

Re: Altium Protel?

Source: <http://coding.derkeiler.com/Archive/General/comp.arch.embedded/2005-09/msg00097.html>

- *From:* google.4.oracle@xxxxxxxxx
 - *Date:* 1 Sep 2005 07:41:27 -0700
-

Eric wrote:

> Hello All,
>
> My company currently uses Altium's PCAD 2004 and are considering
> switching to Altium Protel 2004.
>
> For the people using Protel, how do you like it?
>

We use the Altium DXP (which is the latest version of the Protel software) schematic package at work. It is the worst schematic package I have ever used. It is buggy and has a lousy user interface. I have used many schematic packages over that past years and I am constantly amazed at how Altium can make the simplest things so complicated.

- Simple features like dragging sections of schematic never works properly. It insists on adding connections between wires that weren't there before.
- Clicking on Undo after a drag and drop causes the program to crash
- The Help files are useless. It has never had any help information on anything I have tried to look up.
- There is no easy way to specify that you want to have seperate ground planes that connect a single point.
- The print function is hopelessly complicated (There are 3 different Print Menu fuctions and settings from one conflict with settings from others).

The list of problems goes on and on....

I think that DXP is an overpriced, underperforming piece of software. I think that the programmers who wrote it should be sentenced to a year of doing 'real' designs with it for an electrical engineer and then see how they like it.

Other than that, I think it sucks.

- *Follow-Ups:*
 - ◆ **Re: Altium Protel?**
 - ◇ *From:* Leon
 - ◆ **Re: Altium Protel?**
 - ◇ *From:* Bob Stephens

- Prev by Date: **Re: Atmega Internal clock source accuracy**
- Next by Date: **Re: general questions regarding ARMs**
- Previous by thread: **Re: Atmega Internal clock source accuracy**
- Next by thread: **Re: Altium Protel?**
- Index(es):
 - ◆ **Date**
 - ◆ **Thread**