

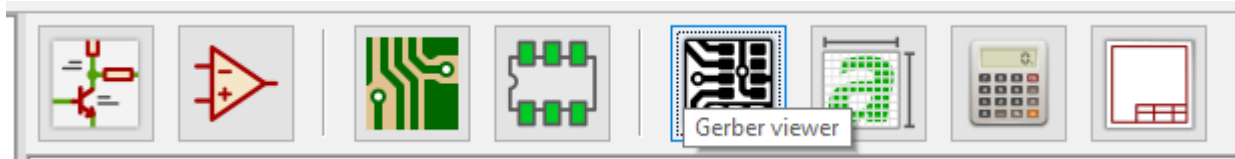
## Viewing and Editing Gerber File Using Kicad

Prerequisite

1. RS274 Gerber File and/or Excellon Drill file
2. Kicad (Version 5.0.1)

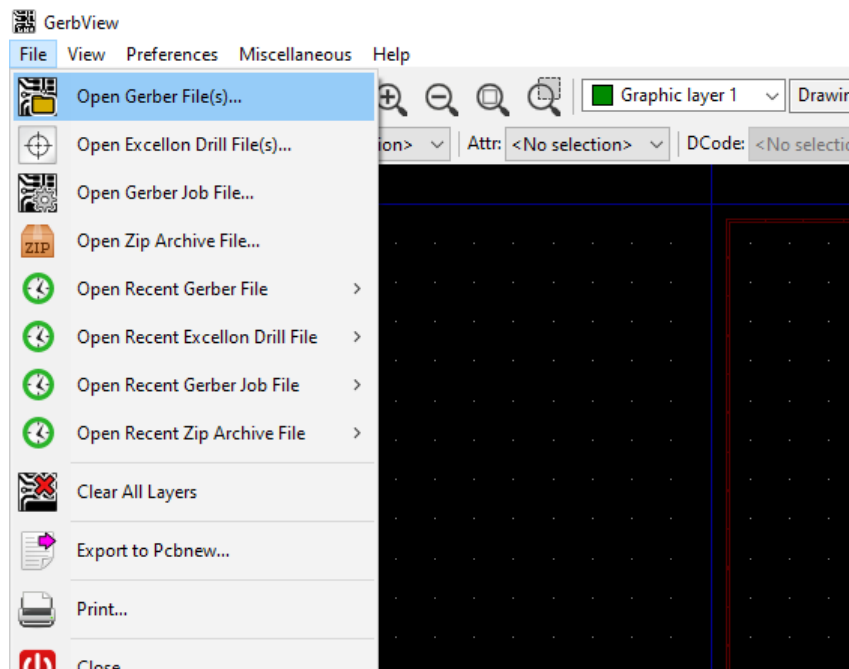
### 1. Open Gerbview

At home menu of Kicad, click on Gerbview icon to start viewing your gerber file.



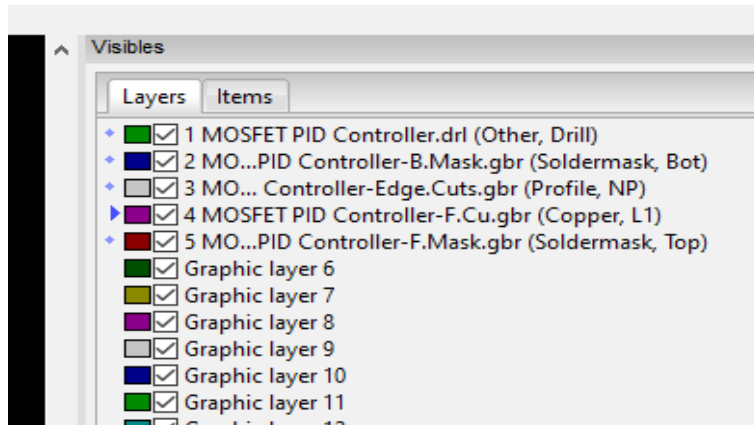
### 2. Open Gerber File

Go to “**File > Open Gerber File(s)**” and locate where your gerber file is. You can select multiple layer of gerber file at once and click OK to continue.

