

# EEVblog Electronics Community Forum



A Free & Open Forum For Electronics Enthusiasts & Professionals

Welcome, **Guest**. Please login or register.  
Did you miss your activation email?

This topic ▼

Search



Forever ▼

Login

Login with username, password and session length



Home



Help



Search

About us

Links




Login




Register

EEVblog Electronics Community Forum » EDA » Eagle » Update PCB from edited Schematic




**Check out our Used Equipment Offers**

**SHOP NOW >>**



**5 PCBs for \$2, any color**

**\$5 off for the first 50 to order with the code "EEJLPCB"**



« previous next »

Pages: [1] Go Down

PRINT

SEARCH



Author

Topic: Update PCB from edited Schematic (Read 27564)

0 Members and 2 Guests are viewing this topic.



**andersendr**

Regular Contributor



Posts: 56

Country: 



**Update PCB from edited Schematic**

« on: February 13, 2012, 01:50:11 am »

Hey Forum,

So I have designed a schematic and board, now I have made changes to the board, added a few more components. Now when I switch to the PCB it says that no forward/backward annotation exists and the new parts are not on the PCB. Is there some way to get the PCB to reflect the schematic changes?



Logged



**bfritz**

Regular Contributor



Posts: 133

Country: 



**Re: Update PCB from edited Schematic**

« Reply #1 on: February 13, 2012, 02:01:17 am »

This is an issue with Eagle.


If you wish your schematic and layout to stay "in sync", it is necessary to make sure Eagle has both the schematic and layout associated with one another and in memory during the changes.

The way to fix your problem, is to use "undo" to get back to where your pcb and schematic are synchronised and correctly reflect one another. Then make your schematic changes, making sure that both the layout and

schematic have windows open at the same time.

This is one of the big problems with Eagle. Other packages have the ability to not have both in memory at the same time, and create forward and back annotation files that make synchronization possible. Eagle just does not do this, and instead tells you to keep both windows open at the same time.

Sorry... but this is a bit "Gotcha" with Eagle.

 Logged

 **andersendr**

Regular Contributor



Posts: 56


Country: 



### Re: Update PCB from edited Schematic

« **Reply #2 on:** February 13, 2012, 03:22:31 am »

OK that is sort of what I thought due to lack of information found through the search engines. These really becomes a pain when multiple people start editing stuff. I wonder if they have fixed this in V6? Since it is now all XML, it would be a pretty easy fix.

 Logged

 **westfw**

Super Contributor



Posts: 3080

Country: 




### Re: Update PCB from edited Schematic

« **Reply #3 on:** February 13, 2012, 09:51:14 am »

I don't believe that CADSoft considers this a "problem", so no, it's not "fixed" in v6.

I don't see how it's any more or less of a problem when multiple people edit stuff. How would having multiple versions of a .SCH, each with "improvements" and one old .brd file be better?

 Logged

 **bfritz**

Regular Contributor



Posts: 133

Country: 



### Re: Update PCB from edited Schematic

« **Reply #4 on:** February 13, 2012, 04:47:39 pm »

**Quote from: westfw on February 13, 2012, 09:51:14 am**

I don't believe that CADSoft considers this a "problem", so no, it's not "fixed" in v6.

I don't see how it's any more or less of a problem when multiple people edit stuff. How would having multiple versions of a .SCH, each with "improvements" and one old .brd file be better?

The issue is when you have multiple people working on a schematic, and want them all to be able to edit.

With Eagle, they must sequentially edit the schematic, or one person must be responsible for making all schematic edits.

Most professional schematic capture and layout packages support forward and back annotation, OrCad offers tools that fulfill this need. Multiple people can edit a released schematic, and then the person in charge can choose to accept edits from the different people. This is less of an issue for hobbyists, but is a real problem in a commercial engineering environment,

where you may have multiple engineers working on the same project, each responsible for sections of the design.

 Logged

☐ **andersendr**

Regular Contributor




Posts: 56

Country: 

### Re: Update PCB from edited Schematic

« **Reply #5 on:** February 14, 2012, 01:57:51 am »

Well what I meant was that I may open the schematic and board, but another person editing it may not remember to open both. The more people the greater the chance that one of them will forget to open the schematic and pcb together.

 Logged

☐ **westfw**

Super Contributor



Posts: 3080

Country: 


### Re: Update PCB from edited Schematic

« **Reply #6 on:** February 14, 2012, 08:07:59 am »

That's like saying "the more people who are editing a Word document, the more likely someone will forget to save their changes before exiting."

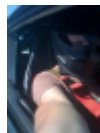
True, but essentially silly. You don't "forget to open both"; it's part of the standard practice when using EAGLE. And every time you open one, it will specifically ask whether you want to open the other was well...

If you're going to let a bunch of people edit your schematics without learning how to use the software, this is probably the least of your worries :-)


 Logged

☐ **charliex**

Frequent Contributor



Posts: 324

Country: 


Car Hacker



### Re: Update PCB from edited Schematic

« **Reply #7 on:** February 15, 2012, 04:22:07 am »

you can turn off the load requester by setting Option.AutoLoadMatchingDrawingFile then it should always load both.

 Logged
Pages: [1] [Go Up](#)

PRINT

SEARCH

« previous next »

Share me





[EEVblog Main Site](#)

[EEVblog on Youtube](#)

[EEVblog on Twitter](#)

[EEVblog on Facebook](#)

SMF 2.0.15 | SMF © 2017, Simple Machines

Simple Audio Video Embedder

SMFAds for Free Forums

[XHTML](#) [RSS](#) [Mobile](#) [WAP2](#)