==

Application: kicad
Version: 6.0.0-rc1-unknown-**a5c892d**~84~ubuntu14.04.1, release build
[GS note: commit a5c892d9cd36829b5288a29f1682f22a4c78830e was built 18 Oct. 2018 at 15:15:22]
Libraries:
   wxWidgets 3.0.2
   libcurl/7.35.0 OpenSSL/1.0.1f zlib/1.2.8 libidn/1.28 librtmp/2.3
Platform: Linux 4.4.0-112-generic x86_64, 64 bit, Little endian, wxGTK
Build Info:
   wxWidgets: 3.0.2 (wchar_t,wx containers,compatible with 2.8) GTK+ 2.24
   Boost: 1.54.0
   OpenCASCADE Community Edition: 6.8.0
   Curl: 7.35.0
   Compiler: GCC 4.8.4 with C++ ABI 1002

Build settings:
   USE_WX_GRAPHICS_CONTEXT=OFF
   USE_WX_OVERLAY=OFF
   KICAD_SCRIPTING=ON
   KICAD_SCRIPTING_MODULES=ON
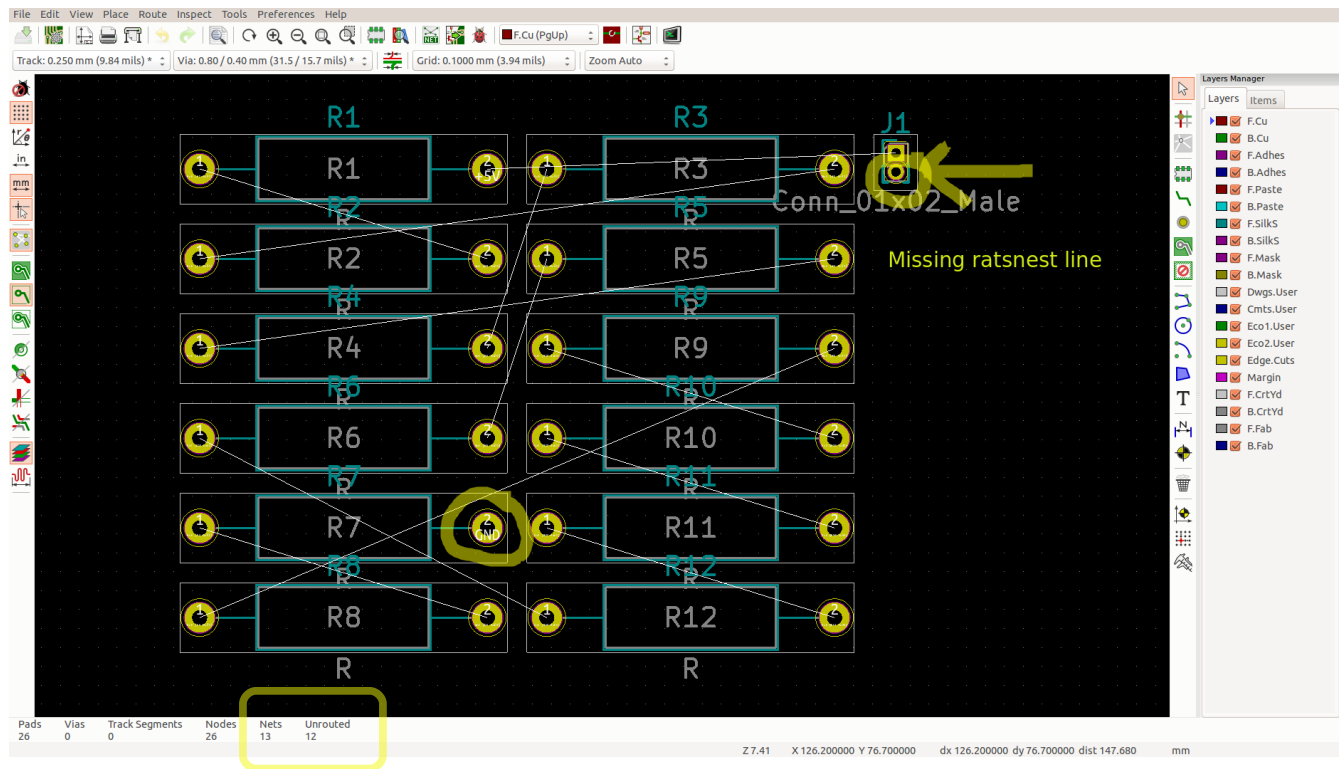   KICAD_SCRIPTING_WXPYTHON=ON
   KICAD_SCRIPTING_ACTION_MENU=ON
   BUILD_GITHUB_PLUGIN=ON
   KICAD_USE_OCE=ON
   KICAD_USE_OCC=OFF
   KICAD_SPICE=ON

