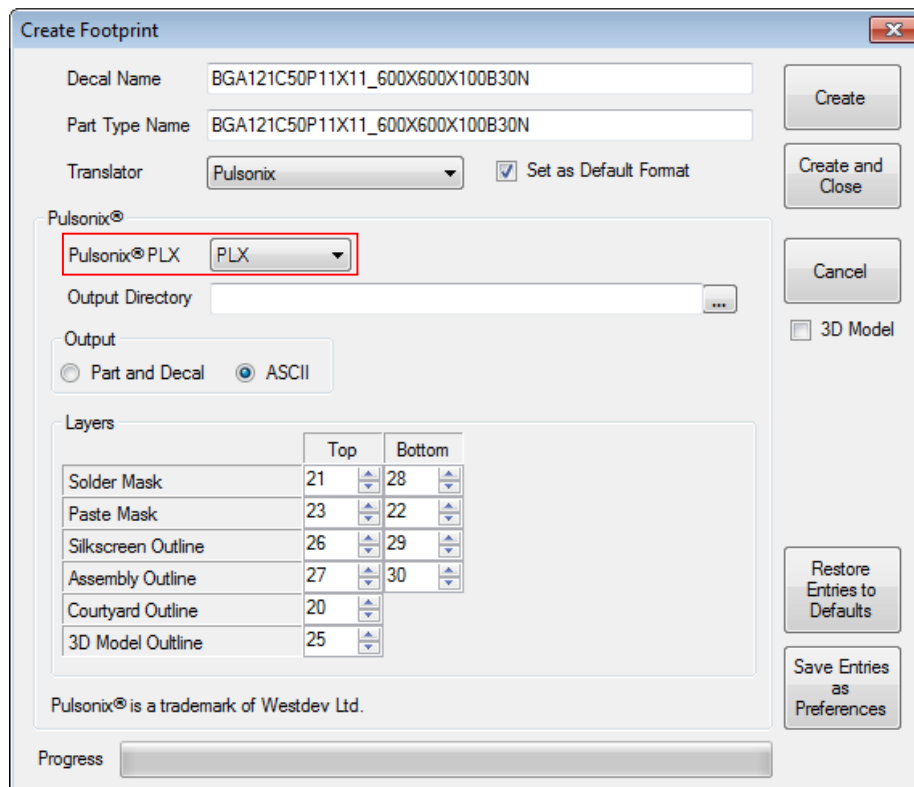


Pulsonix PCB Library Expert Interface

Exporting from PCB Library Expert

Choose the **Pulsonix PLX** format option within the PCB Library Expert dialog and using **Create**, export your footprint to a file using the file extension .plx.



Importing PLX files to Pulsonix

During the import, you can optionally elect to use a **Technology File** and **Mapping File**. These can be your own files or you can download ones that are supplied by Pulsonix on the web site under <http://www.pulsonix.com/downloads/tools/supplementary/PCBFootprintExpertPulsonixPLX.zip>

Technology File

The Pulsonix technology file PCBFOOTPRINTEXPERT_IMPORT.ptf contains line styles, text styles, layers, layer classes, and pad styles set-up to aid import of footprints generated from PCB Footprint Expert into Pulsonix.

Before starting the import, you should save the technology file into a location that can be browsed to using the **Folders** dialog and **Technology** page.

As an alternative method, after importing the footprints, you can selectively load these Technology settings into your existing technology using the **Load Technology** button on the **Setup** menu.

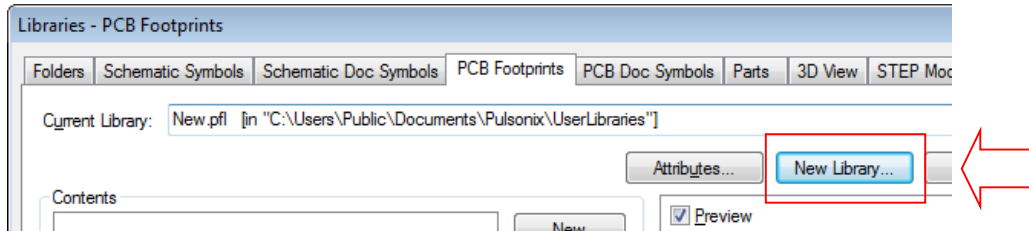
Mapping File

The mapping file supplied further facilitates the import but mapping import names to names used by Pulsonix. This can of course, be edited to suit your own requirements and layer names etc.

