FpcRef

A Panelization Utility for FreePCB

User Guide

Version 1.10
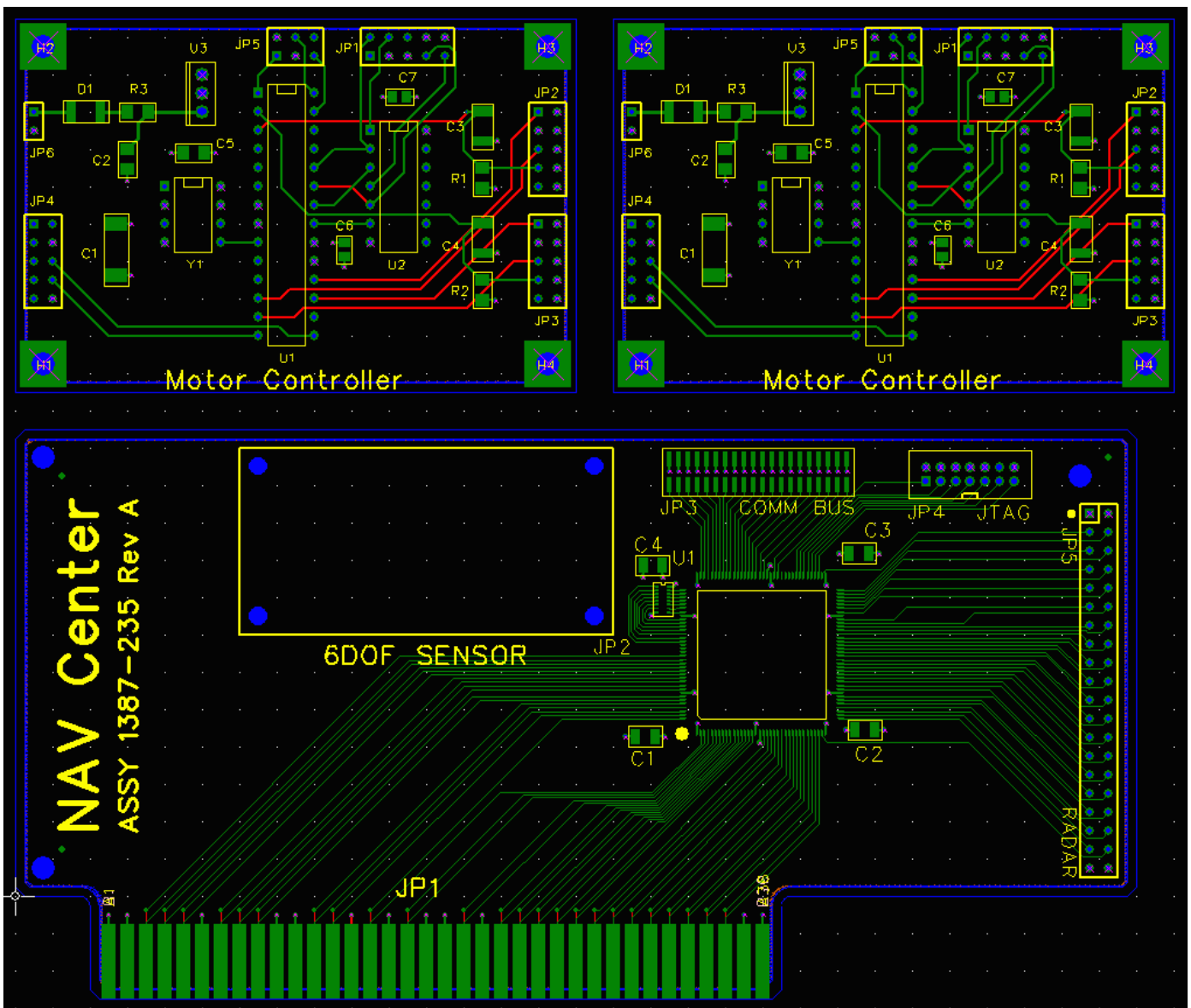
Contents

FpcRef End User License Agreement (EULA)

General

FpcRef Software is distributed as Freeware. You may use the Software on any number of computers for as long as you like. The Software is NOT Public Domain software. You are allowed to freely distribute the Software, but Bruce Parham retains ownership and copyright of the Software in its entirety. You may use and/or distribute the Software only subject to the following conditions:

1. You may not modify the program or documentation files in any way.

2. You must include all the files that were in the original distribution.

3. You may not reverse engineer, decompile, or otherwise reduce the Software to a human perceivable form.

4. You may not sell the Software or charge a distribution fee, except to recover the media costs.

5. You understand and agree with this license and with the Disclaimer of Warranty and the Limitation of Liability printed below.

