



Sierra Proto Express

Introducing our Micro Electronics Division



PCBs for MICROELECTRONICS

Design For Manufacturability

Presented by

Ken Bahl

President, Sierra Proto Express

at

PCB Design West Conference

March 2006

Santa Clara, CA

(these slides have been slightly modified for web download)



Presented at
PCB Design West Conference
Santa Clara, CA, March 2006





What are Microelectronics PCBs?

- ✓ **Microelectronics PCBs** generally refers to circuit boards with less than 2 Mil (0.002") Lines and Spaces.
- ✓ These are very tight, small boards which serve the rapidly growing microelectronics market, whose end products include applications like cell phones, PDAs, and personal game machines. Conventional PCBs with 4 mil or more trace/space are just not suitable for the increasing complexity and decreasing form factors of the devices of tomorrow.



We can produce Microelectronic PCBs with following Construction Types

- ✓ Single sided
- ✓ Double sided
- ✓ Multilayer sequential build up
- ✓ Blind via
- ✓ Buried via
- ✓ Sub-Composite Via



We use the following Laminate Material Types

- ✓ High Temp Lead-free (IS410 or equivalent)
- ✓ Polyimide
- ✓ BT
- ✓ Low Dk/Df (N4000-13 or equivalent)
- ✓ Standard FR4

