

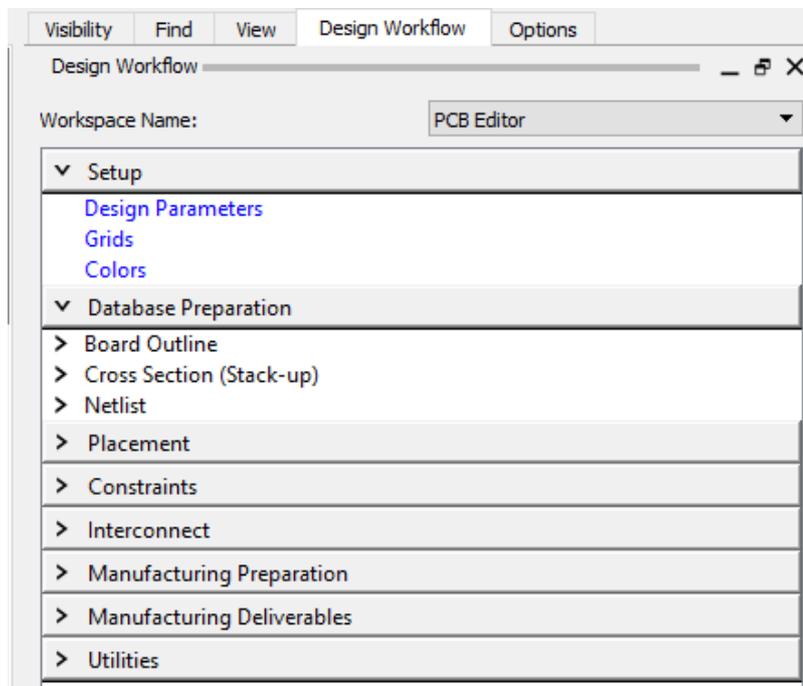


How to customise the Workflows in PCB Editor

PCB Editor Workflows

Workflows are available inside PCB editor to help users follow a flow of steps when designing a pcb. There may be instances where you would like to add a new workflow into the Cadence® Allegro® and OrCAD® technology PCB Editor.

The default workflow is an xml based text file that describes the groups and actions that are in the UI. You can customise these to suit so that instead of looking for the menu commands, they can be found more easily through the use of a workflow. The workflow pane can be docked in any of the axis of the working area or docked to create tabs with other panes (as shown below). They can also be undocked and placed onto another monitor.



You can create a new workflow for PCB Editor by making a copy of the workflow.xml file and provide a new name for the file that is in <your_install_dir>\share\pcb\text\workflows and edit it in a plain text editor such as Notepad, although using a more intelligent text editor like Editpad Pro can assist with open and closing statements like workflow to ensure you're editing is accurate.

Line 2 of this file contains the name of the workflow. By changing the name of the workflow you can create your own example. You **MUST** change the name from the default of PCB Editor.

From

```
<workspace version="1.0" name = "PCB Editor" defaultTool="PCB">
```

To

```
<workspace version="1.0" name = "My Workflow" defaultTool="PCB">
```

How to customise Workflows in PCB Editor

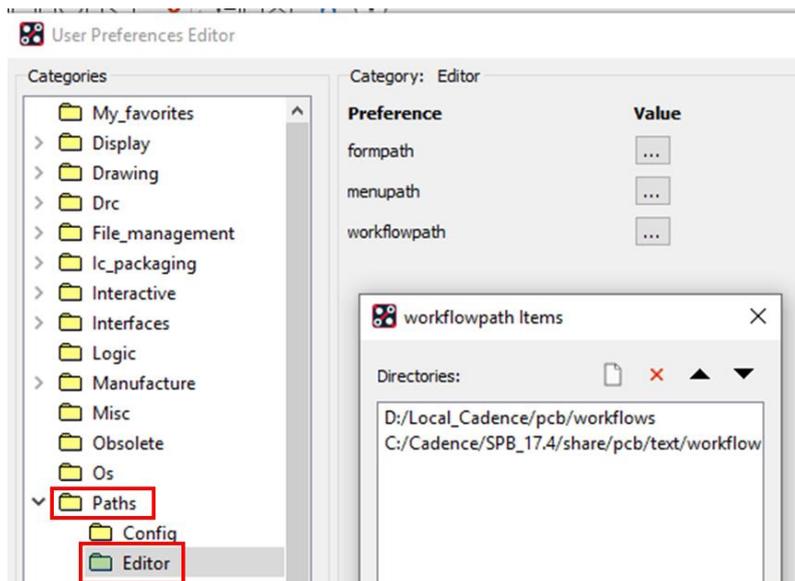
Create a location that you will save the file to and save it. An example would be to add a Workflow directory to your %home% or %CDS_SITE% directory

Example:

C:\SPB_Data\pcbenv\Workflows

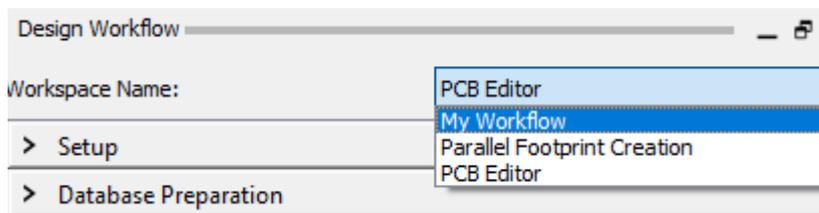
In PCB Editor go to **Setup > User Preferences**

In the Paths/Editor category select the workflowpath epsilon (...) and select the **New (Insert)** icon



Browse to the directory you created above add the new entry. Adjust the position to make this the top of the directory list and then OK all the forms.

