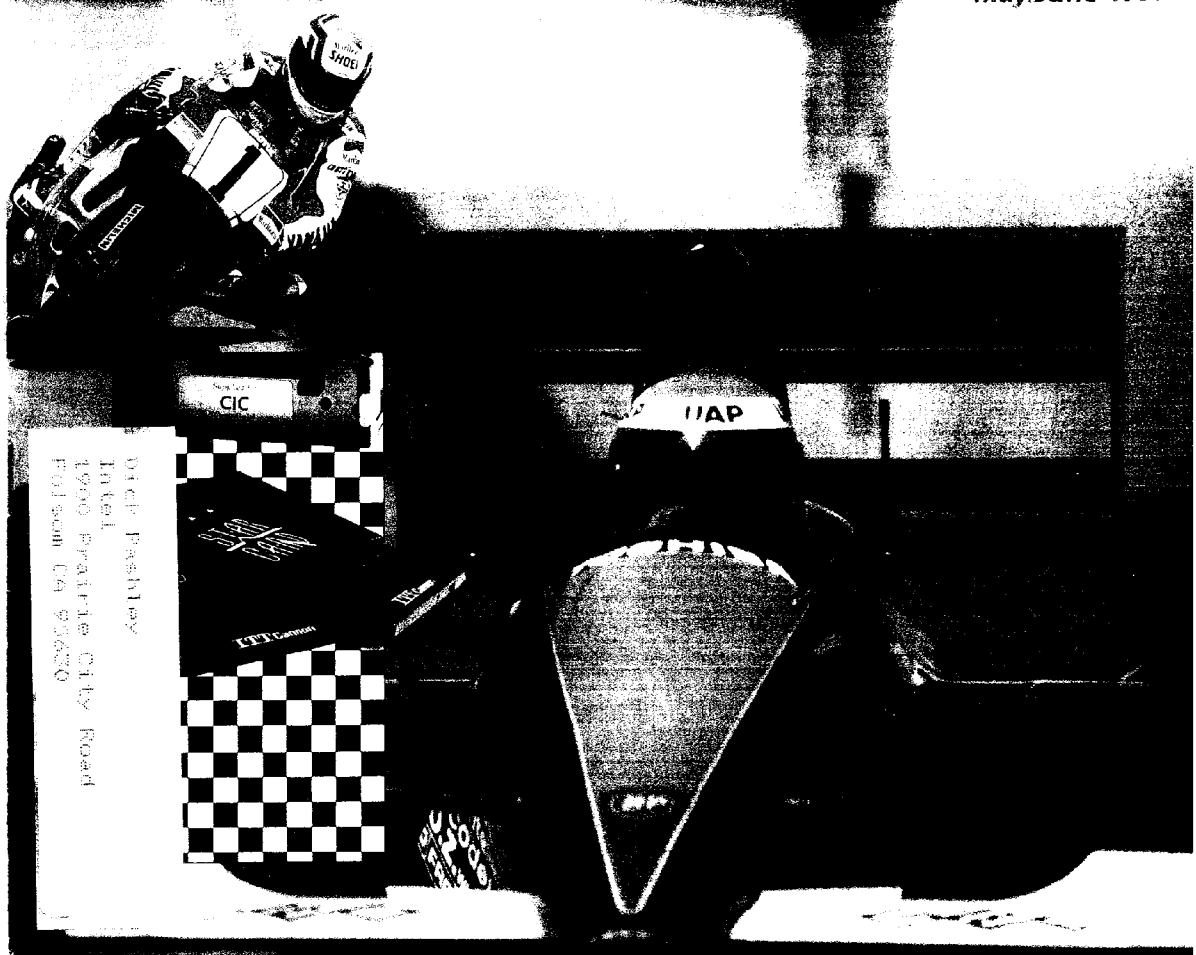
EXHIBIT H

MEMORY CARD

Systems & Design

May/June 1991



Successful User Profile: Cranfield Impact Center

CARD PHYSICAL

OVERVIEW OF THE PCMCIA "PC CARD STANDARD"