6. The Software may be bundled and distributed together with other software products provided the same conditions apply.

Agreement to this license

By using, copying, transmitting, distributing or installing the Software, you agree to all terms of this license. If you do not agree to all of the terms of this License, then do not use, copy, transmit, distribute or install the Software and immediately remove the Software from your storage device(s).

Warranty disclaimers and liability limitations

The Software and related documentation are provided "as is", without warranty of any kind. None whatsoever. Bruce Parham disclaims all warranties, express or implied, including, but not limited to, warranties of design, merchantability, or fitness for a particular purpose or any warranty of title or non-infringement. Bruce Parham does not warrant that the functions contained in the Software or documentation will meet your requirements, or that the operation of the Software will be error-free, complete, or that defects in the Software or documentation will be corrected.

Limitation of Liability

Under no circumstances shall Bruce Parham be liable for any lost revenue or profits or any incidental, indirect, special, or consequential damages that result from the use or inability to use the Software or related products or documentation, no matter what legal theory it is based on.

Copyright and trademarks

FpcRef is Copyright © 2007-08 by Bruce Parham. All Rights Reserved.

Introduction

FpcRef is a panelization cosmetic preprocessor implemented as a Win2K/XP console (command line) application. It replaces component reference designation strings on both sides of the board with text. This is done by adding text strings that duplicates each reference designation and hiding the real references by setting both the height and stroke width to zero. The original reference designations values, though invisible, are otherwise unchanged.

Each .fpc file processed results in a new file being created with "_ref" appended to the file name, the original file is unchanged. For example *demo.fpc* produces *demo_ref.fpc*. Reference designation text should only consist of normally printable characters (code 0x20 - 0x7e). Characters outside this range may cause text location error on bottom side test.

```
        Ex:

    D:\Code\FpcRef\RefTest>fpcref "text test" demo          ← file test text.fpc is quoted

            FpcRef - A FreePCB Post Layout RefDes Converter
                    Ver 1.10R (c)2007-08 Bruce Parham

      File "text test.fpc" loaded, 758 lines read.
        File "text test_ref.fpc" created
        34 Component RefRes strings hidden by "Height/Stroke" method
        in "text test_ref.fpc"

      File "demo.fpc" loaded, 1168 lines read.
        File "demo_ref.fpc" created
        25 Component RefRes strings hidden by "Height/Stroke" method
        in "demo_ref.fpc"

      Done.
        2 files loaded.
        2 files created

      Elapsed time: 0.113 Sec
```

Installation and Setup

FpcRef is distributed as a set of files in a zip archive that includes the executable, the user guide and a selection of example files.

To install *FpcRef*, extract the file **FpcRef.exe** to any handy directory such as C:\Program Files\ FpcRef\ or C:\Program Files\FreePCB\bin\.

If the path to **FpcRef.exe** is not already included in the environment **PATH** variable, it should be added.

For those unfamiliar with setting environment variables, assume for example that **FpcRef.exe** is installed in the C:\Program Files\FreePCB\bin directory:

- Open the **System Properties** form by

  - Right clicking on **My Computer** and clicking on **Properties**.
  or
  - Open the **Control Panel**.
  - Open the **System** tool.

- Click the **Advanced** tab and open the **Environment Variables**.

- Scroll the variable list down to and highlight the **path=** entry and click **EDIT**.

- Click in the value field to clear the highlight.

- Scroll to the end of the field and add **;C:\Program Files\FreePCB\bin** to the existing value.

- Click **OK** to exit the edit window.

- Click **OK** to save and exit the Environment Variables window.

- Click **OK** to exit System Properties.