Starting PCB Editor will now show the workflow you added with the Workspace name pulldown loading your workflow.xml file



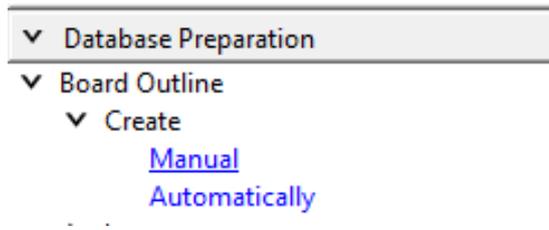
You can add or delete items in the workflow by adding a new section or removing a section.

The Bold name in the Workspace is the <workflow name> entry

This is then broken down into task name and then if needed even further with group name and task name entries

In the example below, the group name is Board Outline with a subgroup name of Create and then the task name Manual, and Automatically.

How to customise Workflows in PCB Editor

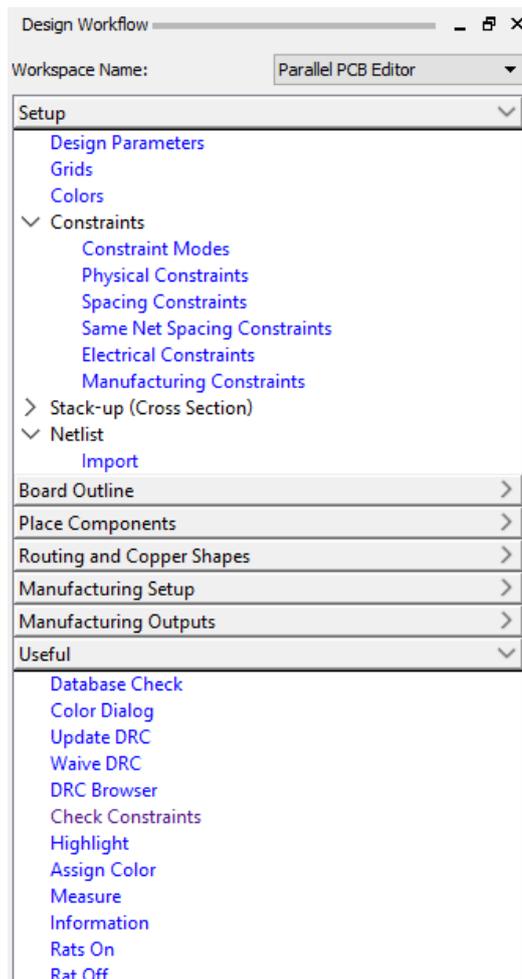


The .xml file has the following:

```
<workflow name="Database Preparation">
  <group name="Board Outline">
    <group name="Create">
      <task name="Manual" command="FORM mini class BOARD GEOMETRY; FORM mini subclass
DESIGN_OUTLINE ;shape add"></task>
      <task name="Automatically" command="board outline"></task>
    </group>
  </group>
```

To add entries, you would open the copy of the .xml file in a plain text editor and add the Workflow name, group name(s) and if needed sub-group names and the task.

As an example the Parallel PCB Editor workflow is shown below.



For the "Useful" workflow shown above you would need to edit the xml file as per:-

```
</workflow>
<workflow name="Useful">
  <task name="Database Check" command="dbdoctor"></task>
    <task name="Color Dialog" command="color192"></task>
    <task name="Update DRC" command="drc update"></task>
    <task name="Waive DRC" command="waive drc"></task>
    <task name="DRC Browser" command="browse drcs"></task>
    <task name="Check Constraints" command="cns show"></task>
    <task name="Highlight" command="hilight"></task>
    <task name="Assign Color" command="assign color"></task>
  <task name="Measure" command="show measure"></task>
    <task name="Information" command="show element"></task>
    <task name="Rats On" command="rats all"></task>
    <task name="Rat Off" command="unrats all"></task>
</workflow>
```

To assist in locating the default commands you can type scriptmode +e followed by a return at the PCB Editor command line. Now every command you invoke from a menu or icon will show the actual command syntax used and this is the syntax that is used to drive the command name.

The link below contains a zip file that has a PCB Editor workflow and a PCB Footprint workflow supplied by Parallel Systems Ltd.

http://www.mediafire.com/file/jue5a5zu3wh2jg0/Parallel_Workflows.zip

The following are trademarks or registered trademarks of Cadence Design Systems, Inc. 555 River Oaks Parkway, San Jose, CA 95134
Allegro®, Cadence®, Cadence logo™, Concept®, NC-Verilog®, OrCAD®, PSpice®, SPECCTRA®, Verilog®

Other Trademarks

All other trademarks are the exclusive property of their prospective owners.

NOTICE OF DISCLAIMER: Parallel Systems is providing this design, code, or information "as is." By providing the design, code, or information as one possible implementation of this feature, application, or standard, Parallel Systems makes no representation that this implementation is free from any claims of infringement. You are responsible for obtaining any rights you may require for your implementation. Parallel Systems expressly disclaims any warranty whatsoever with respect to the adequacy of the implementation, including but not limited to any warranties or representations that this implementation is free from claims of infringement and any implied warranties of merchantability or fitness for a particular purpose.