Home   Mail   News   Sports   Finance   Weather   Games   Groups   Answers   Flickr | More

Search Groups   Search Web   Sign In   Mail

kicad-users

Home
Messages
   Attachments
Links

Members Only
Post
Files
Photos
Database
Polls
Members
Calendar
Promote

The Yahoo! Groups Product Blog

Check it out!

Info | Settings

Group Information

Members: 3499
Category: Open Source
Founded: Aug 31, 2005
Language: English

Already a member?
Sign in to Yahoo!

Yahoo! Groups Tips

Did you know...
Hear how Yahoo! Groups has changed the lives of others. Take me there.

Messages   Message # [   ] Go Search: [   ] Search Advanced Messages Help

Topic List   < Prev Topic | Next Topic >

Re: Orcad V9.0 replacement with Kicad   < Prev | Next > | Reply

Posted By: Tue Apr 23, 2013 1:38 pm | Options

Hello Don-
Can you provide us with your max2brd.exe? I'm not capable of duplicating your
effort without learning more than I have time for right now.
Regards
Phil
pguiet@...

--- In kicad-users@yahoogroups.com, "Michele" <m.costant@...> wrote:
>
> Brilliant. Thank you very much Don.
> Regards,
> Michele
>
> --- In kicad-users@yahoogroups.com, "DH" <donh2@> wrote:
> >
> > Michele,
> >
> > I am able to compile and run max2brd using Cygwin. It works under Win 7 for
> > me also.
> >
> > Don
> >
> > -----Original Message-----
> > From: kicad-users@yahoogroups.com [mailto:kicad-users@yahoogroups.com] On
> > Behalf Of Michele
> > Sent: Monday, April 22, 2013 1:28 PM
> > To: kicad-users@yahoogroups.com
> > Subject: [kicad-users] Re: Orcad V9.0 replacement with Kicad
> >
> > Hi Phil,
> > yes, you are right :-) But there is of course to talk about the Orcad
> > Capture to Kicad EEschema conversion yet.
> >
> > Regarding max2brd, I use the executable (the Debian installer)on linux Mint
> > 32 running it on a windows virtual machine.
> >
> > So I would suggest doing the same (download the oracle virtual machine,
> > Linux Mint 32, install it and then install max2brd from the debian
> > binaries).
> >
> > If anybody has better solution, simplifying the matter, please let us know,
> > that would be very much appreciated.
> >
> > Regards,
> > Michele
> >
> >
> > --- In kicad-users@yahoogroups.com, "pg2845" <pg2845@> wrote:
> > >
> > > Hi Michele-
> > > You've had a busy time since I started this! I have downloaded the
> > application max2brd (for example from
> > http://fpgalibre.sourceforge.net/eda.html), and must to admit to being
> > stumped about what to do next. I assume that what I got was source code for
> > max2brd.??? I don't think I have the resources to generate a Windows (V7 or
> > XP) binary. Any suggestions?
> > > Best Regards
> > > Phil
> > >
> > > --- In kicad-users@yahoogroups.com, "Michele" <m.costant@> wrote:
> > > >
> > > >
> > > >
> > > > Hello John,
> > > > thanks. It would be really good to have scripts to automate the
> > additional editing required.
> > > >
> > > > For example one additional thing that I do is running the application
> > kcentr which I created myself and can be downloaded from the files area.
> > This makes the footprints on the brd file to have their anchor point in the
> > centroid of the pads making easier to create neath footprint libraries and
> > re-use the same footprint on other translated boards.
> > > >
> > > > In multilayer boards I also prefer to recreate planes (gnd, power) as

