



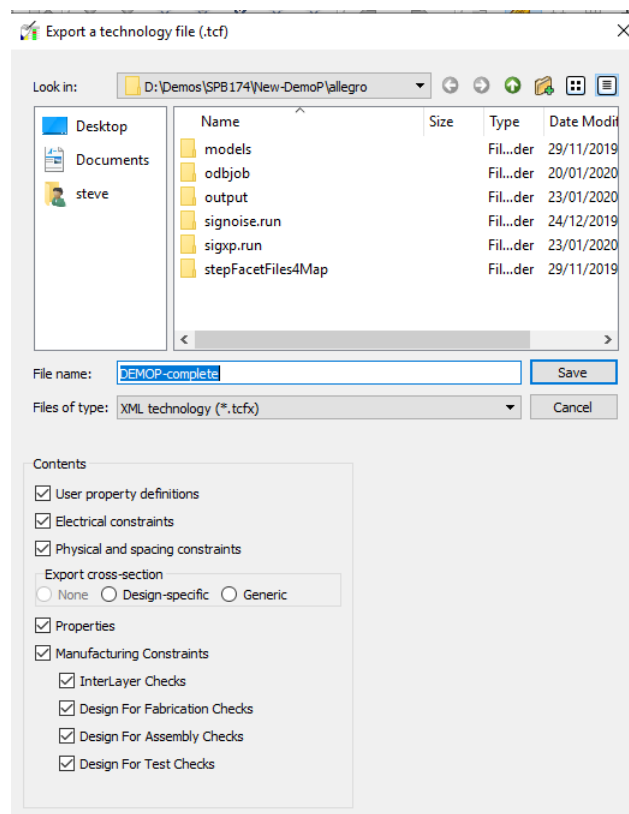
## Introduction

Many companies today use a default set of PCB Constraints (or rules) that are standard across all their designs. This may just be, for example, a line (track) thickness for signal and power, line-line or pad-pad spacing's, or may include differential pair rules, matched length rules. They may also have a need for a standard stackup for specific PCB designs. A standard 2-layer, 4-layer, 6-layer board with correct layer names and materials defined. PCB Editor gives users the option to create and store tech files (\*.tcfx) which contain the layer and rule information for a PCB design.

## How to create a Technology File.

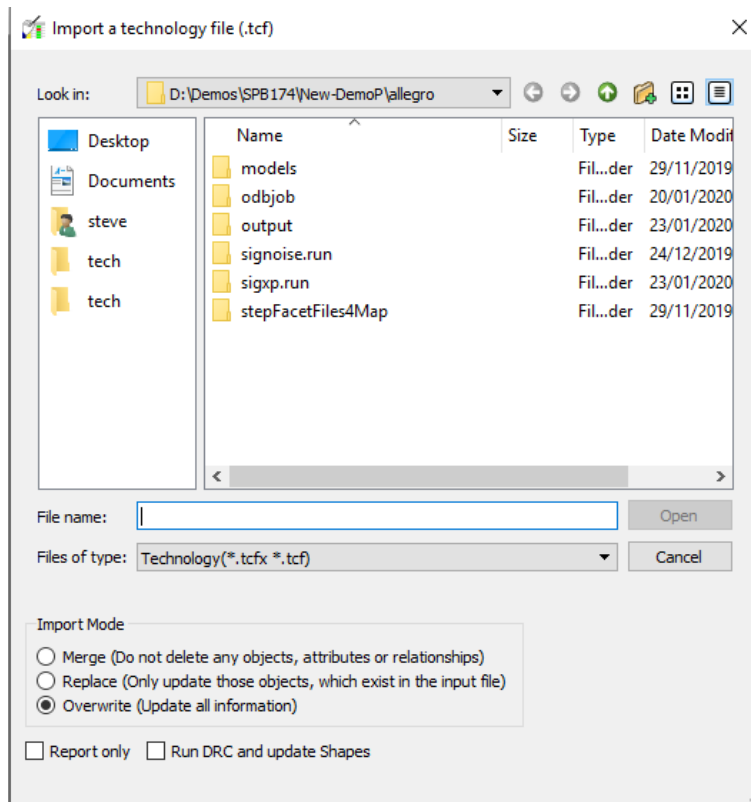
Start with a PCB Editor Board (filename.brd) file that already has the defined layers and rules that you wish to use as a default. Both options create a .tcfx file. You now have two options, either:-

1. Use File > Export > Techfile (Allegro) or Export > More > Techfile (OrCAD), specify an output filename and click on Export. When you write out the techfile all instances of all constraint sets are written to the techfile. You cannot specify which constraints are exported. Results of the export appear in the techfile.log file in the current directory.
2. Open Constraint Manager using Setup > Constraints > Constraint Manager (Allegro) or Setup > Constraints (OrCAD), then from the File menu use File > Export > Technology File, specify an output filename and click on Save. You can use the **Contents** option of this command to control the contents of the file. These options include the current design's cross-section, constraints (electrical, physical and spacing), user property definitions, and manufacturing constraints.



To import a technology file into a PCB Editor Board file (.brd) you also have two options, either:-

1. File > Import > Techfile (Allegro) or Import > Techfile (OrCAD), browse to the technology file (.tcf) and click on Import. When you read the techfile all values in the design are overwritten. If a constraint in the techfile does not exist in the design, it is added. If an error occurs in the techfile, the tool continues reading the file, writing warning and error messages, but does not update the design. Use this command to import either a legacy techfile (.tech) or the new Technology Constraint File (.tcf) file into your design.
2. Open Constraint Manager using Setup > Constraints > Constraint Manager (Allegro) or Setup > Constraints (OrCAD), then from the File menu use File > Import > Technology File.



Using this form, you can import the techfile in one of three modes: **Merge**, **Replace**, or **Overwrite**. Overwrite mode is enabled by default. These options give users more control about what is imported in PCB Editor.

There are two further options:-

**Report Only** - Produces a text description of the import results without performing the import task. The report is similar to the report generated by the File > Import > Constraints command in Constraint Manager.

**Run DRC and update Shapes** - When enabled, Constraint Manager first imports constraints, and then runs design rule checks and updates dynamic shapes; when disabled, no design rule checks, or shape updates occur on import. The above two check boxes and the different import modes are not available when importing from within PCB Editor.

It is recommended that Users import or export constraints through the Tech file options available directly in Constraint Manager. This preferred method allows more flexibility for constraints.

Another useful feature is the ability to use the Library button to access a central store where companies can store techfiles. This may not be necessary because you have the ability to Browse the windows directory, but the Library function automatically takes the user to the correct network location for techfiles. To use this option, you need to use Setup > User Preferences > Paths > Library > techpath. Click on the value button and define the path to the location of the techfiles. Once stored when the Library button is selected, that location is used to locate techfiles.

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.