KiCAD - Setting Your PCB Gerber Origin

Ron Taylor G4GXO

1. Introduction

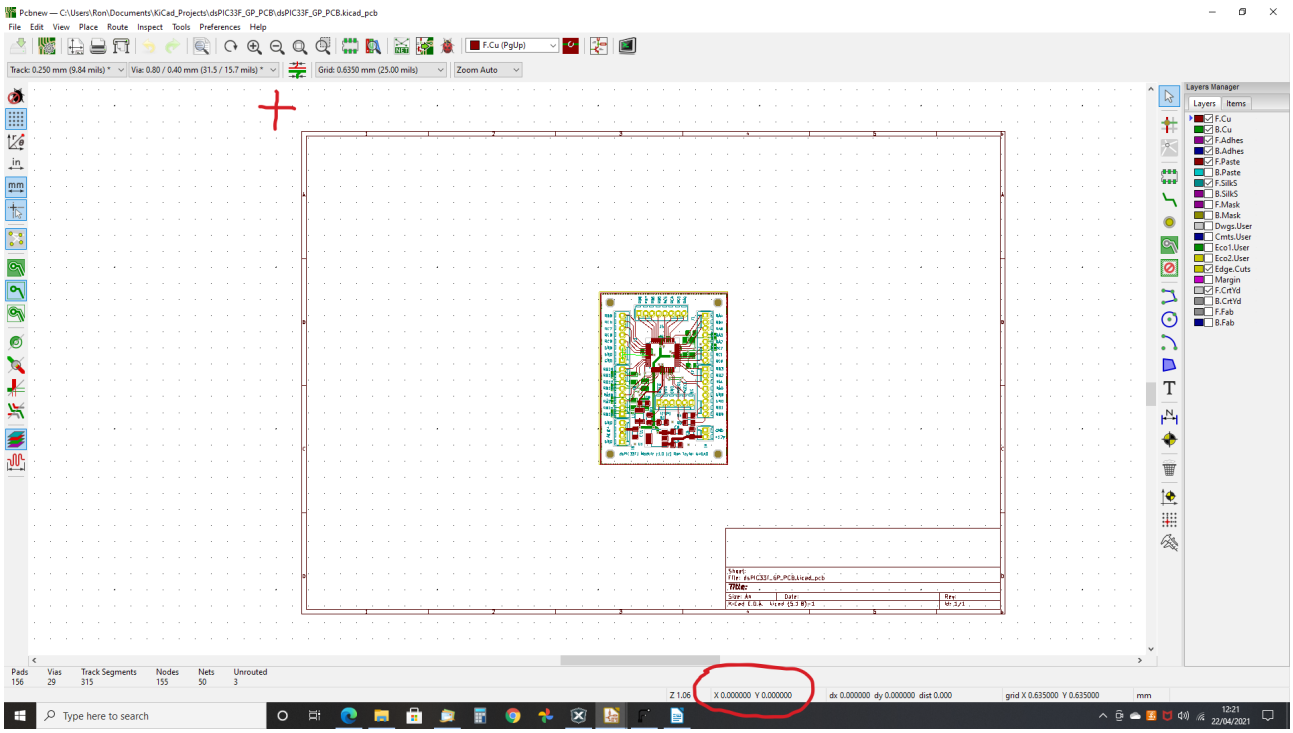
KiCAD is a powerful schematic and PCB layout application that produces PCB Gerber layout and Excellon drill Computer Aided Manufacture (CAM) files for board manufacture. An X-Y coordinate system referenced to a zero “origin” is used to describe the layout.

If CAM files are sent to PCB manufacturer then they will take care of any arbitrary X-Y origin offset when they panelise a layout for manufacture. However, should you wish to make your own boards on a CNC from CAM files, you may come across an unexpected problem in that the X-Y coordinates of your design as defined y KiCAD, fall outside the co-ordinate area of your CNC table!

Fortunately this is an easy one to fix using a handy option within the KiCAD Plot menu which is used to generate the CAM files.

2. X-Y Coordinates

When KiCAD produces a layout from schematic data, it sets an X-Y origin (0,0) position in Pcbnew just outside the top left corner of the layout design frame. You can see this by examining the X and Y values shown at the bottom of the screen in Pcbnew as you move the cursor over the screen.