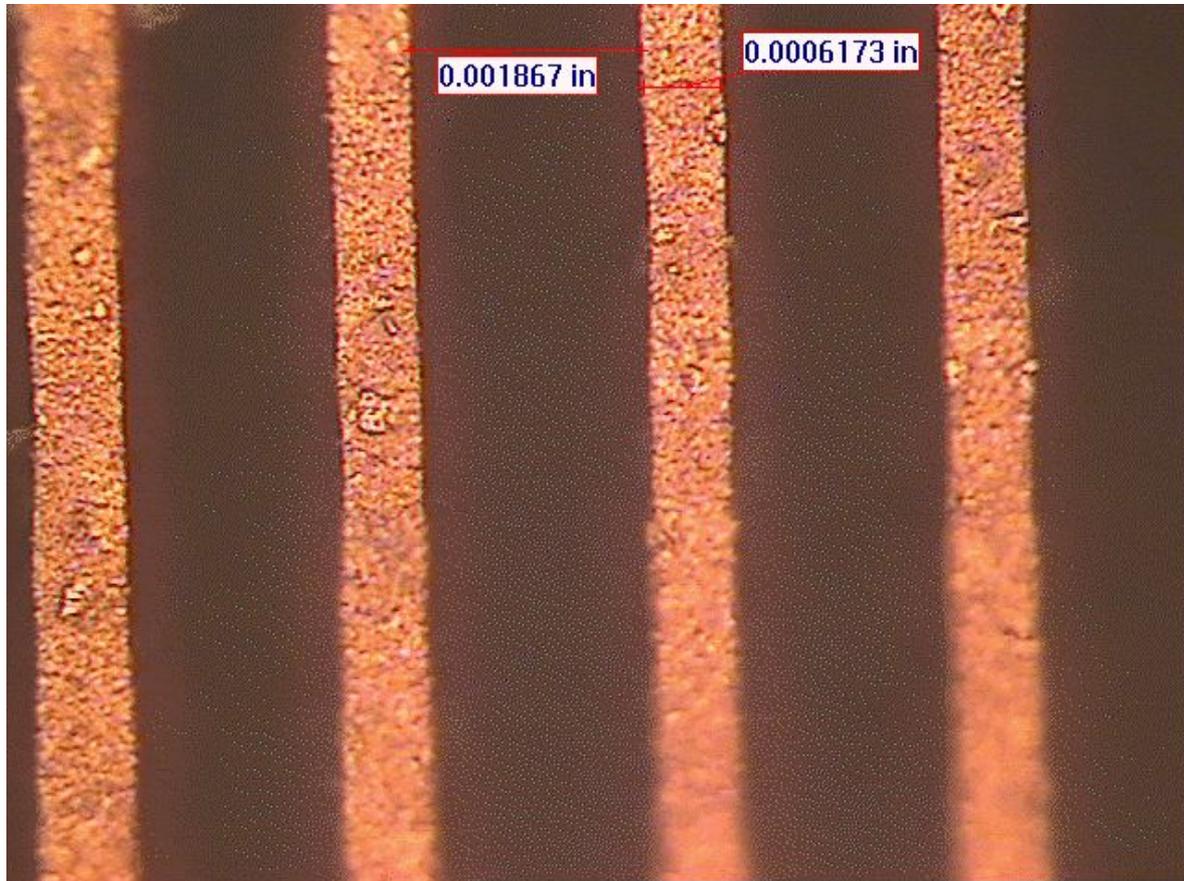


Design Guidelines: Minimum Line and Space

- ✓ Minimum line of 0.00125"
- ✓ Minimum space of 0.00125"
- ✓ If conflict, reduce line-width, but must maintain minimum space



Close up of fine traces and spaces
Actual board details shown. (1.8 Mil space, 0.6 Mil trace!!)
[just imagine what we can do for your hi tech applications]



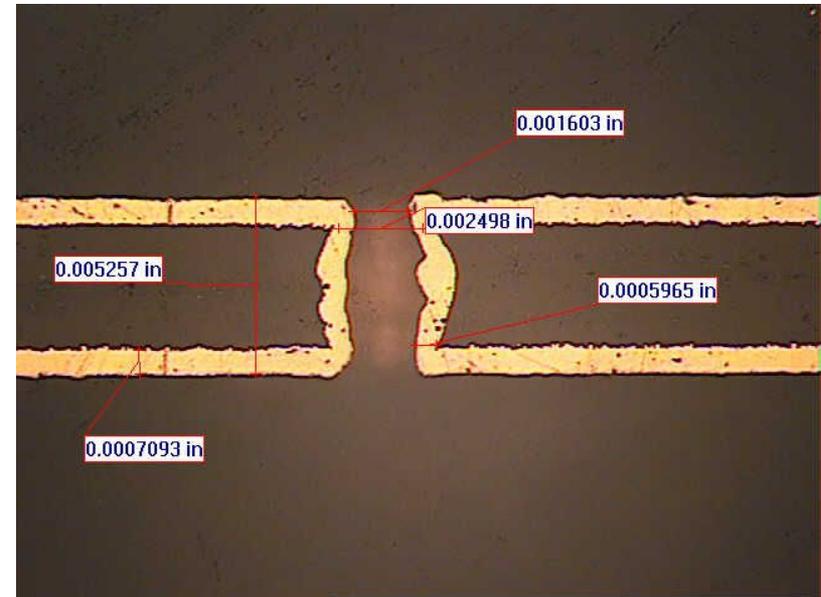
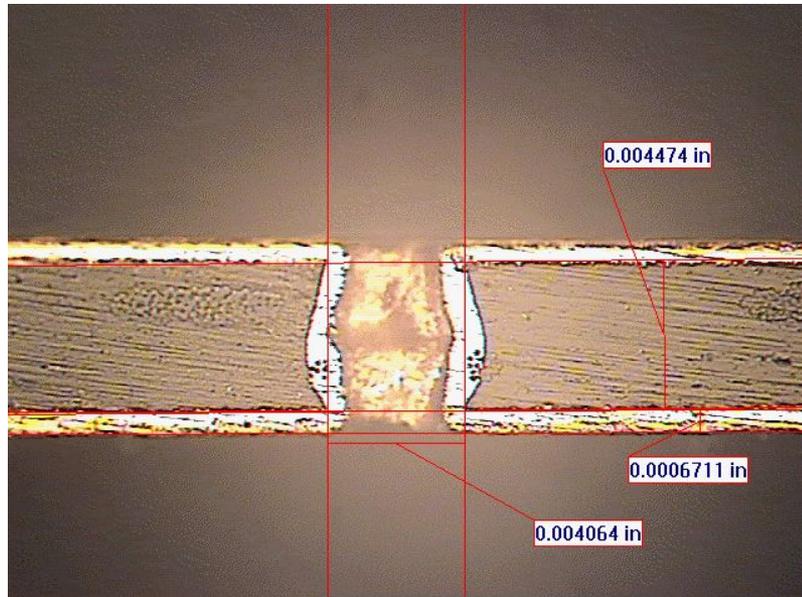