commit 47f0f52b28471a8be6e5fc28bd8407dbf9090f26

==

Application: kicad
Version: 5.0.0-fee4fd1~66~ubuntu14.04.1, release build
Libraries:f
    wxWidgets 3.0.2
    libcurl/7.35.0 OpenSSL/1.0.1f zlib/1.2.8 libidn/1.28 librtmp/2.3
Platform: Linux 4.4.0-112-generic x86_64, 64 bit, Little endian, wxGTK
Build Info:
    wxWidgets: 3.0.2 (wchar_t,wx containers,compatible with 2.8) GTK+ 2.24
    Boost: 1.54.0
    OpenCASCADE Community Edition: 6.8.0
    Curl: 7.35.0
    Compiler: GCC 4.8.4 with C++ ABI 1002

Build settings:
    USE_WX_GRAPHICS_CONTEXT=OFF
    USE_WX_OVERLAY=OFF
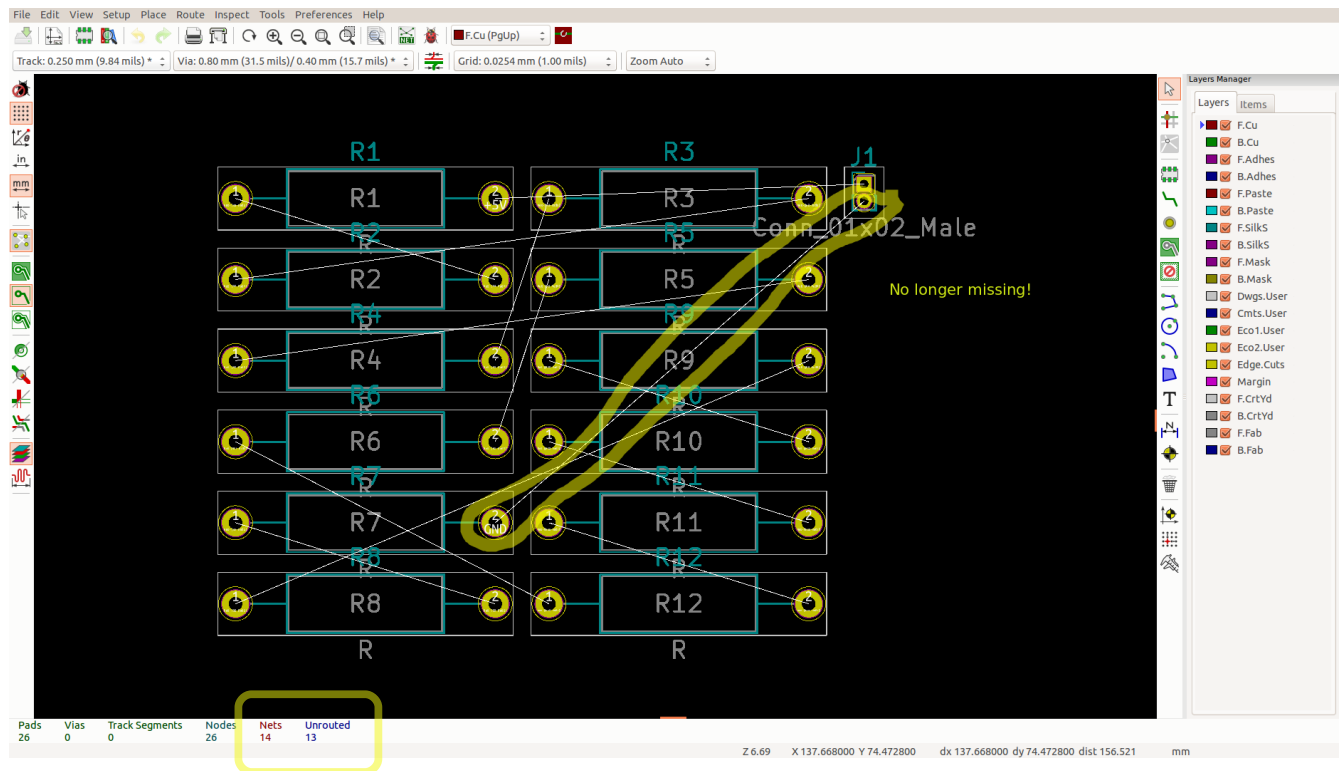   KICAD_SCRIPTING=ON
    KICAD_SCRIPTING_MODULES=ON
    KICAD_SCRIPTING_WXPYTHON=OFF
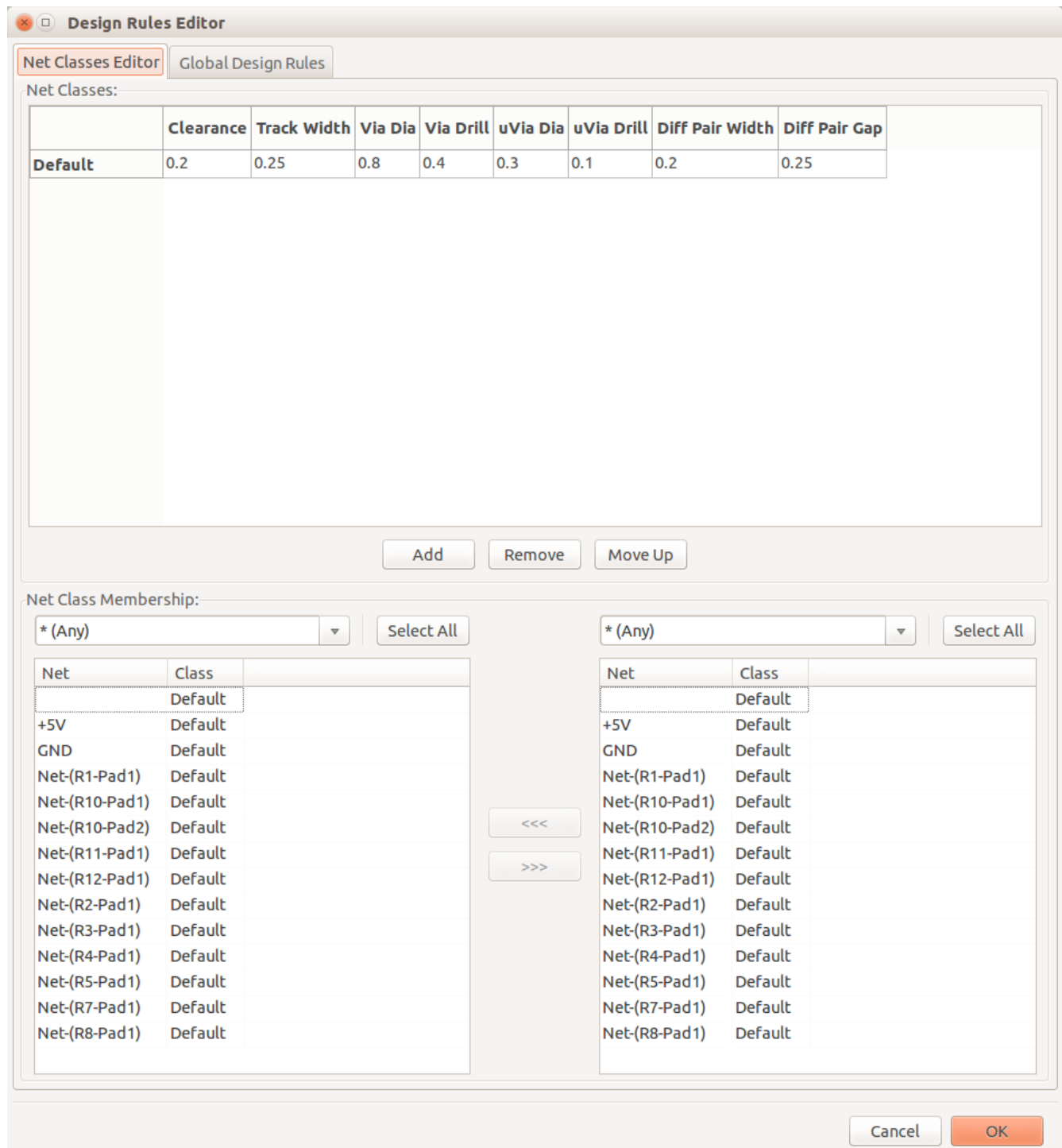   KICAD_SCRIPTING_ACTION_MENU=ON
    BUILD_GITHUB_PLUGIN=ON
    KICAD_USE_OCE=ON

