside—for training the machine, but we dipped into our project stock for passive SMDs.) Later, during assembly, we ran out of chip resistors on the last pc board. Next time we'd prefer to have a much larger supply of components available for training purposes.

A long way to see a small thing

Another problem we encountered during placement-machine training was caused by the minute size of SMDs. The layout of the placement machine forced us to stand about two feet from the pc board as we tried to align the components with 0.001-in. accuracy. The placement head often obscured our view, forcing us to drop the component on the board, move the placement head away from the component, and then check our positioning accuracy. We repeated this process for



every component on our board. (Now, you can attach video cameras to the placement head on many placement machines for a bird's-eye view of the placement site. Our experience with the placement machine gives us a great appreciation for this feature.)

By early evening on the first day, we had completed the machine's training for all of the passive-component sites and for the locations of some ICs. Mr Pickert spent another two hours that evening to complete the training for the remaining ICs. We were ready to start assembling boards early the next morning.

As a first step in assembling the boards, we set up the screen printer. Dr Hutchins had the solder screen made for us at Microcircuit Engineering Corp, the vendor he normally uses for TI's SMT boards. The solder screen slipped into a deHaart (Burlington, MA) screen printer. We used double-sided adhesive tape again, this time to hold our pc board in place on the screen printer's platen. Next, we had to align the solder screen with the pc board. That process turned out to be more difficult than it appeared, because the adjustment controls on the screen printer allow three degrees of freedom: X, Y, and θ. Lack of solder-screen targets on our pc board also complicated this task.

Some SMDs needed the human touch

We finally lined up the solder screen with a sample pc board and then screen-printed solder paste on all 10 boards. That part of the job took very little time. Then we carried each screened board to the placement machine for the 5-minute operation that put most of the SMDs onto the board. We hand-placed the surface-mount switches and the pin header because the SMT Center didn't have component carriers for those devices. We also added one of the PLCCs to each board by hand because they didn't fit into the component carriers. Apparently, the leads on these particular PLCCs were formed just a

SMT at Texas Instruments

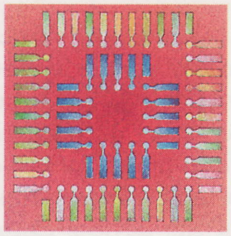
As a vendor and user of SMT ICs, Texas Instruments has a strong interest in furthering surface-mount technology. To that end, the company created a lab called the TI SMT Center to help customers learn about SMT. We assembled the EDN project board at TI's SMT Center. Dr Charles Hutchins heads that lab and manages the company's SIP module assembly facility as well. Because SIPs incorporate SMDs, the lab and assembly line complement each other quite well. The arrangement also worked well for us because each EDN project board incorporates eight 256k×9-bit SIP memory modules.

The SMT Center contains all of the equipment needed to build SMT boards, including screen printers, placement equipment and component feeders for the placement machines, IR and vapor-phase soldering machines, and inspection equipment. The SIP assembly line uses the IR reflow-soldering machine on a regular basis. TI customers can arrange to visit the SMT Center for training through their local TI field sales engineer or distributor.

TI recently expanded the scope of its SIP assembly facility by branching out into a product it has dubbed memory-intensive modules (MIMs). These products incorporate surface-mounted memory devices like SIPs and contain additional devices like address decoders and memory-array drivers to create an entire memory subsystem on a plug-in card. TI builds MIMs for each customer by contract. You can design the module, or the company will develop a design to your specifications.

Jerry S Jones

Dr Charles Hutchins manages Texas Instruments' SMT Center and the SIP-module assembly line in Houston. Next to Dr Hutchins is the Quad Systems Model 34 placement machine we used to assemble our project board.



little larger than normal. The time required to put all of the components on each pc board totaled about six minutes per board.

We planned to run five of the boards through TI's IR reflow-soldering furnace and the remaining five boards through its vapor-phase reflow-soldering unit. However, the SIP manufacturing line was using the IR furnace. Every SMT board requires a different heat profile for the IR soldering operation, so in order to use the IR furnace we would have had to stop the SIP production, change the temperature profile on the furnace, solder our boards, and then reset the profile for the SIP modules. The SIP production people probably wouldn't have appreciated our interruption, so we elected to solder all of the boards using the vapor-phase process.

Inside a vapor-phase, reflow-soldering tank, heaters vaporize an inert, fluorocarbon fluid. As the cold pc board with its load of solder paste and SMDs drops into the tank, it causes some of the fluorocarbon vapor to condense. The vapor releases its heat of vaporization as it condenses and melts the solder paste to form soldered joints.

