

# 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/msg00031.html>

---

- *From:* "Brad Velander" <[bveland@xxxxxxxxxxxxx](mailto:bveland@xxxxxxxxxxxxx)>
  - *Date:* Thu, 12 Apr 2007 06:51:09 GMT
- 

Okay Usman,

Your task is not quite so daunting then. Yes as Peter mentioned, you can probably get your values from the comment field of the parts within the PCB. That is the usual location for the values to end up in if you used the Part type field in the original schematic. You could also use the output tools to generate a BOM from the PCB and include any information that was passed through to the PCB parts through parameters or any other means.

Then of course you will also need all of the original symbols, should be simple as long as you had a library that wasn't deleted. Then the interconnects more or less manually, then check the schematic against the PCB until you get no changes being performed as an ECO between the schematic and the PCB.

Your earlier comments sounded like the rookie or someone expecting a single push button solution, hope you didn't take my comments too harshly.

—  
Sincerely,  
Brad Velander.

"Peter Bennett" <[peterbb@xxxxxxxxxxxxxxxxxxxxx](mailto:peterbb@xxxxxxxxxxxxxxxxxxxxx)> wrote in message [news:qi1q139kno33kpd92ed7531vu55eal5rd@xxxxxxxxxxxxxxxxxxxxxxxxx](mailto:news:qi1q139kno33kpd92ed7531vu55eal5rd@xxxxxxxxxxxxxxxxxxxxxxxxx)

On 11 Apr 2007 02:14:36 -0700, "waatage" <[waatage@xxxxxxxxx](mailto: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 interconection 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

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

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

new newsgroup users info : <http://vancouver-webpages.com/nnq>

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

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