Operation

FpcRef is a command line application. To use it, open a command shell (Start→Run→Cmd) and change the working directory to where the FreePCB file(s) are located.

Command Line Syntax

The command line syntax was designed for simple yet flexible operation. The full syntax consists of the program name followed by optional switches and one or more file names:

```
C:> fpcref [switches] [path1]name1[.ext1] [[path2]name2[.ext2]] ...
```

The number of files that may be processed is limited only by the DOS command line length. If the file name includes an extension, as determined by the presence of a period character, the name will be used as entered; if no extension is found, the extension **.FPC** will be append. If the name includes any spaces, it must be enclosed within double quotes ("). The optional command line switches are not case sensitive.

Available Switches

-D	Enable *__Debug__* mode.
-H or -?	Display a short switch and usage *__Help__* summary and quit.
-V	Enable the *__Visibility__* method for hiding RefDes strings.

-D : Debug Mode

Change operation to debug mode for text position verification. In this mode, the new text is placed on the *inner_1* and *inner_2* signal layers instead of the top and bottom silkscreens and the original reference designation text remains visible but has its stroke width set to zero.
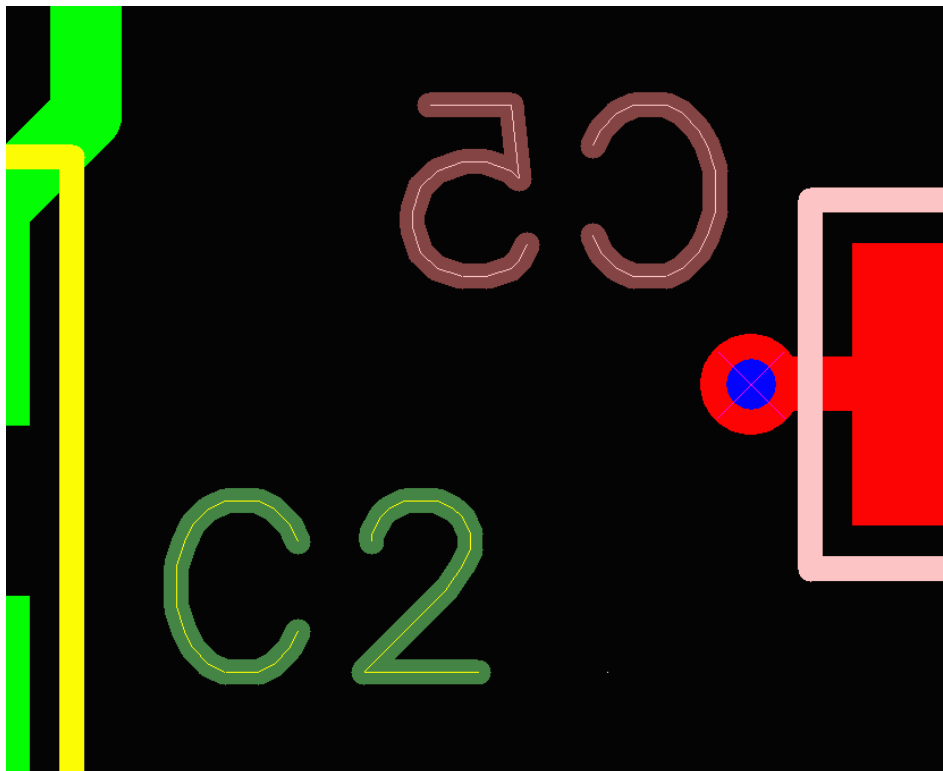
Debug mode will only function with files that have four or more signal layer. The *inner_1* and *inner_2* layers must both be present, if not, debug mode is disabled for that file.

-H or -? : Help

Display a short help screen and terminate.

-V : Enable the Visibility Method

By default, *FpcRef* hides the existing reference designation strings by setting both the height and stroke width to zero. This is referred to as the *Height/Stroke* method. In newer versions of *FreePCB*, reference designations have a visibility control flag that can be used to hide the text. With the *Visibility* method enabled, files created by these newer *FreePCB* versions will use the reference designation visibility flag to hide the existing designations. Sad note: as of version 1.352, *FreePCB* fails to honor the visibility flags, during **Paste Group From File** operation, and makes all imported designation visible. Until a future version of *FreePCB* fixes this situation, continue to use the default *Height/Stroke* method for proper operation.