> > > > In any case I suggest everybody going through this route to compare
> > gerber files produced by Orcad Layout and gerber files produced by kicad
> > Pcbnew making sure they overlap nicely. I use Pentalogic viewmate to do this
> > check.
> > > >
> > > > Regards,
> > > > Michele
> > > >
> > > > --- In kicad-users@yahoogroups.com, John Hudak <jjhudak@> wrote:
> > > > >
> > > > > That explaintion is helpful. Thank you. Having used KiCAD for a
> > > > > while, and only rarely used ORCAD (well a lot long time ago), I
> > > > > have an observation about your step #4. It seem that the majority of
> > the 'fix ups'
> > > > > could be done with a script. Is that true?
> > > > > Has anyone done any of the substeps with a script?
> > > > >
> > > > > Best regards,
> > > > > John
> > > > >
> > > > >
> > > > >
> > > > > On Fri, Apr 19, 2013 at 5:39 PM, Michele <m.costant@> wrote:
> > > > >
> > > > > > **
> > > > > >
> > > > > >
> > > > > > Hello,
> > > > > > To translate your Orcad Layout max file into a Kicad pcbnew brd
> > > > > > file, please follow the following flow (here just sketched, for
> > > > > > details please
> > > > > > ask):
> > > > > >
> > > > > > 1) first thing to do is cleaning up your max file using the
> > > > > > clean up tool provided into Orcad Layout in order to remove
> > > > > > unused footprints, nets and padstacks (Auto->Cleanup Design.
> > > > > > Tick "Remove Unused Padstacks", "Remove Unused Footprints",
> > > > > > "Remove Unused Nets");
> > > > > >
> > > > > > 2) In Orcad Layout, go on Options->Stackup Settings, tick all
> > > > > > the boxes (Solder Paste, Solder Mask, Silkscreen etc...) and
> > > > > > click OK;
> > > > > >
> > > > > > 3) download the application max2brd (for example from
> > > > > > http://fpgalibre.sourceforge.net/eda.html) and use it to
> > > > > > translate the max file into a brd file. Edit the output brd file
> > > > > > with a text editor (I use Notepad++) and add at the end of it
> > > > > > the following line: $EndBOARD
> > > > > >
> > > > > > 4)At this point you should already have obtained a kicad brd
> > > > > > file that pcbnew can open. The file normally have wrong
> > > > > > designators (at least this happens to me) so they need editing.
> > > > > > Normally polygons/copper areas need to be filled. Note that
> > > > > > kicad does not support arcs and circles in polygons on copper
> > > > > > layers so, if the original max file had those, they have to be
> > > > > > re-created (I use an arc/circle on a non-copper layer and then I
> > approximate it on the copper layer using small track segments).
> > > > > > Furthermore, I replace the Free vias used in Orcad with a
> > > > > > special component created in kicad.
> > > > > > Another useful thing is setting the text dimension; I do :
> > > > > > Edit->Reset Module Fields Size and use the following numbers:
> > > > > > Size X=0.9mm, Size Y= 0.8mm, Thickness= 0.2 mm
> > > > > >
> > > > > > The above is a rough introduction on the process but I think it
> > > > > > is good enough to try having a start. There are some other thing
> > > > > > that I do but at this stage I would rather avoid to create
> > > > > > (more) confusion :-)
> > > > > >
> > > > > > I would like to thank Salvador Tropea for the amazing max2brd
> > application.
> > > > > >
> > > > > > I hope the above helps.
> > > > > > Regards,
> > > > > > Michele
> > > > > >
> > > > > >
> > > > > > --- In kicad-users@yahoogroups.com, "Michele" <m.costant@> wrote:
> > > > > > >
> > > > > > > Hello Phil,
> > > > > > > Thanks for your file, I now understand better your needs
> > > > > > > regarding your
> > > > > > > second question.
> > > > > > >
> > > > > > > My answer about it (maybe someone else can give a better one)
> > > > > > > is that

