
revno: 4022
committer: jean-pierre charras <jp.charras@wanadoo.fr>
branch nick: stable
timestamp: Sun 2013-07-07 12:11:28 +0200
message:
 Eeschema: Bug fixes.

revno: 4021
committer: jean-pierre charras <jp.charras@wanadoo.fr>
branch nick: stable
timestamp: Tue 2013-06-11 16:27:46 +0200
message:
 Eeschema: fix Bug #1189893 (ps plot file opened twice)

revno: 4020
committer: jean-pierre charras <jp.charras@wanadoo.fr>
branch nick: stable
timestamp: Thu 2013-06-06 20:06:57 +0200
message:
 Fix eol on repos.
 Fix some minor issues from testing branch

revno: 4019
committer: jean-pierre charras <jp.charras@wanadoo.fr>
branch nick: stable
timestamp: Fri 2013-05-31 20:23:40 +0200
message:
 Fix a minor issue in my previous commit.

revno: 4018
committer: jean-pierre charras <jp.charras@wanadoo.fr>
branch nick: stable
timestamp: Thu 2013-05-30 21:33:31 +0200
message:
 Bug fixed and some enhancements, from testing branch.
 Pcbnew: Fix Bug #1185556. fix issue about Solder Paste Ratio value which was accepting only one digit in mantissa in 3 dialogs (now 6 digits in all dialogs).
 fix potential issue in .kicad_pcb file creation, in some places where a %g or %.16g format was used: at least under Mingw/gcc4.7.2, the floating number was written using scientific notation, not accepted by the S-expr reader.

revno: 4017
committer: jean-pierre charras <jp.charras@wanadoo.fr>
branch nick: stable
timestamp: Sat 2013-05-18 19:01:04 +0200
message:
 Fix bug in gerbview, and minor other fixes.

revno: 4016
committer: jean-pierre charras <jp.charras@wanadoo.fr>
branch nick: stable
timestamp: Thu 2013-05-16 13:47:54 +0200
message:
 Fix some minor issues, from testing branch.

revno: 4015
committer: jean-pierre charras <jp.charras@wanadoo.fr>
branch nick: stable
timestamp: Wed 2013-05-01 18:54:20 +0200
message:
 Pcbnew: minor fix from testing branch.

revno: 4014
committer: jean-pierre charras <jp.charras@wanadoo.fr>
branch nick: stable
timestamp: Fri 2013-04-26 17:27:00 +0200
message:
 Pcbnew: fix read .kicad_pcb file issue