Panelization

FreePCB has, for some time, had the capability to easily panelize a single layout with its Gerber generator. But combining two or more layouts for panelization, with any layout tool, including ***FreePCB***, presents problems with part reference and net name duplication. This is normally solved by reassigning the part references and by either merging or renaming conflicting net.

For panelizing, we don't need to worry about net names but we do care about the part reference changes. ***FpcRef*** solves this dilemma by creating a copy of the layout where the part references are duplicated as silkscreen text objects and hiding the real references. The modified layout can then be combined with other layouts and any reference changes will not be visible either in the layout or the resulting Gerber files.

Tutorial

As an exercise, we'll walk through the steps necessary to create a simple combined layout.

The starting layout does not need to be modified but any imported layouts do. So the first step is to select which layouts *will* be imported and create modified versions with ***FpcRef***. In this exercise, we will only be importing one layout: *motor_1.fpc* so use ***FpcRef*** to create *motor_1_ref.fpc*.

```
D:\PCBs\FpcRef Demo> fpcref motor_1

  FpcRef - A FreePCB Post Layout RefDes Converter
          Ver 1.10D (c)2007-08 Bruce Parham

 File "motor_1.fpc" loaded, 1312 lines read.

   File "motor_1_ref.fpc" created,

   25 Component RefRes strings hidden by "Height/Stroke" method
   in "motor_1_ref.fpc".

Done.

  1 file loaded.

  1 file created.

Elapsed time: 0.062 Sec
```
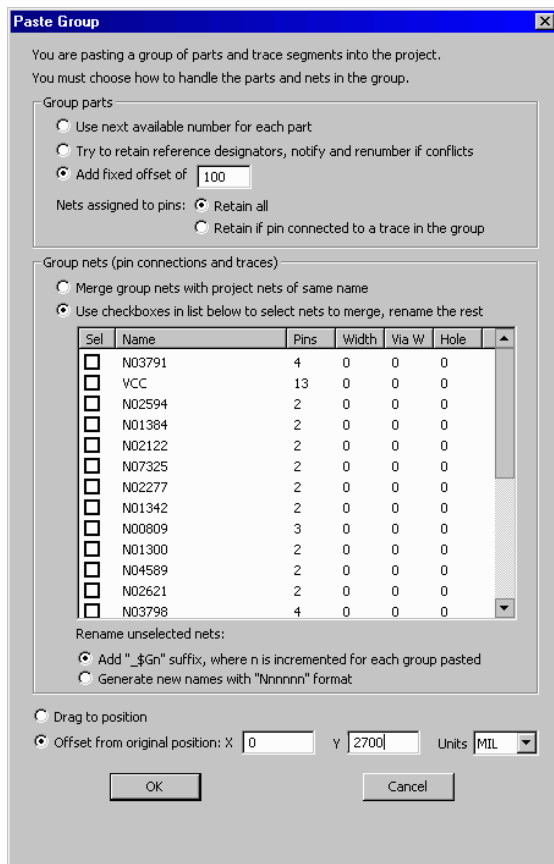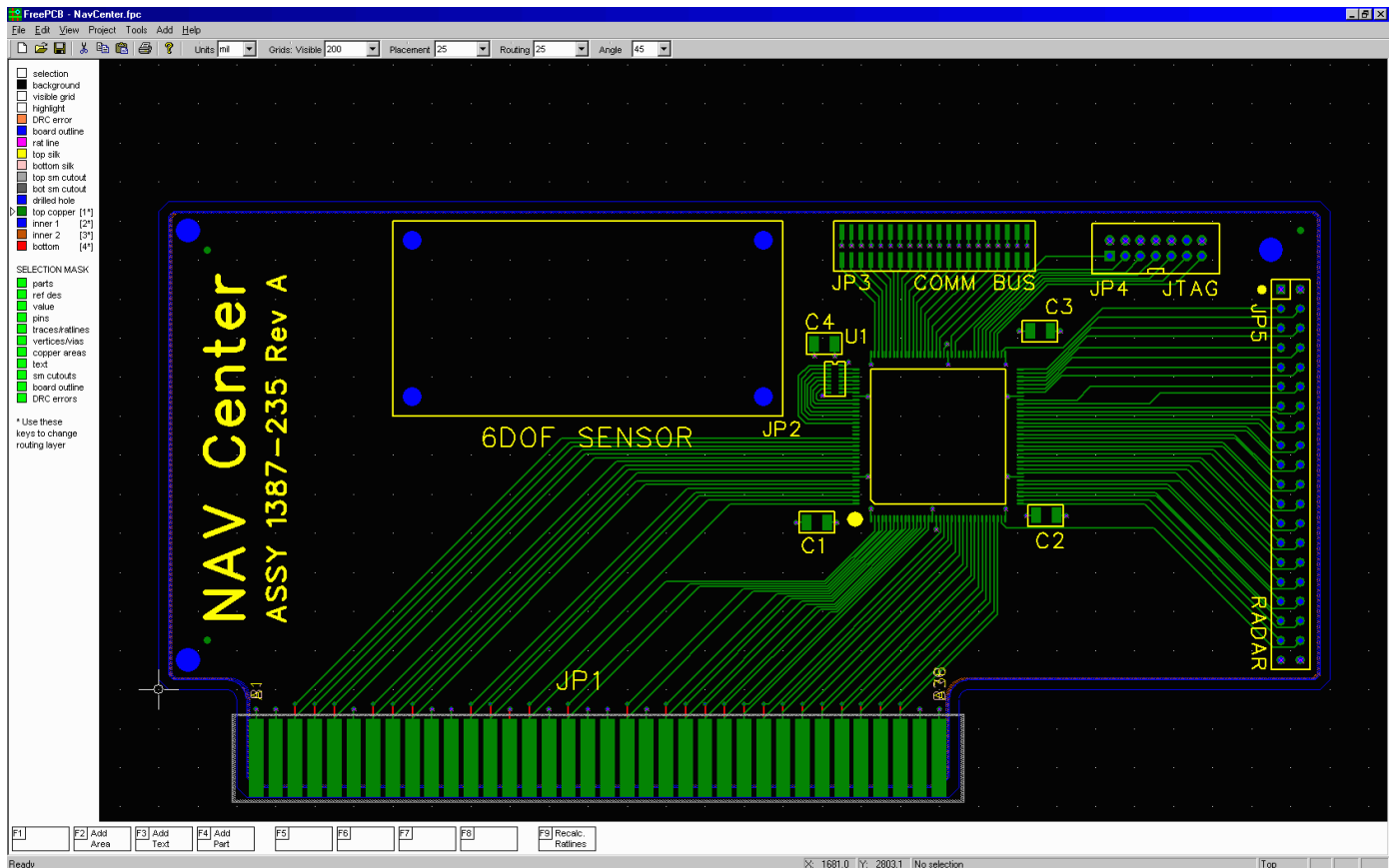
Next, open the starting layout, *NavCenter.fpc* in this case, in ***FreePCB***.