Importing the PLX file into Pulsonix

From within Pulsonix, from the **Setup** menu, or from the PCB toolbar, select **Libraries** (or Libraries icon).

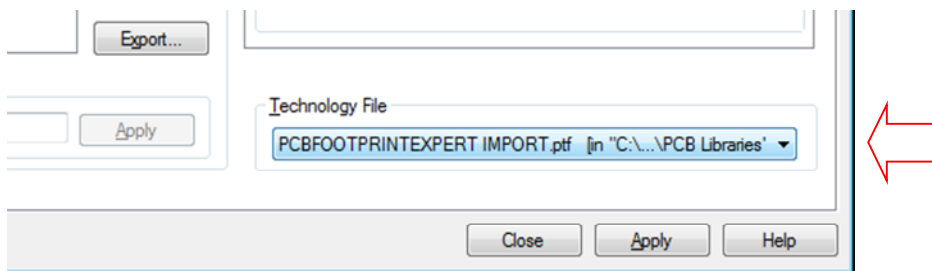
Choose the **PCB Footprint** tab.



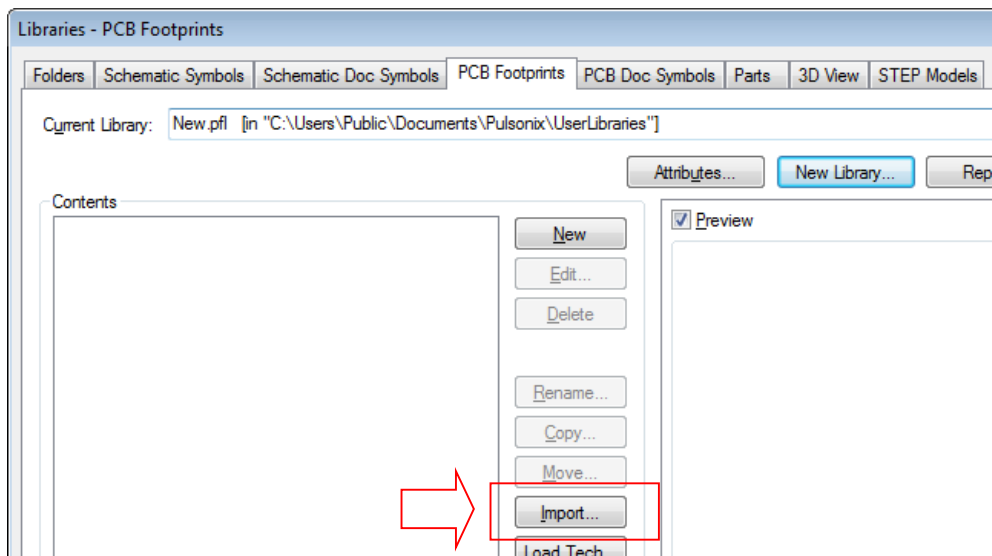
Use the **New Library** button to create a new PCB Footprint library.

The new library name will be displayed in the **Current Library**: list box.

