

Re: How we can convert PCB file to schematic file on Protel DXP

Source: <http://sci.tech-archive.net/Archive/sci.electronics.cad/2007-04/msg00021.html>

- *From:* Peter Bennett <peterbb@xxxxxxxxxxxxxxxxxxxxx>
 - *Date:* Wed, 11 Apr 2007 09:05:27 -0700
-

On 11 Apr 2007 02:14:36 -0700, "waatage" <waatage@xxxxxxxxx> wrote:

Brad,
Thanks for your reply,
BUT
Actually by mistake ,i delete my design which i made on schematic, but i have its PCB and PCB's layout. Now i dnt remeber the acutal values of resistors and capacitors and their interconnection with other components,
Thats why i asked for the method to get back my schematic from my layout.

I knew that doing reverse engineering ,i can get that interconnections but still i wouldnot be able to have the values of resistors and capacitors

.
So if u know any advance method for this please tell me,

With regards.

Usman

I've used Protel 99SE, not DXP. In 99SE, I think there may be a way to produce a bill of materials that would include the component value (but I've never used it...). If not, you can examine each component on the PCB and look at its "comment".

Did you not print the schematic before designing the board? You should also be able to find backup schematic files somewhere.

--

Peter Bennett, VE7CEI
[peterbb4 \(at\) interchange.ubc.ca](mailto:peterbb4@interchange.ubc.ca)
new newsgroup users info : <http://vancouver-webpages.com/nnq>

Re: How we can convert PCB file to schematic file on Protel DXP

GPS and NMEA info: <http://vancouver-webpages.com/peter>

Vancouver Power Squadron: <http://vancouver.powersquadron.ca>