The Card Physical Task Force Group was established in November 1989 by the Technical Chairman under the direction of the Personal Computer Memory Card International Association (PCMCIA) Board of Directors and Executive Committee. The charter to the Card Physical Task Force Group as defined by the PCMCIA Technical Chairman, Board of Directors, and Executive Committee is to define all mechanical parameters required for the PC Card. These mechanical parameters include the package outline for the PC Card and electrical interface. In conjunction with the package outline definition, the electrical interface (connector) and PC Card reliability and environmental parameters are defined. In December 1989 PCMCIA and Japan Electronic Industry Development Association (JEIDA) agreed to cooperate in publishing a unified PC Card Specification. The JEIDA IC Memory Card Guidelines, Version 4, and the PCMCIA PC Card Standard, Revision 1.0, specify the same Card Physical parameters.

In order to begin the Card Physical definition task, the PC Card pin assignments were required. The Electrical Task Force Group defined the minimum pin assignments required. The Card Physical Task Force Group then investigated the various interconnect systems proposed by the Card Physical Committee members. The edge card as well as pin and socket interconnect systems were considered. Both the edge card and pin and socket interconnect systems have both positive and negative aspects. The committee discussed in great length the pros and cons of both interconnect systems. Because of the desires of the Committee to specify the interconnect system which exhibits the best compromise for cost, reliability, availability, and compatibility between the various manufacturers, the pin and socket was chosen over the edge card interconnect system. The 68 position pin and socket interconnect system was the PCMCIA choice and also the JEIDA interconnect system. It was felt the 68 position interconnect system as proposed by JEIDA required addition refinement. In February and in May 1990, members of PCMCIA traveled to Tokyo for a joint meeting with JEIDA to discuss differences between the two proposed specifications. PCMCIA proposed three pin lengths instead of two. The three pin lengths allowed the card detect pins to be

About the Author

Stan Sharp is the Manager of New Technology Development at Foxconn International, located in Sunnyvale, California. He recieved both his BSME and MSME degrees at the University of Texas at Arlington. Stan has over 25 years experience in electronic packaging design and production. He holds patents in electronic packaging and electrically interconnecting.

Stan has been an active member of PCMCIA since the association was formed in 1989. He served one year (9/89 to 9/90) as an elected member of the Board of Directors. He was the Chairman of the Card Physical Task Force Group from its inception to September 1990 and is presently the Co-Chairman of the Card Physical Task Force Group. He visited Japan twice in 1990 to present the PCMCIA Card Physical recommended specification changes to JEIDA.

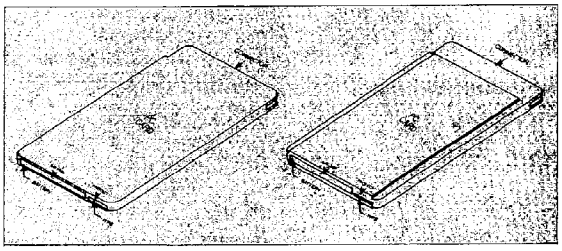


Figure 1.

	LENGTH	WIDTH	INTERCONNECT AREA	SUBSTRATE AREA
TYPE I	3.370 ±.008 (85.6 ±0.20)	2.126 ±.004 (54.0 ±0.10)	.065 ±.002 (1.65 ±0.06)	.965 ±.004 (1.65 ±0.1)
TYPE II	3.370 ±.008 (85.6 ±0.20)	2.126 ±.004 (54.0 ±0.10)	.065 ±.002 (1.65 ±0.06)	.098 MAX (2.5)

Table 1.