revno: 4013
committer: jean-pierre charras <jp.charras@wanadoo.fr>

```
branch nick: stable
timestamp: Mon 2013-04-22 21:32:55 +0200
message:
  Update Kicad manager. Pcbnew: fix curious behavior of hotkeys W and CtrlW
-----
revno: 4012
fixes bug(s): https://launchpad.net/bugs/1171090 https://launchpad.net/bugs/1170973
committer: Wayne Stambaugh <stambaughw@verizon.net>
branch nick: stable
timestamp: Sun 2013-04-21 18:00:38 -0400
message:
  Fix pcb_calculator build bug and KiCad crash bug. (fixes lp:1170973 and 1171090)
-----
revno: 4011
committer: jean-pierre charras <jp.charras@wanadoo.fr>
branch nick: stable
timestamp: Fri 2013-04-19 18:50:04 +0200
message:
  Bugs fixes and remove outdated files.
-----
revno: 4010
committer: jean-pierre charras <jp.charras@wanadoo.fr>
branch nick: stable
timestamp: Wed 2013-04-10 21:21:42 +0200
message:
  Pcbnew: fix Bug #1167364
  pcb_calculator: fix bug Bug #1167345
-----
revno: 4009
committer: jean-pierre charras <jp.charras@wanadoo.fr>
branch nick: stable
timestamp: Wed 2013-04-03 18:28:45 +0200
message:
  Pcbnew: Fix bug in netlist reader (Kicad format, when a component library is named components.lib).
-----
revno: 4008
committer: jean-pierre charras <jp.charras@wanadoo.fr>
branch nick: stable
timestamp: Sun 2013-03-31 14:13:17 +0200
message:
  Minor cosmetic fixes (mainly on Linux) in dialogs.
-----
revno: 4007 [merge]
committer: jean-pierre charras <jp.charras@wanadoo.fr>
branch nick: stable
timestamp: Sat 2013-03-30 19:59:35 +0100
message:
  Bug fixes
-----
revno: 4006
fixes bug(s): https://launchpad.net/bugs/1161843
committer: Wayne Stambaugh <stambaughw@verizon.net>
branch nick: stable
timestamp: Sat 2013-03-30 14:08:46 -0400
message:
  Fix Eeschema field position bug in edit component in schematic dialog.
-----
revno: 4005 [merge]
committer: jean-pierre charras <jp.charras@wanadoo.fr>
branch nick: stable
timestamp: Mon 2013-03-25 21:07:47 +0100
message:
  Pcbnew: fix issue in zone keepout dialog: layers not displayed on Linux, and minor changes
-----
revno: 4004 [merge]
committer: jean-pierre charras <jp.charras@wanadoo.fr>
branch nick: stable
timestamp: Tue 2013-03-19 09:51:39 +0100
message:
  Pcbnew: design rules dialog: fix incorrect size of track sizes list grid
-----
revno: 4003
```

```
committer: jean-pierre charras <jp.charras@wanadoo.fr>
branch nick: stable_2013-03-15_BZR4003
timestamp: Fri 2013-03-15 08:44:29 +0100
message:
  3D viewer: fix very minor issues (better rendering of 3D view).
-----
revno: 4002
committer: jean-pierre charras <jp.charras@wanadoo.fr>
branch nick: stable_2013-03-13_BZR4001
timestamp: Thu 2013-03-14 20:15:29 +0100
message:
  3D viewer: fix very minor issues (better rendering of 3D view).
-----
revno: 4001
committer: jean-pierre charras <jp.charras@wanadoo.fr>
branch nick: testing
timestamp: Wed 2013-03-13 20:06:46 +0100
message:
  Last (minor) fixes for the stable version
-----
revno: 4000
committer: Dick Hollenbeck <dick@softplc.com>
branch nick: testing
timestamp: Wed 2013-03-13 11:38:54 -0500
message:
  EAGLE_PLUGIN cu layer map fix broke footprint library loading
-----
revno: 3999
committer: Dick Hollenbeck <dick@softplc.com>
branch nick: testing
timestamp: Wed 2013-03-13 11:17:16 -0500
message:
  EAGLE_PLUGIN cu layer map fix
-----
revno: 3998
committer: jean-pierre charras <jp.charras@wanadoo.fr>
branch nick: kicad
timestamp: Tue 2013-03-12 17:37:41 +0100
message:
  Pcbnew: fix a bug in autoplac.cpp
-----
revno: 3997
committer: jean-pierre charras <jp.charras@wanadoo.fr>
branch nick: kicad
timestamp: Tue 2013-03-12 16:17:44 +0100
message:
  Pcbnew: Fix many minor issues in print dialog, mainly in modedit.
-----
revno: 3996
committer: Miguel Angel Ajo <miguelangel@nbee.es>
branch nick: kicad
timestamp: Tue 2013-03-12 01:37:45 +0100
message:
  Reverted back the ( i ) parameters for python, applied the PyGILState_Ensure from Dick into his
  PyLOCK class
-----
revno: 3995
committer: jean-pierre charras <jp.charras@wanadoo.fr>
branch nick: kicad
timestamp: Mon 2013-03-11 20:30:58 +0100
message:
  Pcbnew: print dialog: code cleanup and bug fixes (added some fixes from Sergey A. Borshch)
-----
revno: 3994
committer: Dick Hollenbeck <dick@softplc.com>
branch nick: testing
timestamp: Mon 2013-03-11 03:09:48 -0500
message:
  improvements to python's GIL acquisition and release, but not done yet, since I think the GIL needs
  to be acquired even when not involving wxPython.
-----
revno: 3993
```

committer: jean-pierre charras <jp.charras@wanadoo.fr>

branch nick: kicad

timestamp: Sun 2013-03-10 20:55:21 +0100

message:

Kicad manager: fix bug: .kicad_pcb files not saved in zip file

revno: 3992

committer: jean-pierre charras <jp.charras@wanadoo.fr>

branch nick: kicad

timestamp: Sun 2013-03-10 14:59:47 +0100

message:

Pcbnew: fix issues: Ensure pads type connector have a drill size= 0, like SMD pads. Ensure keepout zones have no net.

revno: 3991

committer: Miguel Angel Ajo <miguelangel@nbee.es>

branch nick: kicad

timestamp: Sun 2013-03-10 02:35:24 +0100

message:

Project templates now can find the system templates from KICAD environment variable

revno: 3990

committer: jean-pierre charras <jp.charras@wanadoo.fr>

branch nick: kicad

timestamp: Sat 2013-03-09 20:36:31 +0100

message:

Pcbnew: Partial merge from pcard2kicad (from Alexander Lunev). Fix an old bug related to pads with offset: sometimes the connection to a zone was not detected.

revno: 3989

committer: jean-pierre charras <jp.charras@wanadoo.fr>

branch nick: kicad

timestamp: Fri 2013-03-08 08:29:30 +0100

message:

Pcbnew: fix bug in export vrml (bad Z position of 3D shapes). Very minor code cleaning.

revno: 3988

committer: Wayne Stambaugh <stambaughw@verizon.net>

branch nick: testing

timestamp: Thu 2013-03-07 12:12:59 -0500

message:

Documentation and coding policy fixes.

- * Fix all Doxygen warnings except polygon files.
- * Add footprint library table tasks to TODO.txt.
- * Add definition to drag.h to prevent nesting.
- * Coding policy fixes.

revno: 3987

author: Baranovskiy Konstantin <baranovskiykonstantin@gmail.com>

committer: Dick Hollenbeck <dick@softplc.com>

branch nick: testing

timestamp: Thu 2013-03-07 08:15:35 -0600

message:

Width factor for DXF plotting text used an inverted aspect ratio.

revno: 3986

committer: jean-pierre charras <jp.charras@wanadoo.fr>

branch nick: kicad

timestamp: Wed 2013-03-06 20:32:59 +0100

message:

Gerbview: Minor fixes

revno: 3985

committer: jean-pierre charras <jp.charras@wanadoo.fr>

branch nick: kicad

timestamp: Wed 2013-03-06 10:04:27 +0100

message:

Pcbnew: fix Bug #1148785 (pcbnew crashes when using only one layer in autorouter) , and a minor bug in dialog cleanup.

revno: 3984 [merge]

committer: jean-pierre charras <jp.charras@wanadoo.fr>

branch nick: kicad

timestamp: Mon 2013-03-04 20:59:12 +0100

message:

Pcbnew, module edit dialogs: fix some "cosmetic" issues.

revno: 3983

fixes bug(s): <https://launchpad.net/bugs/1143465>

committer: Wayne Stambaugh <stambaughw@verizon.net>

branch nick: testing

timestamp: Mon 2013-03-04 11:04:53 -0500

message:

Minor Pcbnew dialog spacing and consistency fixes. (fixes lp:1143465)

- * Fix spacing and consistency issues and debug warning due to improper flex grid sizer in module editor module properties dialog.

- * Fix spacing and consistency issues in Pcbnew module properties dialog.

- * Fix spelling error in DRC dialog message box text.

revno: 3982

committer: jean-pierre charras <jp.charras@wanadoo.fr>

branch nick: kicad

timestamp: Mon 2013-03-04 13:08:35 +0100

message:

Pcbnew: fix strange things in module edit dialogs.

revno: 3981

committer: Andrey Fedorushkov <andrf@mail.ru>

branch nick: kicad-lp

timestamp: Sat 2013-03-02 20:32:05 +0400

message:

pcbnew: fix build in linux gcc-4.5.2

revno: 3980

committer: jean-pierre charras <jp.charras@wanadoo.fr>

branch nick: kicad

timestamp: Fri 2013-03-01 20:59:29 +0100

message:

Eeschema: fix broken mirroring for texts and labels in block mirror X and Y (this fixes Bug #1135505)

Pcbnew: add in module popup menu an option to edit the footprint in Modedit (like in Eeschema for components)

minor icons enhancements.

revno: 3979

committer: Wayne Stambaugh <stambaughw@verizon.net>

branch nick: testing

timestamp: Fri 2013-03-01 08:32:55 -0500

message:

Update coding policy to remove support for MSVC.