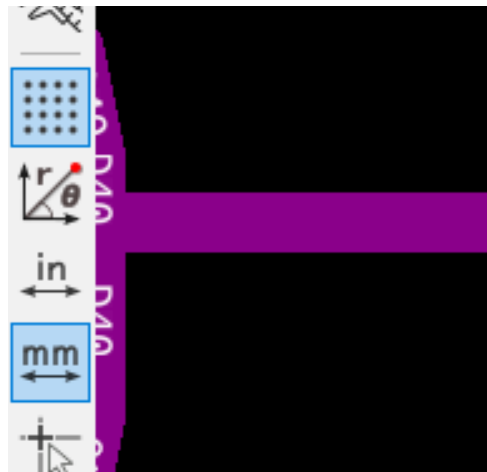


#### 4. Check thickness of trace/track/polygon size

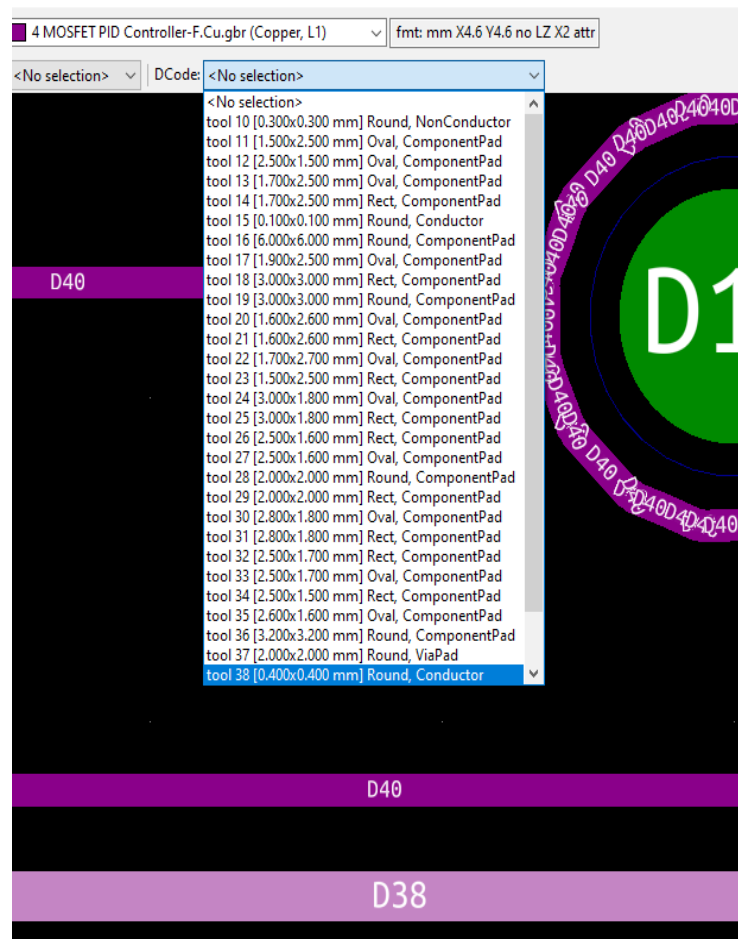
Make sure you activate the layer that you are interested in by clicking on the file name of your left panel. For this example, the F.cu layer was activated and indicated by the small arrow next to the layer color.



Make sure you set the proper unit for your drawing either mm/mils. On your left panel click on “in” or “mm” icon to choose your measurement unit”