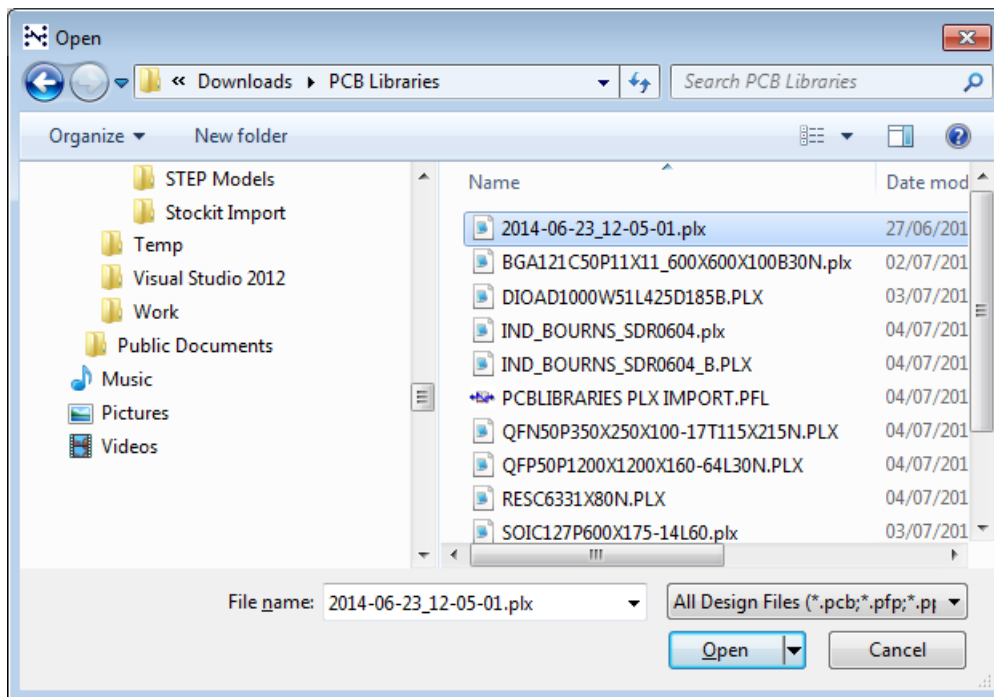
Before importing the .plx files, change the Technology file to either your own file or the one downloaded previously: PCBFOOTPRINTEXPERT IMPORT.ptf



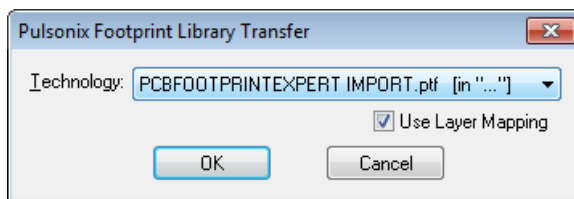
Now import your footprint using the **Import** button on the dialog:



From the **Open** dialog, choose the .plx file to import. Press **OK**.

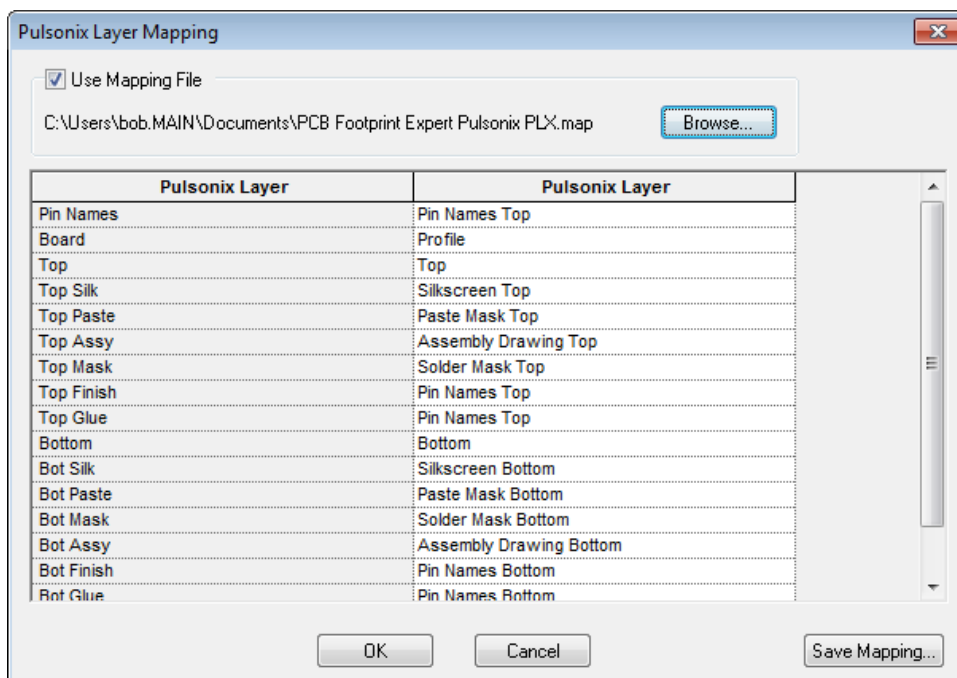


The **Pulsonix Footprint Library Transfer** dialog is displayed:

