

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/msg00016.html>

- *From:* "Brad Velander" <bveland@xxxxxxxxxxxxx>
 - *Date:* Wed, 11 Apr 2007 06:31:54 GMT
-

Usman,

There is no automatic manner to get there. You would have to manually compile the schematic from reverse engineering the PCB design yourself, then check the schematic for accuracy/correctness against the PCB until you got it right.

I would have to believe you are looking for a shortcut but you don't understand the process even by which the PCB was originally designed. There are no shortcuts, only managers and engineers that don't understand the process think that CAD design is simply punching the right buttons.

--

Sincerely,
Brad Velander.

"waatage" <waatage@xxxxxxxx> wrote in message
news:1176271966.006657.292740@xx

Hello Friends,

Does any one know how to convert PCB file to schematic file on the DXP Protel 2004 or other version? please tell me the steps to follow, if it is possible.

With
regard

Usman