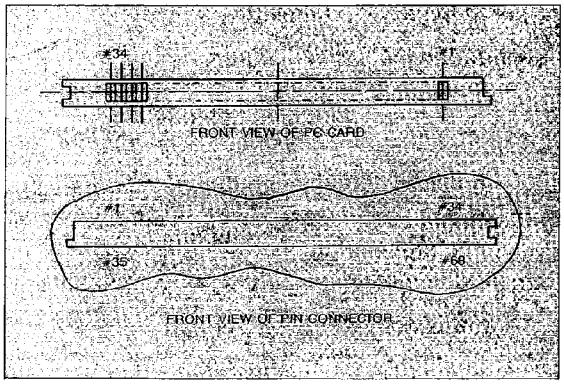


Figure 2.

"last mate and first break." The short card detect pins allow the PC Card to be fully mated before signals are sent to the card.

Other proposed changes to the JEIDA specification were the location and direction of the Write Protect Switch (WPS) protect mode, location of the battery and position of the positive side of the battery, minimum and recommended length of the pin connector card guide, and bottom cover label recommendations. These PCMCIA proposals were discussed in great detail with JEIDA at the February meeting. The PCMCIA proposed that specification changes be judged on their technical merit only. Both PCMCIA and JEIDA had stated that specification change arguments would not be considered without valid technical merit. The PCMCIA proposed specification changes were considered to be very important to the overall specification content.

The PC Card package outline considerations included the JEIDA proposal, IC memory packages currently available, intended PC Card use, and compatibility with the end user. It should be noted that two PC Card package outlines are specified: Type I and Type II.

The preferred package outline is the Type I. The Type II package is specified because of specific PC Card applications which require IC memory chips which will not fit within the Type I package. It should be noted that the Type II package outline specifies a substrate area and a connector area (Table 1).

The PC Card thickness dimensions listed in Table 1 are from the center line of the card to either surface. The thickness from the center line insures the PC Card connector is symmetrical. The connector area is defined so that one connector and card guide may be used for either type PC Card.

The 68 position PC Card polarization is specified in Figure 2.

It was noted that other PC Cards were being proposed with the same package outline but with different numbers of pins and intended uses. The polarization specified or proposed for each of the different PC Cards help guarantee that the various PC Cards will not mate or be electrically damaged if they are inserted into the wrong connector. It should be noted that the PCMCIA Specification states in paragraph 3.1.3 that a mismated PC Card and connector shall withstand a minimum of 22 pounds (10 Kg) static load without damage to the PC Card or connector.

The WPS and battery location were

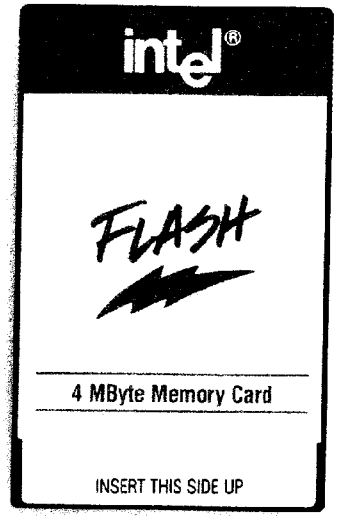
Make all these disappear.

Poof, they're gone. Bulky battery packs, slow hard disks and more. You see, where memory cards make sense, the new Intel 4MB Flash Memory Card provides a high-density, read/write, truly nonvolatile portable storage solution.

The real trick, of course, is that it allows you to store large amounts of code or accumulate data in a small, rugged card that uses less power than traditional memory technologies. So now you can design products that are faster, lighter, tougher and more energy efficient.

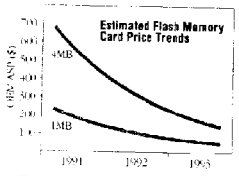
Not tomorrow, but today. Thanks to our System Developer's Kit. A DOS-compatible package that includes a 1MB Flash Memory Card, an add-in board as well as design information. In addition, it includes an evaluation copy of Microsoft's Flash File System software. All the stuff to get you started—now!

Plus, Intel's Flash Memory Card is the first to comply with the 68-pin PCMCIA/JEIDA standard. This allows the card and its data to be interchangeable between a host of



Actual Size.