KICAD_USE_OCC=OFF
KICAD_SPICE=ON

Nets:

Design Rules Editor

Net Classes Editor | Global Design Rules

Net Classes:

	Clearance	Track Width	Via Dia	Via Drill	uVia Dia	uVia Drill	Diff Pair Width	Diff Pair Gap
Default	0.2	0.25	0.8	0.4	0.3	0.1	0.2	0.25

Add | Remove | Move Up

Net Class Membership:

* (Any) ▼ | Select All | * (Any) ▼ | Select All

Net	Class		Net	Class
	Default			Default
+5V	Default		+5V	Default
GND	Default		GND	Default
Net-(R1-Pad1)	Default		Net-(R1-Pad1)	Default
Net-(R10-Pad1)	Default		Net-(R10-Pad1)	Default
Net-(R10-Pad2)	Default	<<<	Net-(R10-Pad2)	Default
Net-(R11-Pad1)	Default		Net-(R11-Pad1)	Default
Net-(R12-Pad1)	Default	>>>	Net-(R12-Pad1)	Default
Net-(R2-Pad1)	Default		Net-(R2-Pad1)	Default
Net-(R3-Pad1)	Default		Net-(R3-Pad1)	Default
Net-(R4-Pad1)	Default		Net-(R4-Pad1)	Default
Net-(R5-Pad1)	Default		Net-(R5-Pad1)	Default
Net-(R7-Pad1)	Default		Net-(R7-Pad1)	Default
Net-(R8-Pad1)	Default		Net-(R8-Pad1)	Default

Cancel | OK

===

Application: kicad
Version: 6.0.0-rc1-unknown-**82f657**e~84~ubuntu14.04.1, release build
[GS note: 82f657ed036fd5fc3609c1baaf13f2449d77baaa was built 22 Nov 2018 at 01:09:40]
Libraries:

wxWidgets 3.0.2
libcurl/7.35.0 OpenSSL/1.0.1f zlib/1.2.8 libidn/1.28 librtmp/2.3
Platform: Linux 4.4.0-112-generic x86_64, 64 bit, Little endian, wxGTK
Build Info:
wxWidgets: 3.0.2 (wchar_t,wx containers,compatible with 2.8) GTK+ 2.24
Boost: 1.54.0
OpenCASCADE Community Edition: 6.8.0
Curl: 7.35.0
Compiler: GCC 4.8.4 with C++ ABI 1002

