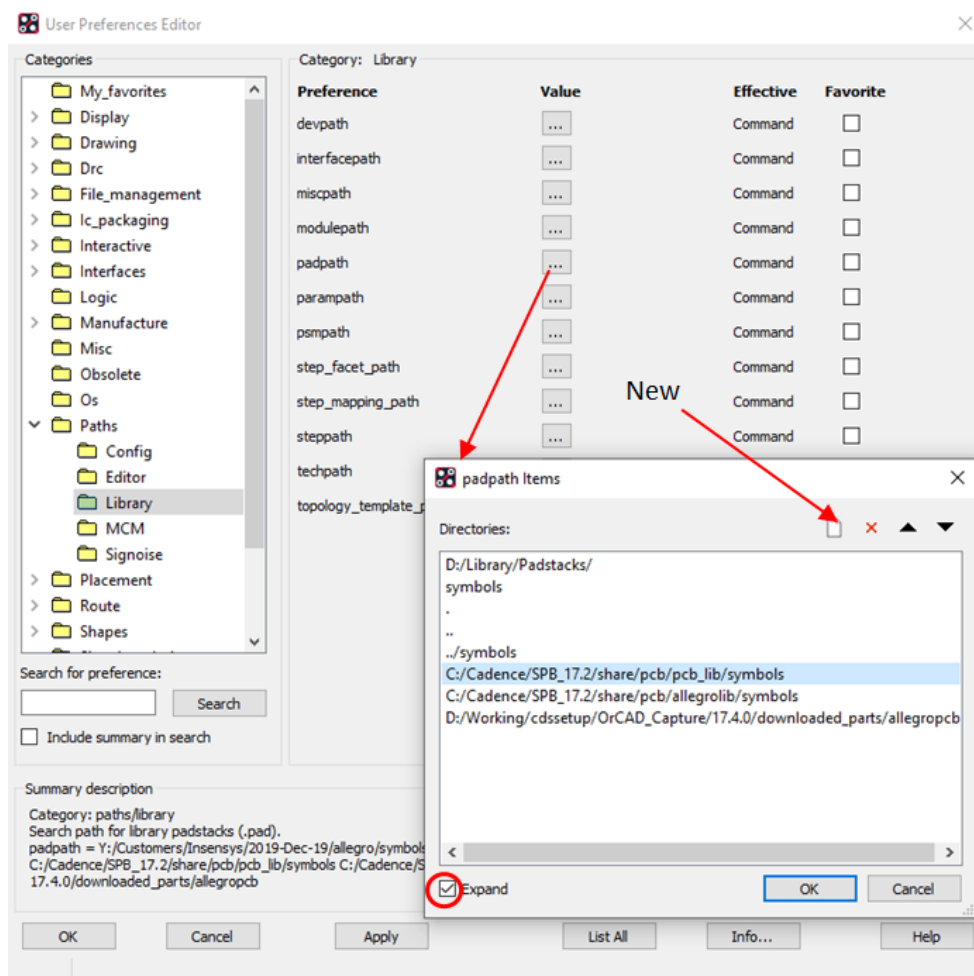## Introduction

This technical note describes how to set PCB Editor up so that you use a user or company defined PCB Footprint, padstack and step model library.

How to define padpath, psmpath and steppath preferences.

PCB Editor stores the default paths to where the PCB Footprints, padstacks and step models are stored. Many customers require a company specific footprint and padstack library. PCB Editor uses the padpath setting to look for directories that contain padstacks, psmpath to look for directories that contain symbols (footprints) and steppath for directories where the step models are stored. You can have multiple entries defined and PCB Editor looks in the first directory in the list, if it finds the footprint name it will use it, if it cannot find the footprint name it will look in the next directory and so on.

To start, open PCB Editor and go to Setup > User Preferences > Paths > Library folder.

We now want to set the padpath and psmpath to point to a specific directory. Click on the Value box next to padpath. The padpath GUI will appear. Check the expand checkbox at the bottom of the GUI (if it is not checked) to see the expanded list of directories. Then click on New and add the new padpath by browsing your windows directory. Click OK to confirm. The new directory is added to the bottom of the list. You can then select the new directory path with the LMB (left mouse button) and use the arrow keys to move this directory path up to the top. This would then mean that the new path added is the first directory that PCB Editor looks in for padstack names. This directory normally contains any files with a *.pad extension.

Many users will remove all the other entries in this GUI so that PCB Editor only looks at their company standard folder for padstacks. It is recommended to leave at least the local job directory .(dot) in the list. Click OK and the padpath is now defined.

Repeat the above steps for the psmpath. This directory should contain any files with a *.dra, *.psm, *.bsm, *.ssm, *.fsm, and *.osm extension. You also need to repeat these steps for the steppath. This directory should contain any step models (*.STP and *.STEP).

You have now defined the directory structure for PCB Editor. These settings are written automatically to the env file which is the default setup for PCB Editor. This is stored in the HOME environment location %HOME%\pcbenv folder.