With this simple card trick.



systems—from PCs to fax machines to bar-code readers.

The secret of the card's magic is Intel's ETOX™II Flash technology—combining RAM, ROM and mass storage capabilities onto a solid-state, energy-efficient medium. A technology so successful that volume continues to rise and prices continue to drop.

Want to know more about the secrets of this amazing technology? Then call (800) 548-4725 today and ask for Literature Packet #C8A03. And start performing your own disappearing tricks.

intel
The Computer Inside.™

INFO/CARD 9

specified for ease of use and to prevent confusion by the end user. The WPS and battery should be located on the end of the PC Card opposite the connector. The WPS will be used while the PC Card is inserted into the system. Therefore, the WPS is located to the right of the centerline and the "Protect" mode is toward the edge of the PC Card (Figure 3).

The battery location and position of the positive side of the battery are also specified (Figure 4).

The end user may desire to change the battery while the PC Card is powered by the host PC. Changing the battery while the PC Card is powered up will help insure no data is lost during the removal and replacement of the battery. The battery holder should be designed so that the positive side of the battery is toward the top cover of the PC Card. Since PC Card manufacturers use a coin type battery, it was felt that with the positive side up, the battery holder would be designed so that the battery will not drop out when the battery holder is removed from the PC Card.

The bottom label has been questioned by various members (Figure 5).

The top cover of the PC Card is used by the manufacturers for identification and other information. It was felt that the bottom cover will be used by the end user to record information. If the manufacturer does not place a bottom label on the PC Card, the end user may use a label which will interfere with the PC Card operation. Some of the problems discussed by the Committee with end user labels were: a) the label would not adhere properly to the cover, causing the label material to remain in the connector area, b) labels being applied in the connector guide area, and c) label residue which could cause electrical opens or shorts. The bottom label is specified so that potential problems may be alleviated when using the PC Card.

The connector specified is a 68 position two piece pin and socket. The socket was placed on the PC Card because of potential electro-static discharge, and because pins are more easily damaged by the end user. The present 68 position PC Card, as specified, is limited to 64 MB of memory. This memory capacity limitation has been discussed by the various Task Force Groups. The general consensus is that 64 MB configuration will saturate at some time in the future. Before the memory saturation occurs, PCMCIA will interface with JEIDA and other PC Card specification Committees in writing the next generation PC Card specification.

The Card Physical and Electrical Task Force Groups felt three (3) pin lengths were required (Table 2).

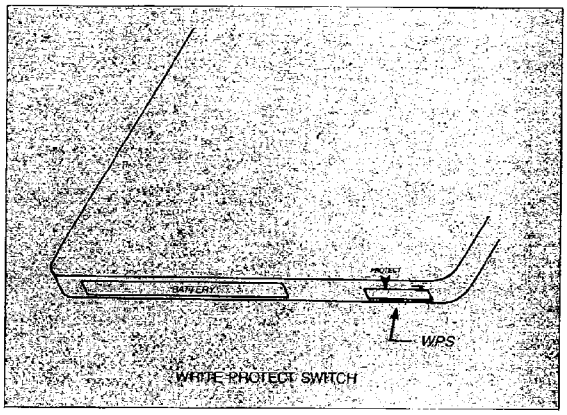


Figure 3.

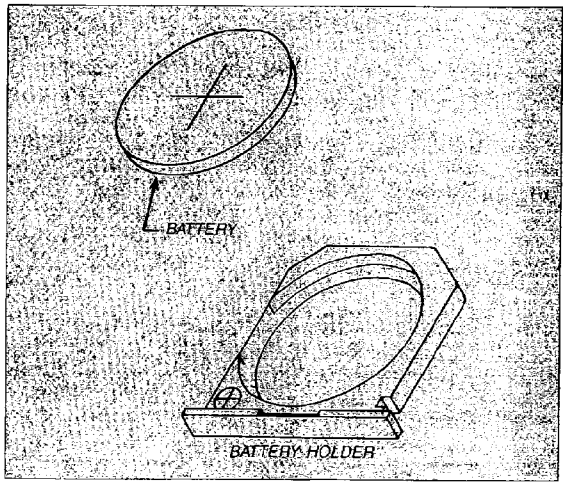


Figure 4.