Design Guidelines: Hole Sizes

- ✓ Must not exceed plating aspect ratio
- ✓ Minimum Laser drilled finished through hole size is 0.002" inches
- ✓ Minimum Mechanical drilled through hole size is .004"
- ✓ Minimum finished blind via hole size is 0.002" inches



Cross section pictures of micro holes (showing actual holes with 2 mil and 4 mil diameter)





Design Guidelines: Plating Aspect Ratio

- ✓ Plated through holes can be built up to a 10:1 plating aspect ratio (Drilled hole Size/Panel thickness)
- ✓ Plated blind holes can be built up to a 0.8:1 plating aspect ratio (Drilled hole size/dielectric thickness)



Design Guidelines: Pad Size

- ✓ For a laser drilled hole, the pad diameter must be 0.006" inches larger than the finished hole size
- ✓ For a mechanical drilled hole, the pad diameter must be 0.008" inches larger than the finished hole size
- ✓ Teardrop pads



Design Guidelines: Copper Thickness

- ✓ Total copper thickness in the hole
 - Minimum of 0.0005" inches
 - Maximum of 0.0008" inches
- ✓ Total copper thickness on the surface
 - Minimum of 0.0005" inches
 - Maximum of 0.0008" inches



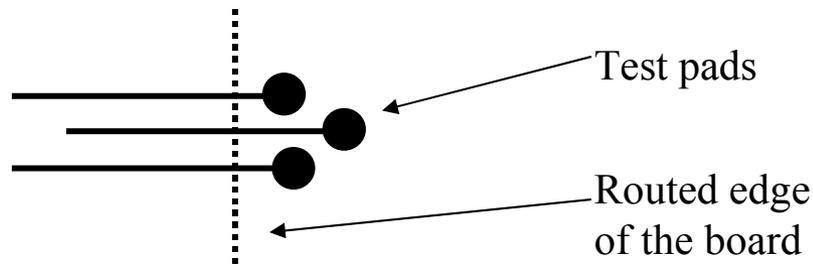
Design Guidelines: Surface Finishes

- ✓ Solder mask over bare copper (SMOBC)
- ✓ ENIG (Electroless Nickel Immersion Gold)
- ✓ Immersion Silver
- ✓ Entek OSP (Organic Surface Protection)



Design Guidelines: Electrical Test

- ✓ Test points must be 0.003" or greater
- ✓ Special Flying Probe test equipment required





Design Guidelines: Finished Board Thickness

- ✓ Must not violate plating aspect ratio
- ✓ Minimum core thickness of 0.0025"
- ✓ Minimum Dielectric is .002"
- ✓ Maximum finished board of 0.062"
- ✓ All materials are made in-house