Once the layout is opened. click on **Edit→Paste Group From File...** and select the file *motor_1_ref.fpc* to open the **Paste Group** form.

In the **Group Parts** section, click on <u>Add fixed offset</u> and enter a large offset like 100.

In the **Group Nets** section, make sure <u>Use Checkboxes...</u> and <u>Add "_Gn" suffix</u> are selected and that no nets are checked or select the <u>Generate new names</u> option. (This step is not strictly necessary but, if nets are allowed to merge, the layout can become cluttered with ratlines.)

Finally, select <u>Offset from original position</u>, enter a Y offset of 2700 mils and click ***OK***. (The offset function refers the position of the origin of the imported layout relative to the origin of the current layout.)

Click outside of the layout to unselect the newly imported items.
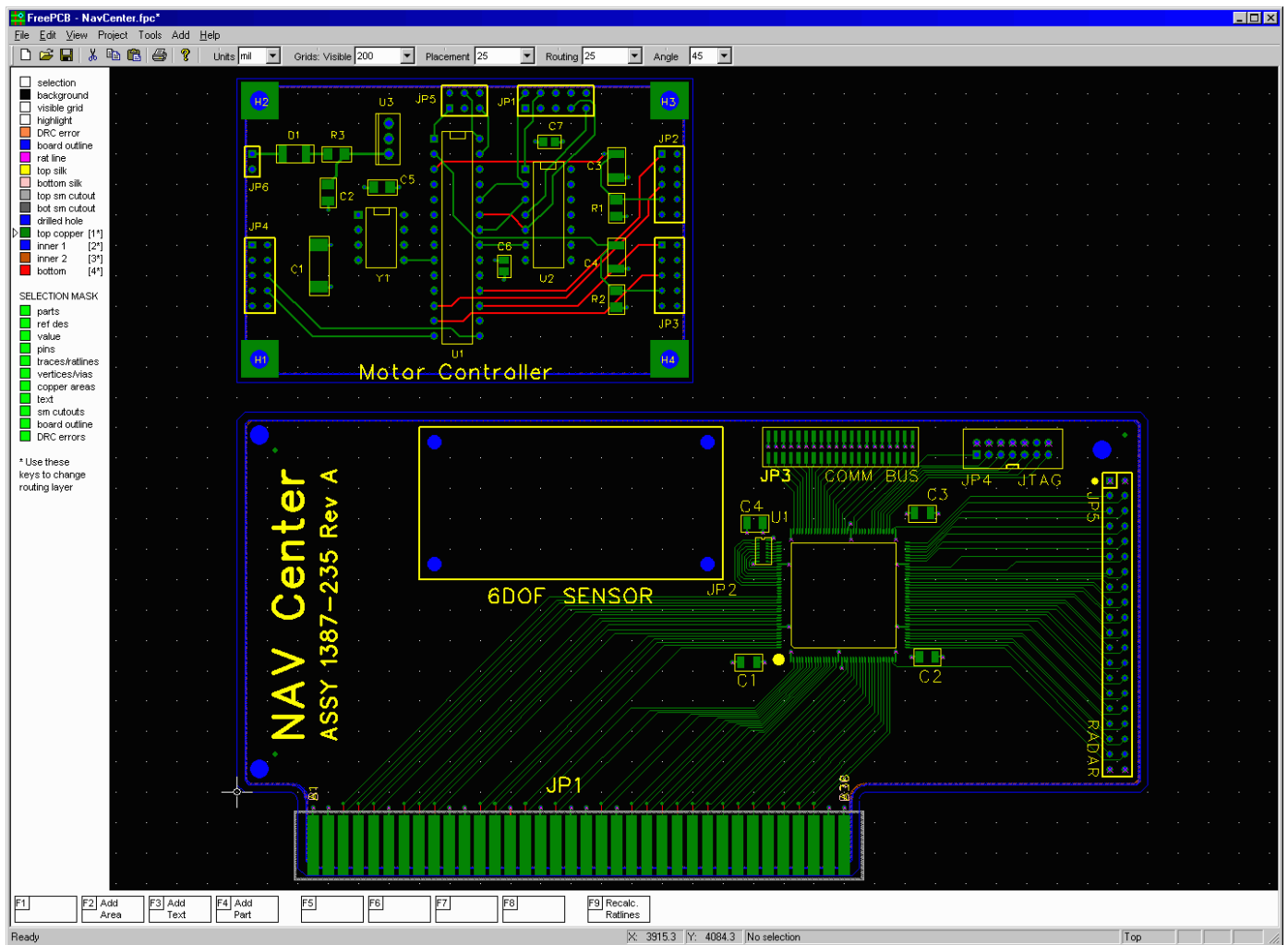
The layout as it appears after the first import operation.