> > > > > > >
> > > > > > > However, I can see that you can produce your document using a
> > > > > > > two steps
> > > > > > approach:
> > > > > > > 1) from pcbnew (kicad pcb layout) you can go:
> > > > > > > File->Print->untick all
> > > > > > copper layers, in the technical layer section select the front
> > > > > > silkscreen layer,in the Page Print section select single page,
> > > > > > and finally you can print using a pdf printer software;
> > > > > > >
> > > > > > > 2) In EEschema (kicad schematic capture) you can add how many
> > > > > > > fields you
> > > > > > want to describe your components (for example, apart from the
> > > > > > standard "value" and "footprint" I use "Manufacturer",
> > > > > > "Manufacturer PN","Supplier 1", "Supplier 1 PN","Supplier 2",
> > > > > > "Supplier 2 PN", a field if the component is critical and cannot
> > > > > > be replaced with equivalent,our internal PN and many others
> > > > > > depending on the articular component). The above fields
> > > > > > information can then be exported in a file which could be the
> > > > > > netlist (we hava a couple of options) or a bom file (again, we have
> > more than one option). The field information file can be printed out with
> > the pdf printer.
> > > > > > > In Kicad, the master document containing the component
> > > > > > > information is
> > > > > > effectively the schematic capture file.
> > > > > > >
> > > > > > > I will write something about the migration Orcad
> > > > > > > Capture/Layout to Kicad
> > > > > > in other posts.
> > > > > > >
> > > > > > > Regards,
> > > > > > > Michele
> > > > > > >
> > > > > > >
> > > > > > >
> > > > > > > --- In kicad-users@yahoogroups.com, "phil.guiet" <phil.guiet@>
> > wrote:
> > > > > > >
> > > > > > > Good morning Michele-
> > > > > > > > Thanks for your reply. My initial post was general to
> > > > > > > > attract someone
> > > > > > with the same target as mine. I'm glad to hear that you feel
> > > > > > this change of tools is worth the effort.
> > > > > > >
> > > > > > > > I see how I can use File->Plot->Generate Drill File, choose
> > > > > > > > your
> > > > > > format in the Drill Map File Format and hit Map File, to get to
> > > > > > where I need to be with regard to a formal PCB fabrication drawing.
> > > > > > >
> > > > > > > > I have uploaded 12017R4.PDF to the forum Files area. All the
> > > > > > > > data
> > > > > > shown here originates from the Capture schematic and is passed
> > > > > > to the Layout 12017R4.max via the netlist. The schematic parts
> > > > > > have user defined fields for corporate part number and part
> > > > > > description. Each Layout footprint has associated fields to
> > > > > > allow me to display their content on the assembly layer of the
> > > > > > .max file for printing. The title block, revision block, part
> > > > > > list format, and the B size form are elements of a footprint which
> > is added to .max file.
> > > > > > >
> > > > > > > > I like this whole approach since the schematic is the
> > > > > > > > controlling
> > > > > > document and forces a close one to one correspondence.
> > > > > > >
> > > > > > > > I hope this clarifies my target, and I look forward to more
> > > > > > conversation with you. I'm going to jump in here with both feet.
> > > > > > >
> > > > > > > > Best Regards,
> > > > > > > > Phil
> > > > > > >
> > > > > > > > --- In kicad-users@yahoogroups.com, "Michele" <m.costant@>
> > wrote:
> > > > > > > >
> > > > > > > > Hello,
> > > > > > > > I am in the same process of moving projects from Orcad
> > > > > > Capture/Layout to Kicad. I think I am now confident with this
> > > > > > process so I am happy to give my help.
> > > > > > > >
> > > > > > > > As I am not entirely sure about your questions:
> > > > > > > > 1) if you need a file with the drill chart (similar to the
> > > > > > > > Layout
> > > > > > drill chart) this is not a problem in Kicad.
> > > > > > (File->Plot->Generate Drill File, choose your format in the
> > > > > > Drill Map File Format and hit Map File);
> > > > > > > >

Search Groups   Search Web   Sign In   Mail

> > > > > > > > >
> > > > > > > > > My general feeling in migrating from Orcad Capture/Layout
> > > > > > > > > to Kicad
> > > > > > is extremely good, it is costing me some work but in my opinion
> > > > > > it is definitely worth it.
> > > > > > > > >
> > > > > > > > > Regards,
> > > > > > > > > Michele
> > > > > > > > >
> > > > > > > > > --- In kicad-users@yahoogroups.com, "pg2845" <pg2845@> wrote:
> > > > > > > > >
> > > > > > > > > > Hi All,
> > > > > > > > > > I'm very interseted in replacing our ORCAD V9.1 Capture
> > > > > > > > > > and layout
> > > > > > with Kicad. Before spending an inordinate amount of time fooling
> > > > > > around with Kicad, I'd like to know if anyone out there is doing
> > > > > > the following with it.
> > > > > > > > > >
> > > > > > > > > > 1. Creating a PCB fab drawing with dimensions and drill
> > chart.
> > > > > > > > > >
> > > > > > > > > > 2. Creating a PCA assembly drawing using data from the
> > > > > > > > > > schematic,
> > > > > > such as corporate part numbers and descriptions in the form of a
> > text list.
> > > > > > > > > >
> > > > > > > > > > With Orcad this can all be accomplished within the
> > > > > > > > > > framework of a
> > > > > > single file which is directly connected to the defining
> > > > > > schematic file. By controlling which layers are on and off, one
> > > > > > can print the required documents to cover PCB fabrication and PCA
> > assembly.
> > > > > > > > > >
> > > > > > > > > > Thanks for your input.
> > > > > > > > > >
> > > > > > > > >
> > > > > > > >
> > > > > > >
> > > > >
> > > > >
> > > > >
> > > > >
> > >
> > >
> >
> >
> >
> >
> > ------------------------------------
> >
> > Please read the Kicad FAQ in the group files section before posting your
> > question.
> > Please post your bug reports here. They will be picked up by the creator of
> > Kicad.
> > Please visit http://www.kicadlib.org for details of how to contribute your
> > symbols/modules to the kicad library.
> > For building Kicad from source and other development questions visit the
> > kicad-devel group at http://groups.yahoo.com/group/kicad-develYahoo! Groups
> > Links
> >
> >