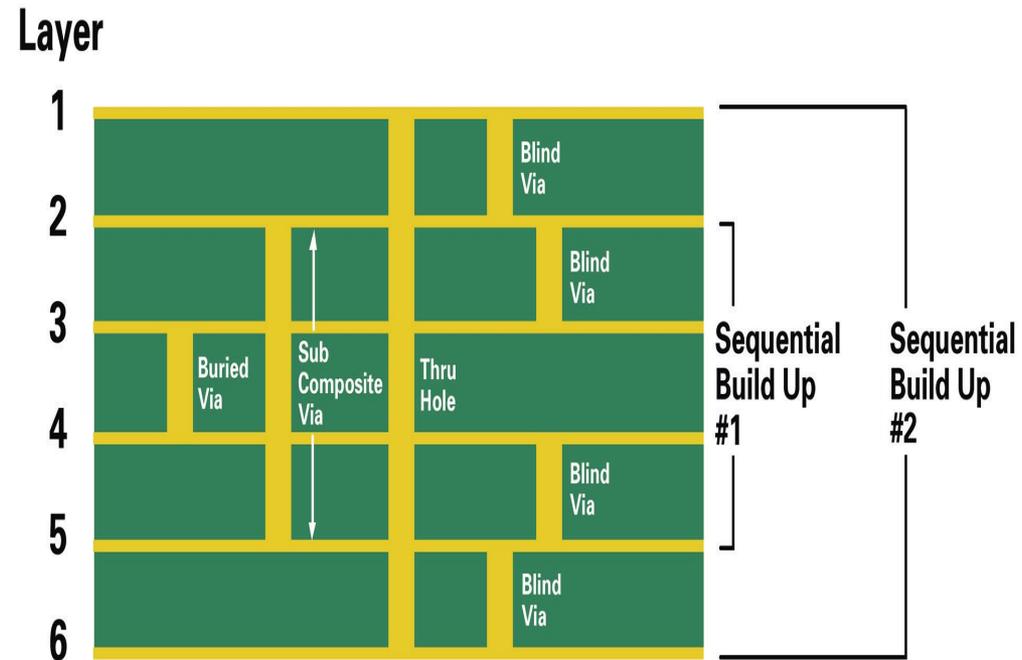


✓ OVERVIEW

- Min.Line & Space - .00125
- Min.Dielectric Thickness - .002"
- Min.Blind Via Finished Hole Size - .002"
- Min.Buried Via Finished Hole Size - .002"
- Min. Thru Hole Finished Hole Size - .002"
- Min. Distance Between Holes - .010"
- Max. Blind Hole Aspect Ratio - .8:1
- Max. Thru Hole Aspect Ratio - 10:1
- Max. Buried Via Aspect Ratio - 10:1



Sample of a complex PCB construction possible at Sierra Microelectronics





MAXIMUM HDI DENSITY (capabilities of Sierra Microelectronics)

- ✓ 18 Layers
- ✓ Nominal PCB thickness = .042"
- ✓ Nominal Copper thickness = .0006"
- ✓ Nominal Drilled hole = .004"
- ✓ Nominal Finished hole = .002"
- ✓ PCB can be built with any combination of Buried, Blind and thru hole designs



More Design Specifications

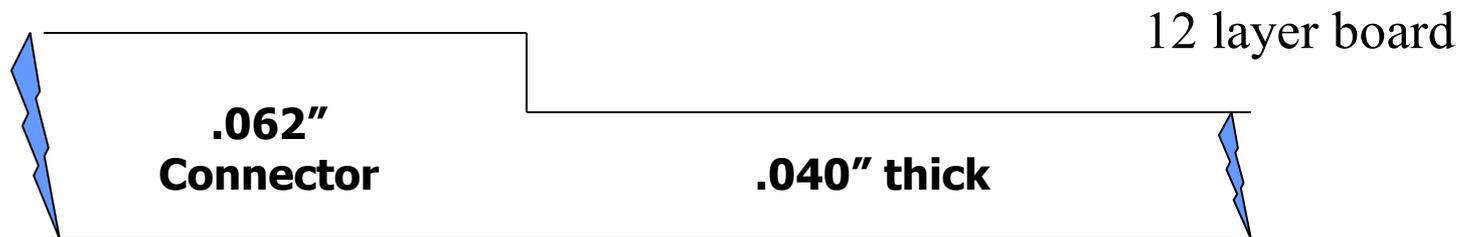
Pitch	Drilled Hole Size	Pad Size	# of Tracks
.5mm = 19.7 Mil	4 Mil.	8 Mil.	4
.4mm = 15.76 Mil	4 Mil.	8 Mil.	2
.3mm = 11.82 Mil	4 Mil.	8 Mil.	1
.25mm = 9.88 Mil	4 Mil.	8 Mil.	0



Another application: Burn In Boards

We recently made a 0.040" thick, 12 Layer PCB with .062" Connector

(We have great capabilities and we enjoy challenges. Tell us what we can do for you!)





Think of us as your PCB partner in the exciting new world of Micro Electronics

For more details visit:

www.protoexpress.com/microelectronics

Contact

Dale Baird, VP & General Manager

daleb@protoexpress.com

(408) 735-7137 X 9941