

Re: CAD tools compatibility ?

Source: <http://sci.tech--archive.net/Archive/sci.electronics.cad/2006-02/msg00154.html>

- *From:* "Brad Velander" <bveland@xxxxxxxxxxxxx>
 - *Date:* Sun, 26 Feb 2006 03:38:51 GMT
-

Leon,

Okay I stand corrected with regards to Pulsonix, I was speaking from my past experiences with numerous packages and commenting on your list of file extensions.

--

Sincerely,
Brad Velander.

"Leon" <leon_heller@xxxxxxxxxxxxx> wrote in message
news:1140849558.597778.90990@xx

Brad Velander wrote:

Yes, Leon is just copying something.
IEON, you had better indicate what files/formats are what or you are seriously misleading these guys. I don't see one library format that I recognize despite Leon's comments.

There are several formats in there that are simply netlists (i.e. PADs *.net & Accel *.net), not the schematics nor PCB files. Next you have the problem that most nobody saves their files in some of those formats (i.e. ascii, Protel *.ASC), so down the road you may not be able to readily deal with them again unless you have some friend with the tools to correctly read them and write the acceptable format/version files back out to an ascii file.

Besides the basic information Leon has offered, there is surely limitations on the actual file versions that it can import. In some cases they may be well out of date versions. Don't be sure it will import any current specific format unless the software supplier will guarantee you. Then get ready to deal with the limitations of the compatibility upon importing. This is true of even the best imports because most packages have features or details which are not supported nor correctly converted on

Re: CAD tools compatibility ?

import.

From what I have seen in over 20 years is that almost no package imports or converts schematic files, it is just not worth the development time. A schematic can be redrawn in a couple of days once you know your new tool. PCBs do have converters because that can save you weeks if not months of time to redevelop the PCB. Then you have to confirm that it is valid and the same as the original, all big costs.

Those were schematic ASCII formats. It will also import