< Prev   Next >   **Reply**

Expand Messages   Author   Sort by Date

Re: Orcad V9.0 replacement with Kicad   pg2845 ✉   Apr 17, 2013
Hi All, I'm very interseted in replacing our ORCAD V9.1 Capture and layout with Kicad.   10:18 pm
Before spending an inordinate amount of time fooling around with Kicad,...

  Re: Orcad V9.0 replacement with Kicad   Michele   Apr 18, 2013
  Hello, I am in the same process of moving projects from Orcad Capture/Layout to   michele26771 ✉   12:12 pm
  Kicad. I think I am now confident with this process so I am happy to give my...

    Re: Orcad V9.0 replacement with Kicad   phil.guiet ✉   Apr 18, 2013
    Good morning Michele- Thanks for your reply. My initial post was general to   10:11 pm
    attract someone with the same target as mine. I'm glad to hear that you feel this...

Re: Orcad V9.0 replacement with Kicad Well, if you think of can be useful, why to hide it? If someone will be solving it later he will not find it... PK....	Pavel KREJCI krepa76 ✉	Apr 19, 2013 9:37 am
Re: Orcad V9.0 replacement with Kicad ... I agree, interesting topic, please continue here! Mitch....	Mitch Davis mitch_davis ✉	Apr 19, 2013 9:43 am
Re: Orcad V9.0 replacement with Kicad Hello All, Ok, Ok I will continue here :-) I will have a look to Phil files this weekend and also I will try to write something about my whole process to...	Michele michele26771 ✉	Apr 19, 2013 10:20 am
Re: Orcad V9.0 replacement with Kicad Yep - another vote to keep this thread going here - If nothing else, I might learn some better documentation techniques as a result. Cheers, Hamish...	Hamish Mead hamish.mead ✉	Apr 19, 2013 10:22 am
Re: Orcad V9.0 replacement with Kicad Hello Phil, Thanks for your file, I now understand better your needs regarding your second question. My answer about it (maybe someone else can give a better...	Michele michele26771 ✉	Apr 19, 2013 8:55 pm
Re: Orcad V9.0 replacement with Kicad Hello, To translate your Orcad Layout max file into a Kicad pcbnew brd file, please follow the following flow (here just sketched, for details please ask): 1)...	Michele michele26771 ✉	Apr 19, 2013 9:39 pm
Re: Orcad V9.0 replacement with Kicad That explanition is helpful. Thank you. Having used KiCAD for a while, and only rarely used ORCAD (well a lot long time ago), I have an observation about...	John Hudak hudak_john ✉	Apr 20, 2013 3:07 pm
Re: Orcad V9.0 replacement with Kicad Hello John, thanks. It would be really good to have scripts to automate the additional editing required. For example one additional thing that I do is running...	Michele michele26771 ✉	Apr 20, 2013 4:19 pm
Re: Orcad V9.0 replacement with Kicad Hi Michele- You've had a busy time since I started this! I have downloaded the application max2brd (for example from...	pg2845 ✉	Apr 22, 2013 4:48 pm
Re: Orcad V9.0 replacement with Kicad Hi Phil, yes, you are right :-) But there is of course to talk about the Orcad Capture to Kicad EEschema conversion yet. Regarding max2brd, I use the...	Michele michele26771 ✉	Apr 22, 2013 8:27 pm
Re: Orcad V9.0 replacement with Kicad Michele, I am able to compile and run max2brd using Cygwin. It works under Win 7 for me also. Don ... From: kicad-users@ yahoogroups.com...	DH donh20032003 ✉	Apr 23, 2013 8:08 am
Re: Orcad V9.0 replacement with Kicad Brilliant. Thank you very much Don. Regards, Michele...	Michele michele26771 ✉	Apr 23, 2013 10:02 am
Re: Orcad V9.0 replacement with Kicad Hello Don- Can you provide us with your max2brd.exe? I'm not capable of duplicating your effort without learning more than I have time for right now. Regards ...	phil.guiet ✉	Apr 23, 2013 1:38 pm

< Prev Topic | Next Topic >