When you create the CAM files for your design, unless you tell KiCAD otherwise it will use this default origin as the reference for the design. When you generate G-code files for your CNC, (using a program such as FlatCAM), the G-code coordinates faithfully reflect the screen position of the design in Pcbnew. This may make your G-code data fall beyond the X-Y reach of your CNC

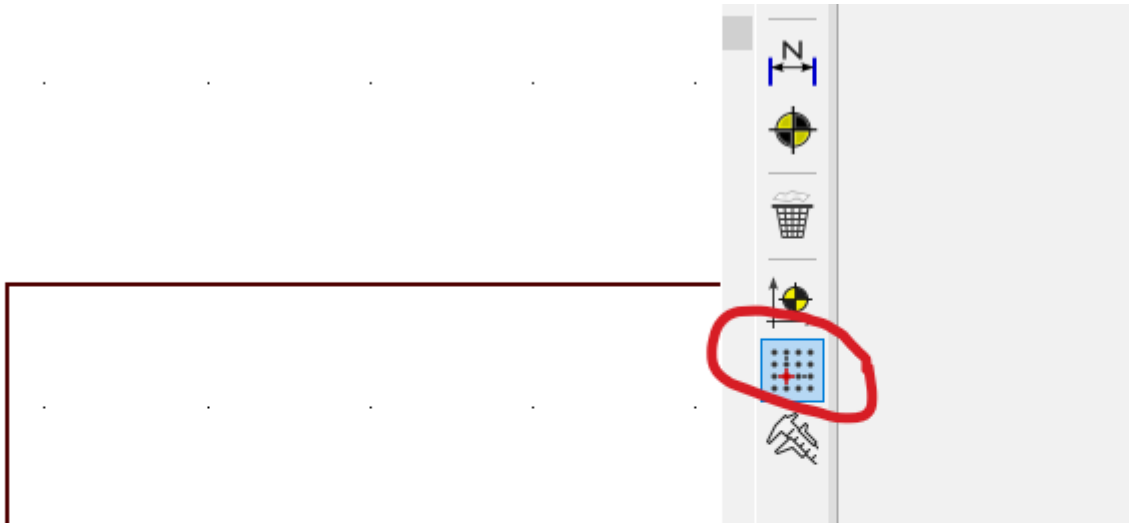
cutting tool. A new origin is required, one that can be set to place your design origin anywhere that you want it.

3. Setting A New Grid Origin

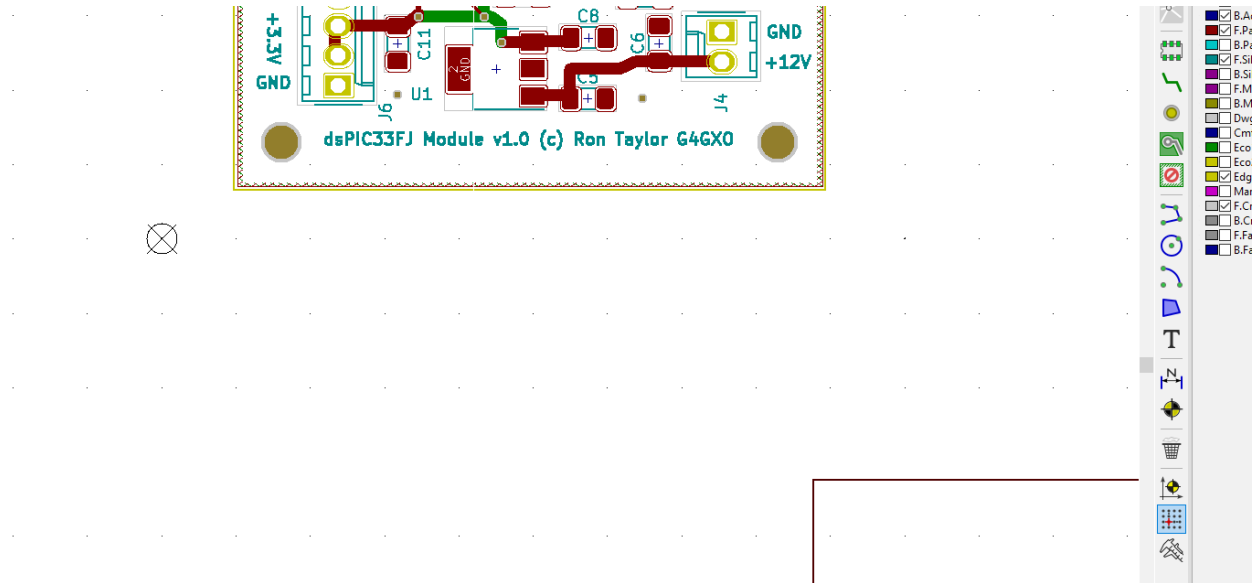
When you are ready to produce the CAM files from your project you also need to set a point on the layout in Pcbnew that will allow you to place the design somewhere convenient on your CNC table. This just involves setting a new Grid Origin close to one of the layout corners. By using the zeroing function on your machine's G-code driver, this new origin will be set at the zero position on the table. The Grid Origin is set and applied as follows;

3.1 Place the Origin

In Pcbnew, on the right hand menu bar select the Set Origin Point for Grid icon. Move your cursor onto the design to the desired point for the new origin and Left Click to place it.



An "X" within a circle will be placed at that point marking the new Grid Origin.