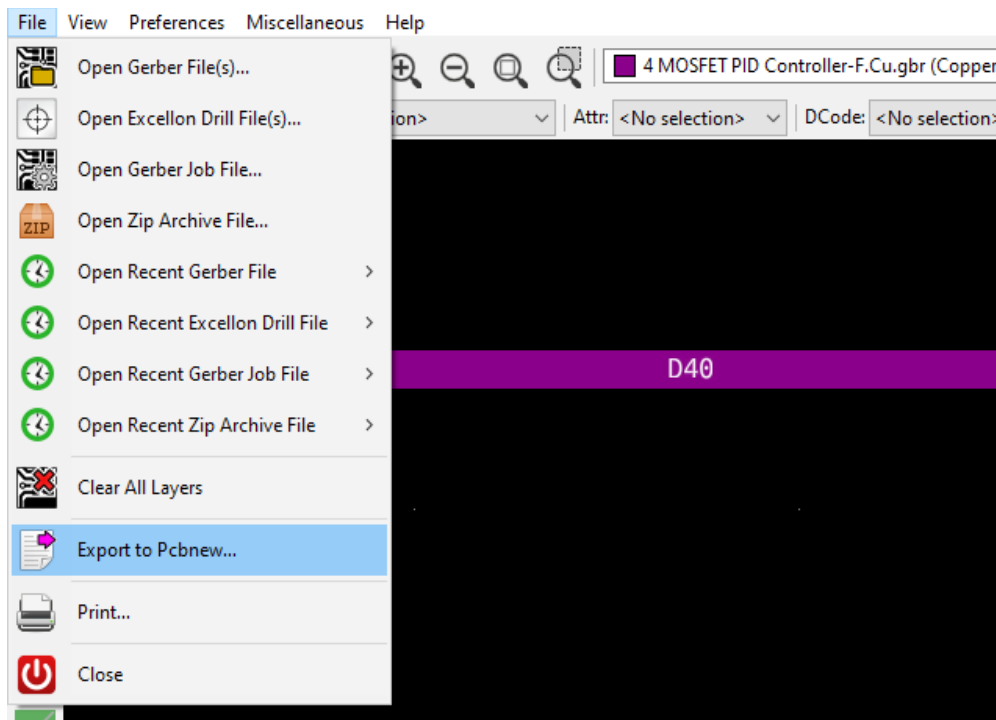
Here, the track that was interested is designated to DCode D38. The DCode windows will then list the polygon size of the trace (which is 0.4mm)



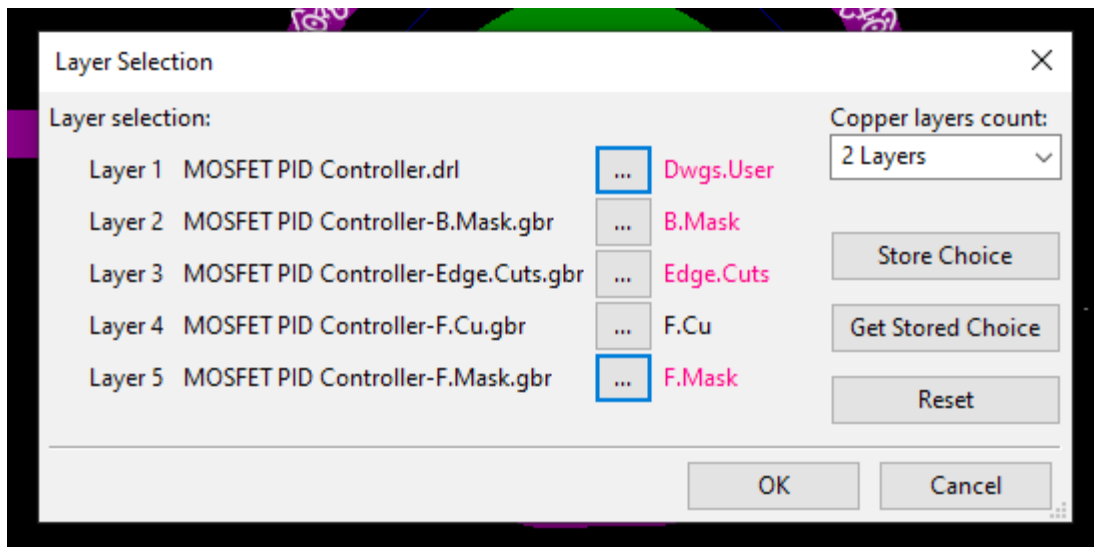
You also can check your drill tool size depending on fabricator specified.

## 5. Export to Pcbnew

Go to “**File > Export to Pcbnew**” and choose where you want it to be saved and name it. A file will be created based on what you set.



When saving the file, make sure you designate the proper layer to each file.



Once saved, Click on the file that you just created to start the Pcbnew application. You can refer to other note on how to use Pcbnew to edit your design.