

Very Basic Question on Protel DXP 7.2

Source: <http://sci.tech-archive.net/Archive/sci.electronics.cad/2007-11/msg00050.html>

- *From:* "Mistaken4" <u39354@uwe>
 - *Date:* Sun, 25 Nov 2007 19:27:03 GMT
-

I have been using ORCAD, now forced to switch to an older version of Altium.

(Design Explorer 7.2.92) circa 2002.

With ORCAD a Netlist is generated in Schematic Capture which is then used by Layout to generate the PCB connections.

I have been unable to figure out how to do the equivalent in DXP. Several Netlist options are available in DXP schematic build, but I've been unable to see how the PCB layout portion loads these. (Or which Netlist option is the proper one to use for that matter.)

Using this older rev of DXP, how do I properly transfer a schematic into a PCB layout?

Thanks much in advance.
Terry

.