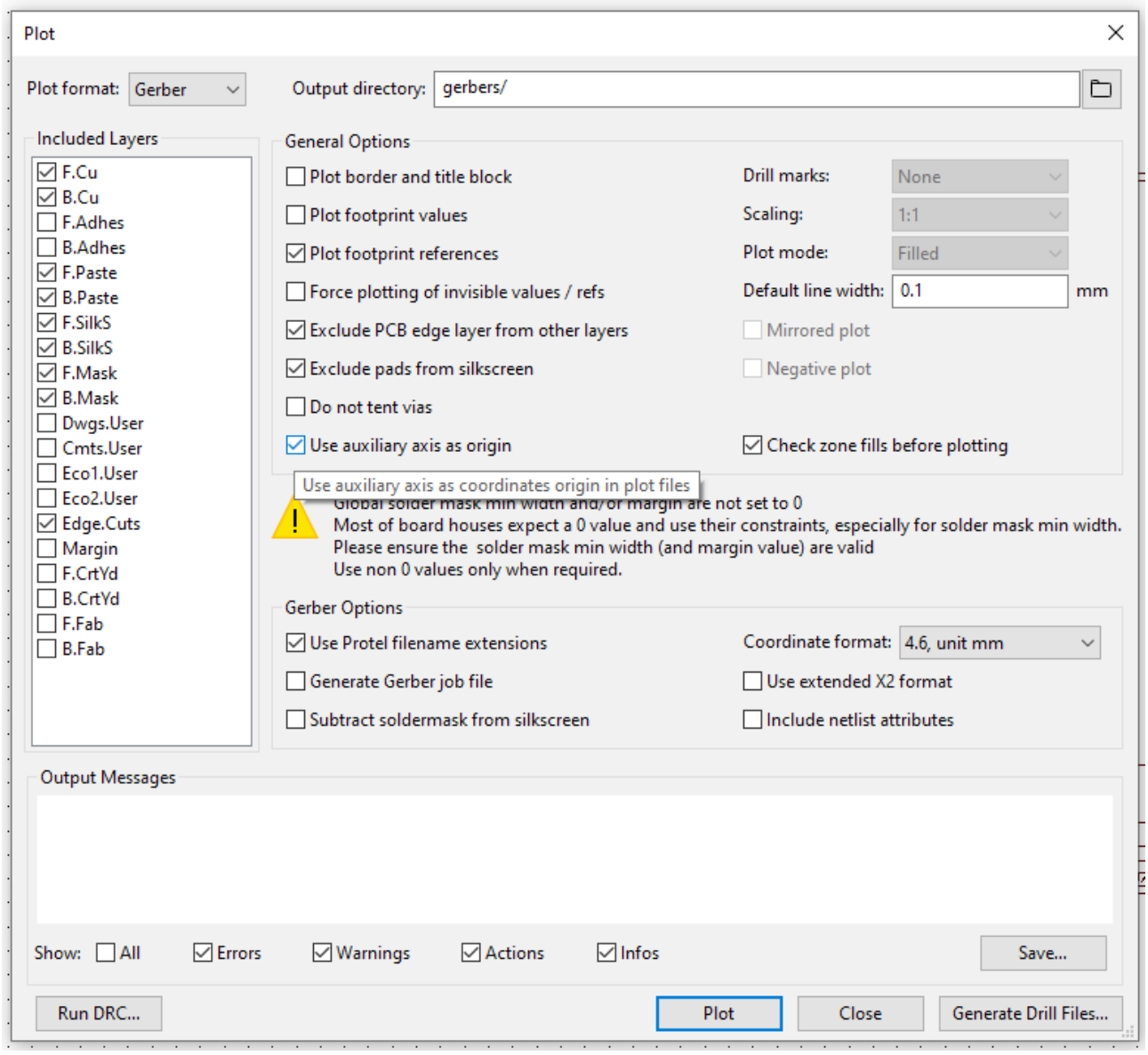


To change the position of the Grid Origin, select the Set Grid Origin Point and define a new point using the same process as above. The Grid Origin will be moved to this new point.

If you wish to delete the origin, you can do so by selecting View/Grid Settings… A sub menu will open, press Reset Grid Origin and the Grid Origin will be cleared.

3.2 Applying the Grid Origin

Having set a Grid Origin on your design we need to make sure that the Plot function uses this as the grid zero reference when generating the Gerber and drill files. This is done by opening the Plot screen using the top menu bar icon or File/Plot and within the General Options checking the box labelled “Use auxiliary axis as origin”.



You can now proceed with generating the layout Gerbers by pressing the “Plot” button and Excellon drill files by pressing the “Generate Drill Files...” button. All CAM data will now be referenced to the Grid Origin that you defined and the design can be conveniently placed anywhere on your CNC table by using the machine’s G-code driver’s zero function.