Build settings:
USE_WX_GRAPHICS_CONTEXT=OFF
USE_WX_OVERLAY=OFF
KICAD_SCRIPTING=ON
KICAD_SCRIPTING_MODULES=ON
KICAD_SCRIPTING_PYTHON3=OFF
KICAD_SCRIPTING_WXPYTHON=ON
KICAD_SCRIPTING_WXPYTHON_PHOENIX=OFF
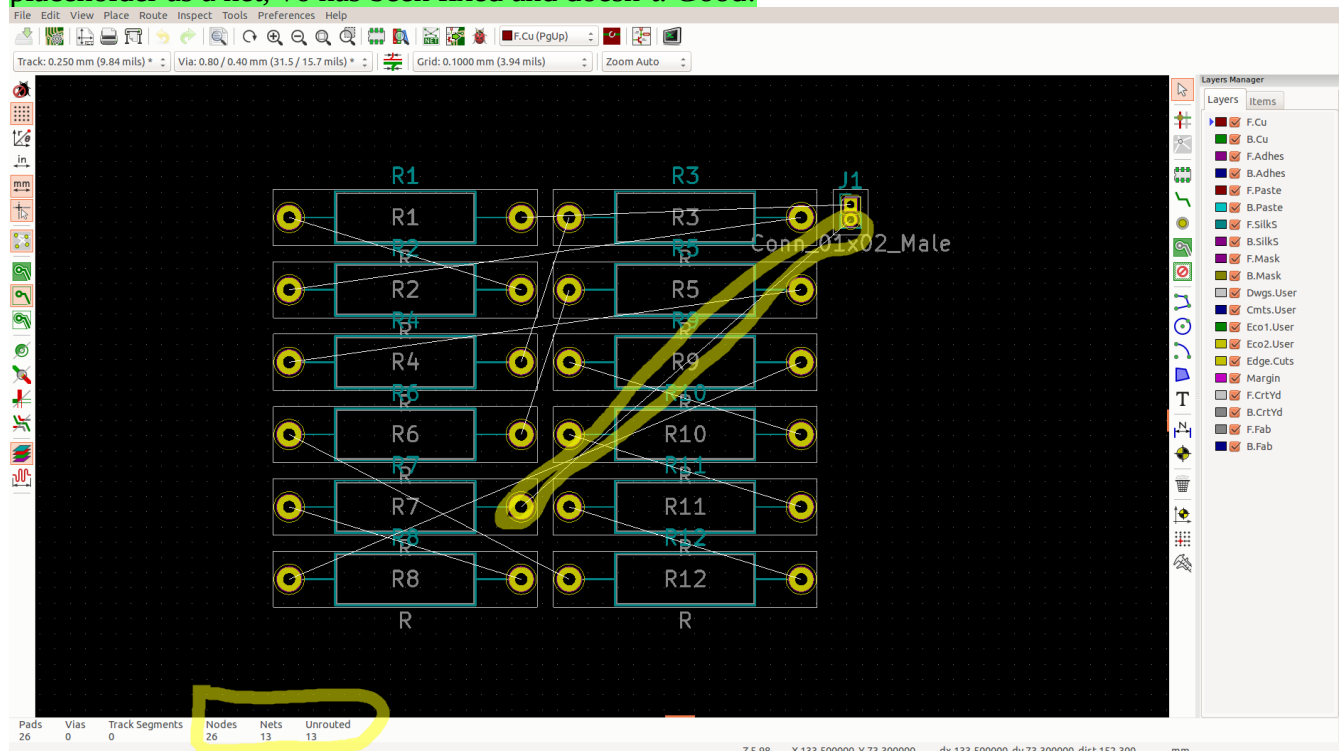KICAD_SCRIPTING_ACTION_MENU=ON
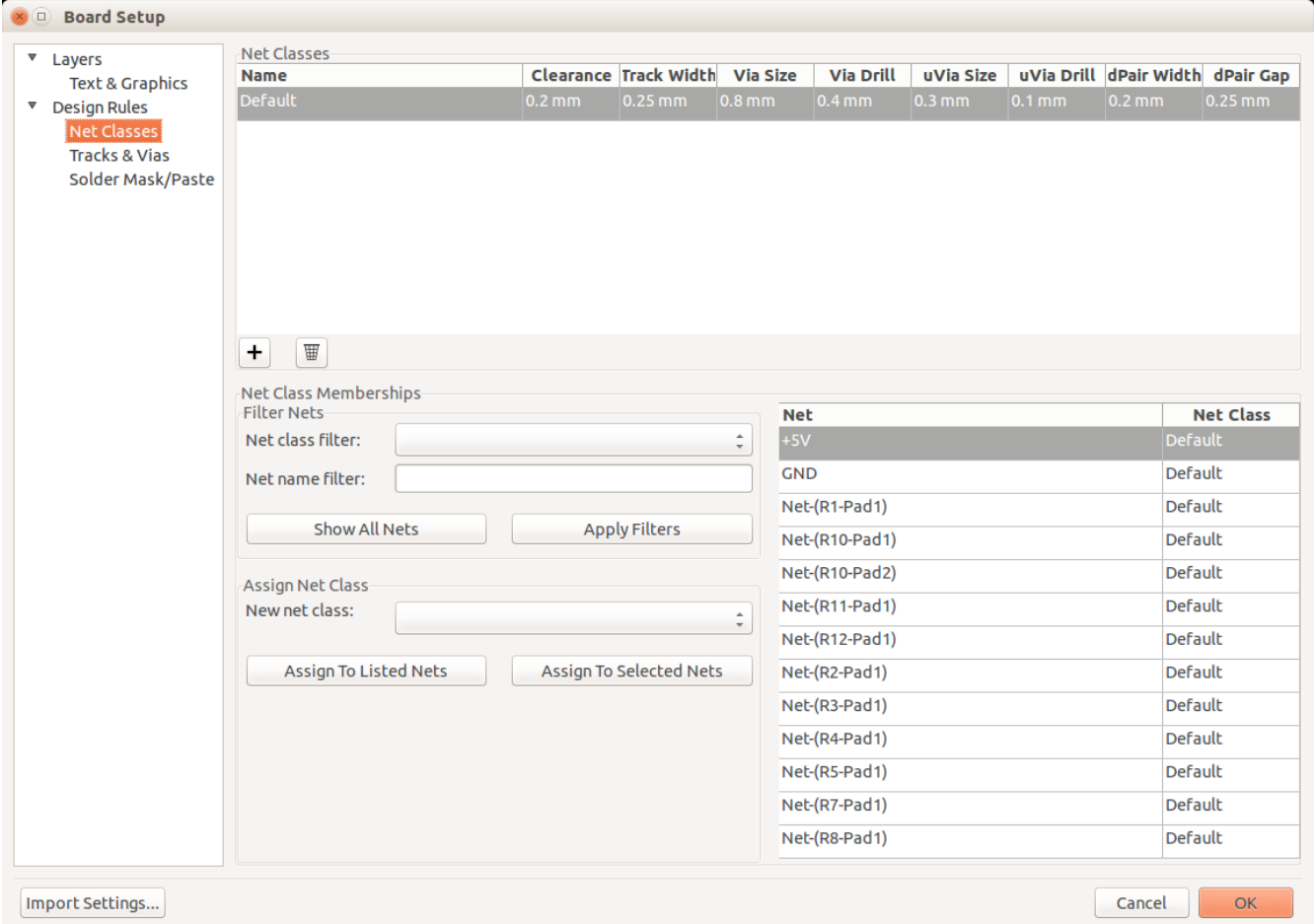BUILD_GITHUB_PLUGIN=ON
KICAD_USE_OCE=ON
KICAD_USE_OCC=OFF
KICAD_SPICE=ON

Looks good! But...it has only 13 nets vs 14, what's the difference? Ans: v5 counted an empty placeholder as a net, v6 has been fixed and doesn't. Good!

Here's the nets:

===

===

===

===

===

===

===

==

==

==

- D
- d
- d
- dd
- d
- d
- dd
- d
- d
- d
- d
- d
- d
- d