NOTES

Flex Hybrid v4.x
January 18, 2002

Naming convention:

4.x files are the "root" files of the design, i.e., all the vendor specific files are derived from these files and meet the specifications of v4.x drawings/documents. Completely enumerated files/drawings/documents contain vendor specific information. For example, v4.1 files are specific to Compunetics, but the only difference I anticipate is that the version number in the top metal of the flex circuit will read "v4.1".

Design files:

- apfhv4xc_eagle.zip — The native Eagle files.
- apfhv4xc_fab.pdf — The fabrication, assembly, construction, wire bonding, schematic, etc. drawings in Adobe Acrobat v3 format.
- apfhv4xc_gerber.zip — RS 274X Gerber files and Excellon drill files. Format information is included in the *.gpi files.
- apfhv4xc_specs.pdf — Vendor specific versions are merged at the front of the vendor's "fab" file,

1. The Gerber file set contains the following:

top.gbr	Top metal layer
bottom.gbr	Bottom metal layer
bottom_mask.gbr	Bottom solder mask, negative
top_mask.gbr	Top solder mask, negative
place.gbr	Circuit outline & component placement
drill.xcl	Excellon drill file
drill.gpi	Drill information file
rack.drl	Drill tool list
botnoglue.gbr	Glue keepout region for flex on frame
outline .gbr	As fabricated cut dimension of flex
paste.gbr	All solder pads, 1x size

   IMPORTANT NOTE: Vendor specific gerber file sets do not include all of the above files, only those needed for bare flex fabrication.

2. The Eagle board files are NOT the final files from which fabrication and assembly are done. Subtle but important changes are made in the Gerber files which are the final, definitive sources for fabrication, assembly and construction. The Eagle schematic files ARE the definitive schematic definition. The Eagle files are provided for easier tracing of nets and checking of features.

Missing file, 20 December, 2001
Bill of materials/coordinate file