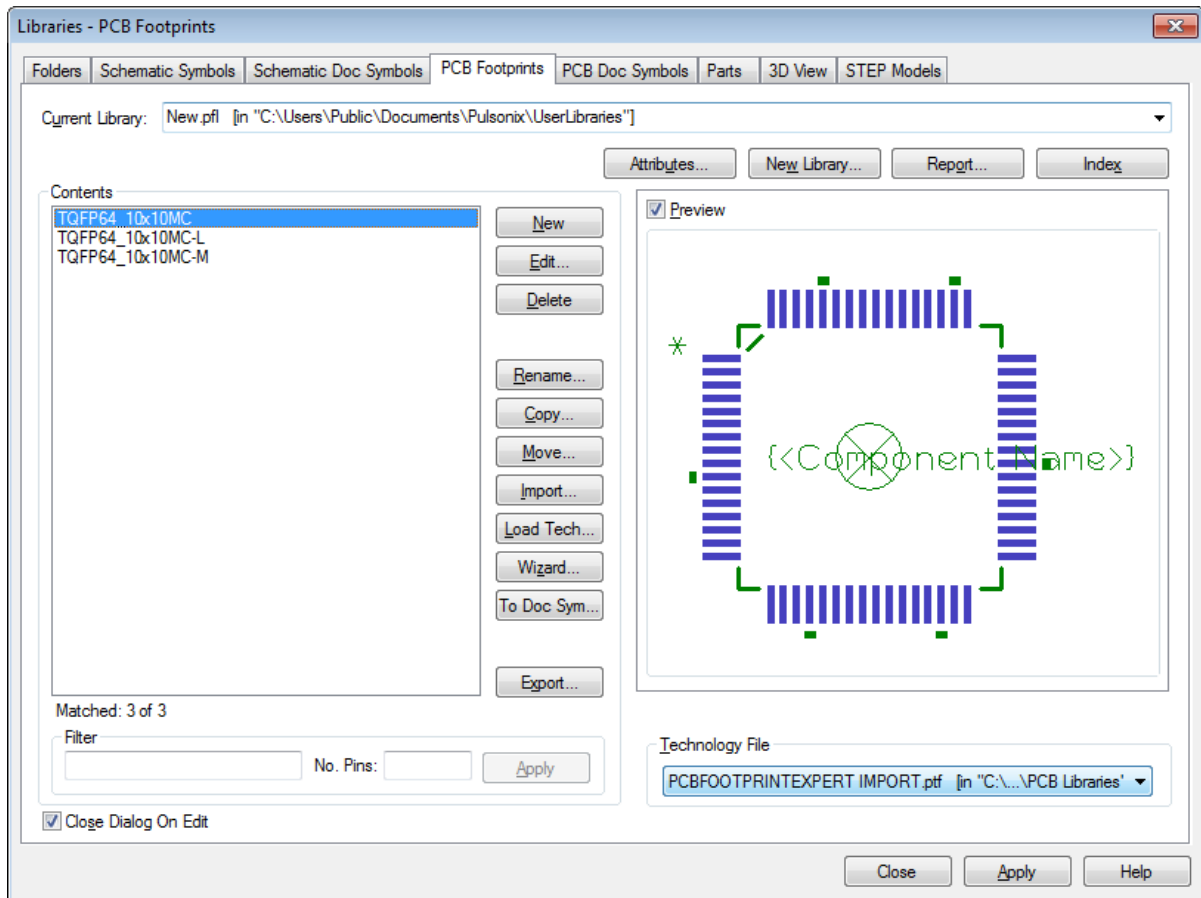


The Technology file will be pre-selected for you (because you set it up in the previous operation). Use this one or select your own Technology file. By selecting a Technology file, you can also optionally select the Use Layer Mapping check box.

If you select this, when you press **OK** to run the import, the **Layer Mapping** dialog is displayed.



When you press the **OK** button, the import will start. You will be returned to the **PCB Footprints** page of the Library Manager dialog from where you can view your newly imported footprints.



Once in Pulsonix, you can edit this footprint in the normal way using the **Edit** button.

Remember, you will also need to create a Part definition using the **Parts** tab in order to use this footprint in your PCB design.