In less than 10 minutes, we had dunked all 10 boards into the reflow bath, soldering all of the SMDs. After reflow soldering, only the through-hole SIMM sockets remained to be added to our pc boards. We hand-placed eight sockets on each board and ran the boards through a hand-operated wave-soldering machine TI uses to add connectors to a memory board it builds. Because we used this hand-soldering machine with its small solder wave, the only portion of our board exposed to the molten solder was beneath the SIMM sockets. Thus, all of the vias in the control-circuit portion of the board remained open.

Plug those vias

We should have plugged the vias to allow the vacuum hold-down on Hewlett-Packard's pc-board ATE to operate properly. To accomplish the plugging, we should have masked the edge-connector fingers and run the entire board over a solder wave. The fiberglass pc board would have acted as an insulator, preventing the reflowed solder from melting and consequently keeping the SMDs in place during the wave-soldering operation.

We left the vias open unintentionally. In fact, we specifically designed the board to allow solder to enter all of the vias during wave soldering by excluding vias from beneath our passive SMDs. We simply didn't think of the con-

For more information . . .

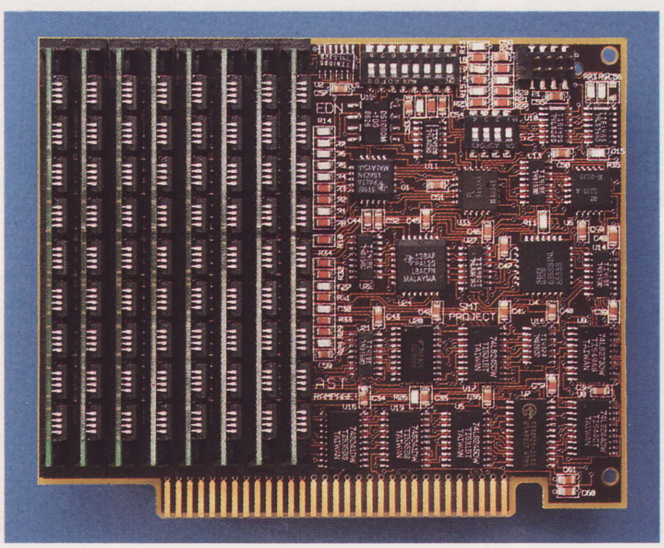
Emulation Technology Inc
422 Ives Terrace
Sunnyvale, CA 94087
(415) 960-0652

Microcircuit Engineering Corp
192 Roncocas Rd
Mount Holly, NJ 08060
(609) 261-1400

Personal CAD Systems
1290 Parkmoor Ave
San Jose, CA 95126
(408) 971-1300

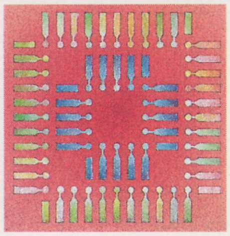
Tektronix Inc
Printed Circuit Board Mfg Div
Box 3000
Forrest Grove, OR 97116
(800) 222-2600

Texas Instruments Inc
Box 1440
Houston, TX 78721
(800) 232-3200



Bill Farrell, The Photo Works

With all the components mounted, the finished EDN project board acquires a gem-like appearance because of the small device geometries.