To finish the layout, repeat the import process, as outlined above, but with a parts offset of 200 and an XY offset of 3200 by 2700 mils.
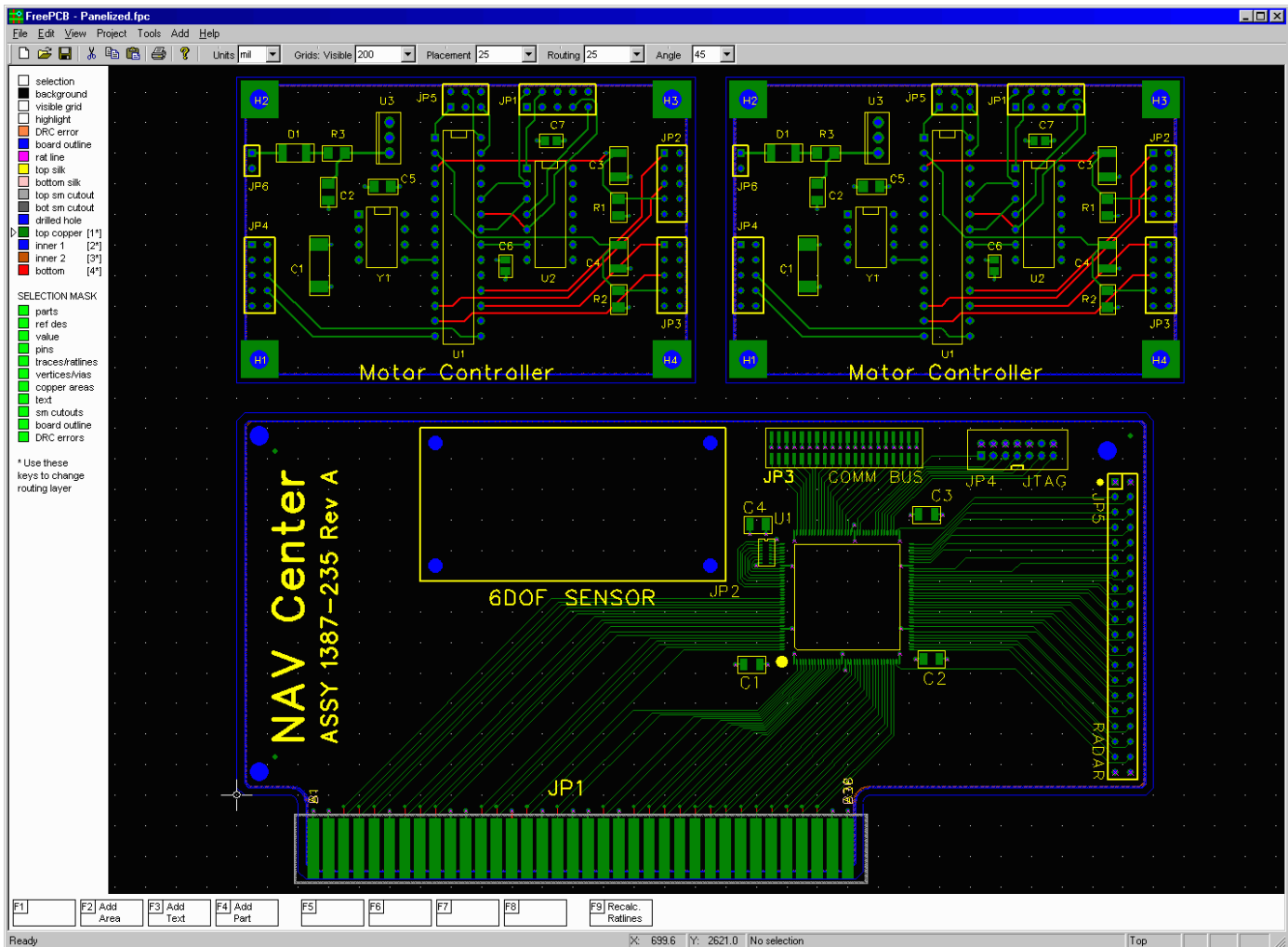
Save the finished layout to a new file using **File→Save As** and the combined layout is done.

Revision History

```
Ver      Date      Comment
-----    ---------  --------------------------------------------------------
1.000    15 Aug 07  Initial release to FreePCB forum.

1.01     16 Aug 07  Fixed bug in debug mode on boards with less than 4 layers.
                    Changed version number to 2 decimal place value.

1.02     19 Nov 07  Updated to FPC V1.339.

1.10      3 Aug 08  Updated to FPC V1.344. Added millisecond resolution elapsed time
                    indicator. Made "Height/Stroke" visibility method the default.
                    Added real User Guide.
```