ELECTRICALENGINEERING

How can I add a logo to PCB with Altium Designer?

Asked 2 years, 3 months ago Active 3 months ago Viewed 4k times

2

I am trying to add a logo to my PCB using Altium Designer 16.1 via "PCB Logo Creator" software. However it was not so neat. Do you have an idea to prepare better logo which would be taken from a photo or picture?

pcb altium

edited Jan 17 at 22:02
JYelton
17.7k ● 28 ● 96 ● 203

asked Jul 12 '17 at 8:38
layout789
103 ● 1 ● 10

Beware this can be a sticking point with vendors... you can easily end up with more data in the output trying to represent your logo than takes to actually represent the silkscreen or circuitry! – Chris Stratton Feb 26 at 22:47

5 Answers

5

A bitmap image can be copied and pasted into Altium designer if it is a true 1-bit monochrome image. I normally use MS Paint. It is not too bad if you start with a massive image and then shrink it in Altium - you can re-size it in Altium after it has been pasted in.

1 One sometimes nice trick is to put the logo on the solder mask layer instead of the silk and place a copper pour under it, then if you are doing an ENIG board you get the logo in gold. Not appropriate everywhere, and a positively bad idea with OSP, but it looks good in the right place. – Dan Mills Jul 12 '17 at 10:53

1 To future googlers: If your logo is converted to multiple primitives: Right click -> Create union from selected objects then Right click -> Resize union – C K Aug 31 '18 at 6:32

5

If it's a black/white logo which can be rasterized, you can also go the "Font" way. Use a Font creator tool (e.g. FontCreator, costs a little) to create a Font with a single letter (you can easily import vector graphics); then, assign that logo to a letter (e.g. L); then install that font in Windows and use it in Altium;

This works for both Schematic and PCB with the added benefit that it doesn't blow up your PCB in size (which the bitmap option does).

Disadvantage: Wherever you want to have your logo displayed, you need that font installed.

answered Jul 12 '17 at 10:12

Tom L.
6,880 ● 1 ● 13 ● 29

This is the best way to go (using a font). It scales to any size perfectly. Works in any application that supports true type fonts (providing a consistent look across all of your documentation), including Altium SCH/PCB, MS Office, and most anything else. Do this. – Chris Knudsen Jul 12 '17 at 14:05

I have done it, i really works. – layout789 Jul 12 '17 at 14:18

Ditto on creating your own font. This works perfect. Here is a free method I used to make mine: rafaltomal.com/how-to-create-and-use-your-own-icon-fonts – Joel Wigton Jul 13 '17 at 4:21

4

If you have your logo in vector form, you should export it as dxf which Altium can import. Might not work well with pictures, but since most Logos also come in "simplified" vector forms, it might still be an option.

The procedure of creating a "solid" logo from the DXF lines is a bit lengthy, but very reliable and gives high quality results. Here it is:

1. Import your dxf file (File>Import>DXF). Choose the correct units.
2. Use the Inspector panel to set the line width of the DXF geometry to a very small value, e.g. 1 mil
3. Draw a polygon pour around the shape (Place>Polygon pour), select pour over same net objects and disable removal of Islands. Set small values for neck width and arc approximation.
4. Adjust the clearance for the polygon to the DXF lines to 0 by adding a specific design rule
5. Repour the polygon, then select it, right click on it and use Polygon Actions>Explode selected Polygons to Free Primitives
6. Now your polygon consists of several copper regions. You can delete all regions that should not be filled, as well as your original DXF lines.
7. Group your regions by selecting them and creating a union from them (Tools>Convert>Create Union from selected objects).

There is also an official Altium video explaining this process.

answered Jul 12 '17 at 13:34

cx05

559 ● 2 ● 8

1

Per the official documentation, there are a few methods:

1. **TrueType Font**

   Embedding images as glyphs in a TrueType font allows you to take advantage of vector graphics and their inherent scalability. *I won't cover how to do this here, but there are many tools such as FontLab or a free online tool such as Fontstruct. This isn't an endorsement nor guarantee of these tools working.)*

2. **Pasting from Clipboard**

   Altium supports metafile formats such as 1-bit bitmaps, lines, arcs, and TrueType characters. *(Copying a 1-bit image from your favorite editor won't necessarily work unless it supports Windows metafile data; as I discovered attempting to use Irfanview.)* The documentation recommends pasting first in Microsoft Word, for example. The current layer will be the target for pasted data.

3. **Placing as OLE Object**

   This works through the `Place > Object from File` command in Altium Designer. Only the BMP format is mentioned.

answered Jan 17 at 22:25

JYelton

17.7k ● 28 ● 96 ● 203

1

To import an image (especially a complicated one) onto a PCB (in Altium) you're going to have to do some work. The basic issue is that your PCB image is going to be 2 tone (black/white, yellow/red, etc) and most source images are not.

PICK A GOOD SOURCE IMAGE

First you need to pick a good image as your source. The best images are going to not have fine details and will be made of large, solid colored, blocks.

PREPARE THE IMAGE

You'll need to import the image into a tool like the GIMP (GNU Image Manipulation Program). Once in GIMP you can use their tool-set to recolor and threshold the image so that it becomes a 2-tone version of the original. Save it as a BMP.

IMPORT INTO ALTIUM

1. Download the Altium Example Scripts and extract them to the `Examples` folder under your Altium installation.
2. Run PCBLogoCreator from within Altium
3. On the PCB Logo Creator dialog box click **Load** and select the BMP file. Click **Board Layer** and choose

MAKING THE LOGO REUSABLE

1. Highlight the whole logo and hit copy
2. Create (or open an existing) `.PcbLib` file and create a New Blank Component
3. Paste the logo into the blank component and save the footprint
4. Create (or open an existing) `.SchLib` file and create a New Puppet
5. Give the new Puppet a shape you can drop onto your schematics, a designator and name you like, and the select the Footprint you created above in the `.PcbLib`.

At this point you can drop your new Puppet onto any schematics and your "footprint" (which is actually your 2-tone image/logo) will be imported into the associated `.PcbDoc` file.

You can read a writeup with pictures and more details here: [How to put a Logo on a PCB in Altium](#).

edited Jul 13 at 1:34

answered Jul 11 at 20:09

begleysm

11 ● 2