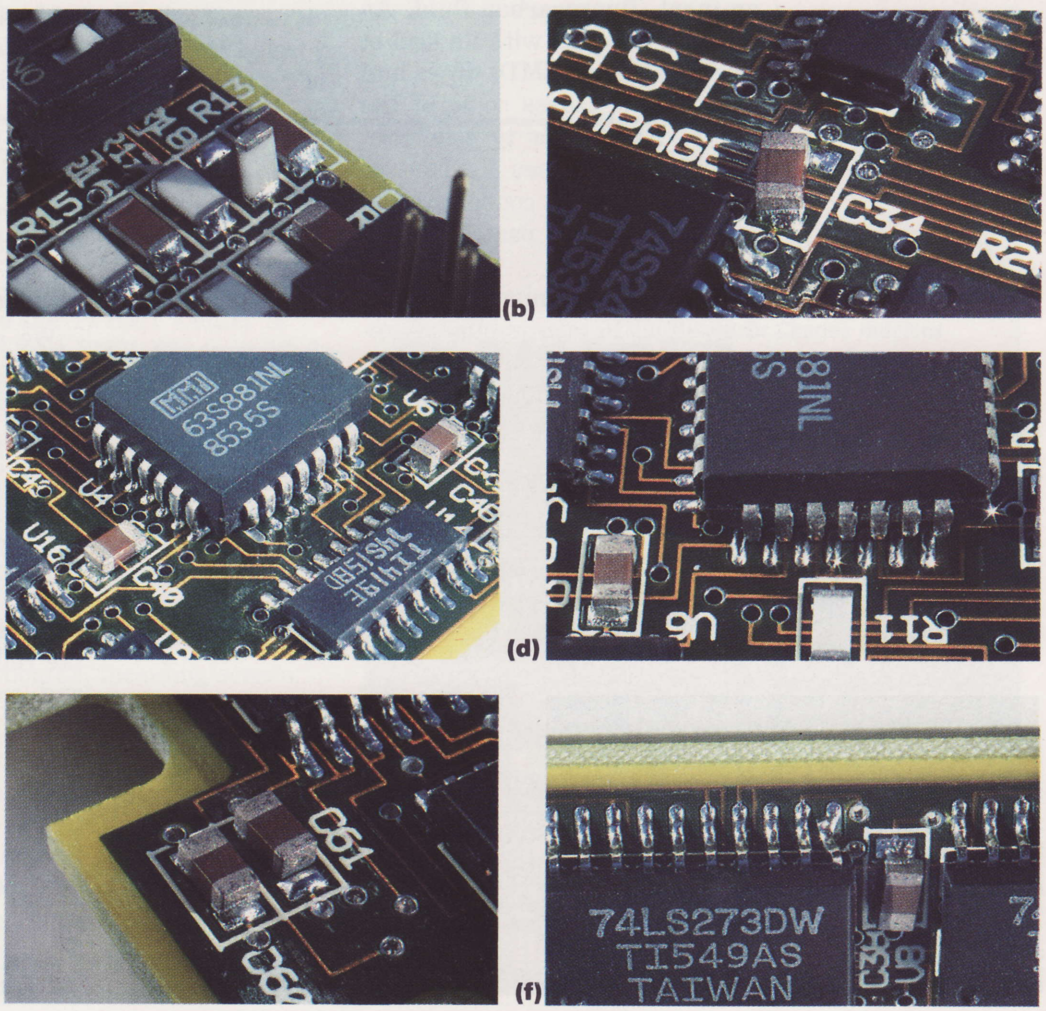


sequences of leaving the vias open when we assembled our boards.

Because we left the vias open, HP had to add a movable lid with mechanical probes that forced the board down onto the tester's bed of nails. The lid increased the test time per board because an operator had to open and close it for each test. The lid also complicated testing because it prevented an operator from flipping the surface-mount switches on our pc board during the test. As a result, we decided to forego testing of those switches on the ATE.

In the final assembly step, we snapped SIP memory modules into all of the SIMM sockets. Then, with our boards completely assembled, we boarded a plane and returned home. We subjected the boards to close visual inspection and found multiple manufacturing defects on eight of the 10 boards **(Fig 2)**. The most common defect was tombstoning of chip resistors, a condition in which one end of a resistor is pulled up off the board, creating an open circuit. We believe the major reason for the tombstoning was improper placement of the devices caused by the use of a chuck designed for 1210 SMDs instead of the 1206 devices we used. (The SMT Center didn't have a 1206 chuck.)

The oversized chuck allowed the devices to move during placement on the board so that both ends of the device were not well inserted into the solder paste. When the solder paste melted during reflow, one end of the resistor experienced more force from the solder's surface tension than the other end did. That differential surface tension caused one end of the device to lift from its pad. None of the chip capacitors tombstoned. We attribute that situation to the extra mass of the capacitor overcoming the ef-



(a) (b) (c) (d) (e) (f)

Fig 2—Our boards contained a rogues' gallery of manufacturing defects, including tombstoning **(a)**, drawbridging **(b)**, open circuits caused by noncoplanar leads **(c)**, and poorly placed SMDs **(d** and **e)**. We also spotted a bent SOIC lead **(f)** that did not cause a problem in this instance but that could have resulted in either an open or short circuit.

Steven Leibson, EDN

fects of surface tension. Our chip capacitors are much heavier than our chip resistors, even though both are 1206-size devices, because the capacitors are much thicker.

We also discovered a few open circuits on some of the PLCC leads that probably were caused by a lack of lead planarity. Remember, all of our PLCCs are programmable devices, and we had subjected them to a lot of handling. We selected PLCCs over SOICs for our programmable devices to avoid lead deformation during handling, but we apparently did not avoid that problem completely.

Two of the boards looked perfect to the unaided eye, so we ripped the lid off our IBM PC, removed the AST Rampage! from its slot, set the switches on our SMT Rampage!, and plugged the short card into the computer. A little apprehensive, we flipped on the power to the computer and waited. In less than 30 seconds, a parity-error message appeared on the computer's display and mocked us. The same event occurred for the other "good" board. Our SMT boards didn't work. That put the ball in HP's court. We needed the ATE board tester to tell us what went wrong. EDN