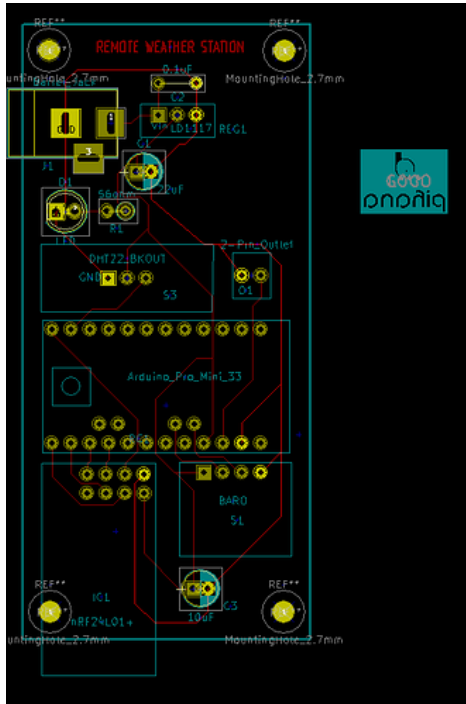# 🔒 Setting layer for LOGO footprint

Vagulus　　　　　　　　　　　　　　　　　　　　　　　　**Jun 26**

I have a LOGO footprint I wish to etch in the copper of the PCB

Jun 26

1 / 6
Jun 26

I set the active layer (little window on the menu bar) to F.Cu but, as you can see by the colour the logo appears in F.Silk.

How can I fix that?

Sep 24

Rene_Poschl　　　　　　　　　　　　　　　　　　　　　**Jun 26**

Kicad version?
How did you create the logo?
Can you share the footprint?

If you created it from any vector graphic you can use **svg2mod** (plus the open source tool inkscape) as it is able to put stuff directly on copper.

Vagulus　　　　　　　　　　　　　　　　　　　　　　　　**Jun 26**

> Rene_Poschl:
>
> Kicad version?

4.0.7

> Rene_Poschl:

> How did you create the logo?

Bitmap2Component

> Rene_Poschl:
>
> Can you share the footprint?

⬇ **PihanaLogo-**

Mirror.kicad_mod (20.1 KB)

Rene_Poschl   **Jun 26**

Use a text editor to replace F.Silks with F.Cu (or B.Cu as this is mirrored.)

The file with B.Cu: ⬇ **PihanaLogo-Mirror.kicad_mod** (20.1 KB)

From the 3d viewer in 4.0.6

Vagulus   **Jun 26**

Got it, Rene, Thanks.
Had to change some references to F.SilkS in the text file.

Thanks again

CLOSED SEP 24

This topic was automatically closed 90 days after the last reply. New replies are no longer allowed.