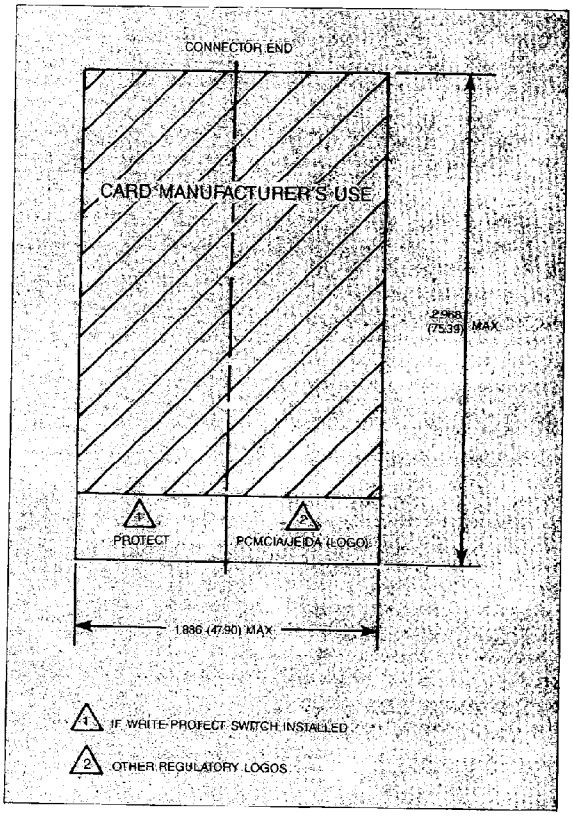


Figure 5.

PIN TYPE	PIN LENGTH (L)	PIN NUMBER
DETECT	.138±.004 (3.5 ±.1)	36,67
GENERAL	.167 ±.004 (4.25 ±.1)	ALL OTHER PINS
POWER	.197 ±.004 (5.0 ±.1)	1, 17, 34, 35, 51, 68

Table 2.

The longest pins (5 mm) are power and ground, the midlength pins (4.25 mm) are for signals, and the short pins (3.5 mm) are card detect. The ground pins are the two (2) outermost pins on each end of the connector and the power pins are in the center of the connector. With the ground pins being the outermost pins, they will make contact with the PC Card first if the PC Card is inserted at an angle. If the PC Card is inserted parallel to the connector then the power and ground pins will mate first. Because of electro-static discharge considerations, the first mated pins should be ground. The system ground will then discharge the PC Card static charge before the signal pins are mated. The card detect pins will be the last pins to mate when the PC Card is inserted and the first to break when the card is removed. The card detect pins allow the PC Card to be fully mated before the card is powered. One consideration for the short card detect pins was hot insertion and removal. The card detect pins do not completely alleviate the hot insertion and removal problem, if the card is inserted or withdrawn very quickly.

The connector pin and socket interface plating is defined as gold or gold compatible (paragraph 3.2.7). The plating also must withstand the Connector Durability specified in paragraph 3.5.0. It was felt that the various connector manufacturers would use the plating techniques they were most familiar with. Specifying that the plating must be gold or compatible with gold allows manufacturing flexibility while maintaining compatibility between the various manufacturer's pin and socket connectors.

The Printed Circuit Board (PCB) footprints are defined for the straight, right angle, and surface mount pin connectors. The footprints have been included so that the PCB designer may lay out the PCB and each manufacturer's pin connector will be compatible at the PCB. It should be noted that the footprints are shown without hole diameters or mounting considerations. Each PCB user adheres to a set of design rules for the particular applications, and each connector manufacturer and user may wish to mechanically fasten the connector to the PCB by various means. □