

Re: What is the Best PCB Layout software ? (Money no object)

Source: <http://coding.derkeiler.com/Archive/General/comp.arch.embedded/2004-01/0917.html>

From: Ralph Malph (*noone_at_yahoo.com*)

Date: 01/13/04

Date: Mon, 12 Jan 2004 23:48:21 -0500

Ian McBride wrote:

>
> *"Ralph Malph" <noone@yahoo.com> wrote in message*
> *news:40036859.F8EA05E7@yahoo.com...*
>
>> *One thing it can't do (without screwing up the DRC) is holes in a PAD.*
>> *I am using a small regulator that requires heat spreaders on the top and*
>> *bottom of the board connected by vias directly under the thermal pad on*
>> *the bottom of the package. I know this is not normal, but TI recommends*
>> *it. I was never able to get rid of the DRC errors this produced.*
>
> *If this is the package I remember, you can make the thermal pad a small pad*
> *with THERMAL=OFF inside a rectangle on the top layer. A pain, but no DRC*
> *complaints.*

I am not trying to make a thermal. I am trying to make a fairly large rectangular pad with six holes (vias) in it. The entire rectangle needs to have the solder mask removed from it. I guess I could have split the pad up into six equal, rectangular areas as pads. But they should be touching and I don't think I can get this past the DRC either unless I allow everything to touch. I believe I have object spacing set for 10 mil at the moment.

Or could I use a polygon to open up an area in the solder mask? I did not try much in that area.