



# kicad



# kicad

## GUI Translation HOWTO

**December 16, 2015**

## Contents

<b>1</b>	<b>Needed files and tools</b>	<b>2</b>
1.1	Download PoEdit . . . . .	2
1.2	Downloading KiCad sources . . . . .	2
1.3	Download existing translations and docs . . . . .	2
<b>2</b>	<b>Find sentences to translate</b>	<b>2</b>
<b>3</b>	<b>KiCad tree for translations</b>	<b>2</b>
3.1	Dictionary tree . . . . .	2
3.2	Search path . . . . .	3
3.3	Files . . . . .	4
<b>4</b>	<b>Using poedit</b>	<b>4</b>
4.1	Installation . . . . .	4
4.2	KiCad preparation . . . . .	4
4.3	Poedit Configuration . . . . .	4
4.4	Project Configuration . . . . .	5
4.5	Path and files Configuration . . . . .	6
4.6	Keyword Configuration . . . . .	6
4.7	Save the project . . . . .	6
<b>5</b>	<b>Create or edit a dictionary</b>	<b>6</b>
<b>6</b>	<b>Adding a new language entry in KiCad source code (devs only)</b>	<b>7</b>
6.1	Steps . . . . .	8
6.1.1	Adding a new id in include/id.h. . . . .	8
6.1.2	Adding a new icon (aesthetic purpose only) . . . . .	8
6.1.3	Editing bitmaps_png/CMakeLists.txt . . . . .	9
6.1.4	Editing include/bitmaps.h . . . . .	9
6.1.5	Editing common/edaappl.cpp . . . . .	9
6.1.6	Recompiling . . . . .	10

---

*Reference manual***Copyright**

This document is Copyright © 2010-2015 by its contributors as listed below. You may distribute it and/or modify it under the terms of either the GNU General Public License (<http://www.gnu.org/licenses/gpl.html>), version 3 or later, or the Creative Commons Attribution License (<http://creativecommons.org/licenses/by/3.0/>), version 3.0 or later.

All trademarks within this guide belong to their legitimate owners.

**Contributors**

Jean-Pierre Charras, Fabrizio Tappero, Wayne Stambaugh.

**Feedback**

Please direct any bug reports, suggestions or new versions to here:

- About KiCad document: <https://github.com/KiCad/kicad-doc/issues>
- About KiCad software: <https://bugs.launchpad.net/kicad>
- About KiCad software i18n: <https://github.com/KiCad/kicad-i18n/issues>

**Publication date and software version**

Published on October 15, 2015.

---

## 1 Needed files and tools

Creating and/or maintaining translations do not need any skill in C++ programming: **there is no change to do in KiCad files.**

Translations are easy to do with a tool **PoEdit** that locate (in KiCad sources) sentences to translate and is able to create a dictionary for KiCad from translations created with this tool. So you need to install PoEdit, and get latest KiCad sources, and, for existing translations, get latest translations. Translations can be made under Linux, Window or MacOSX.

### 1.1 Download PoEdit

See: <http://www.poedit.net/>

### 1.2 Downloading KiCad sources

KiCad sources are currently hosted on Launchpad:

<https://launchpad.net/kicad>

Files can be downloaded from Launchpad by using a tool named "bazaar" (bzt in commands). So:

- Install, if not already done, the tool named bazaar (easy to install under all platforms): see <http://bazaar.canonical.com/>
- Download KiCad sources using the command `bzt branch lp:kicad <directory where sources files are copied>`
- You'll find this doc about translation and poedit configuration in Documentation on [http://docs.kicad-pcb.org/en/gui\\_translation\\_howto](http://docs.kicad-pcb.org/en/gui_translation_howto)

### 1.3 Download existing translations and docs

KiCad translations and documentations are also hosted on github at: <https://github.com/KiCad/kicad-i18n/>

Download translations using command:

```
git clone https://github.com/KiCad/kicad-i18n.git
```

## 2 Find sentences to translate

The different menus and tool tips in KiCad are internationalized, and can be easily translated into a local language *without source code modifications*.

The rules are:

- They are written in English.
- All strings which must be translated are written like: `_("hello world")`, and displayed "hello world" but if a dictionary is found translated into the locale language before displaying.
- A dictionary English→locale handle translation (one dictionary by language).

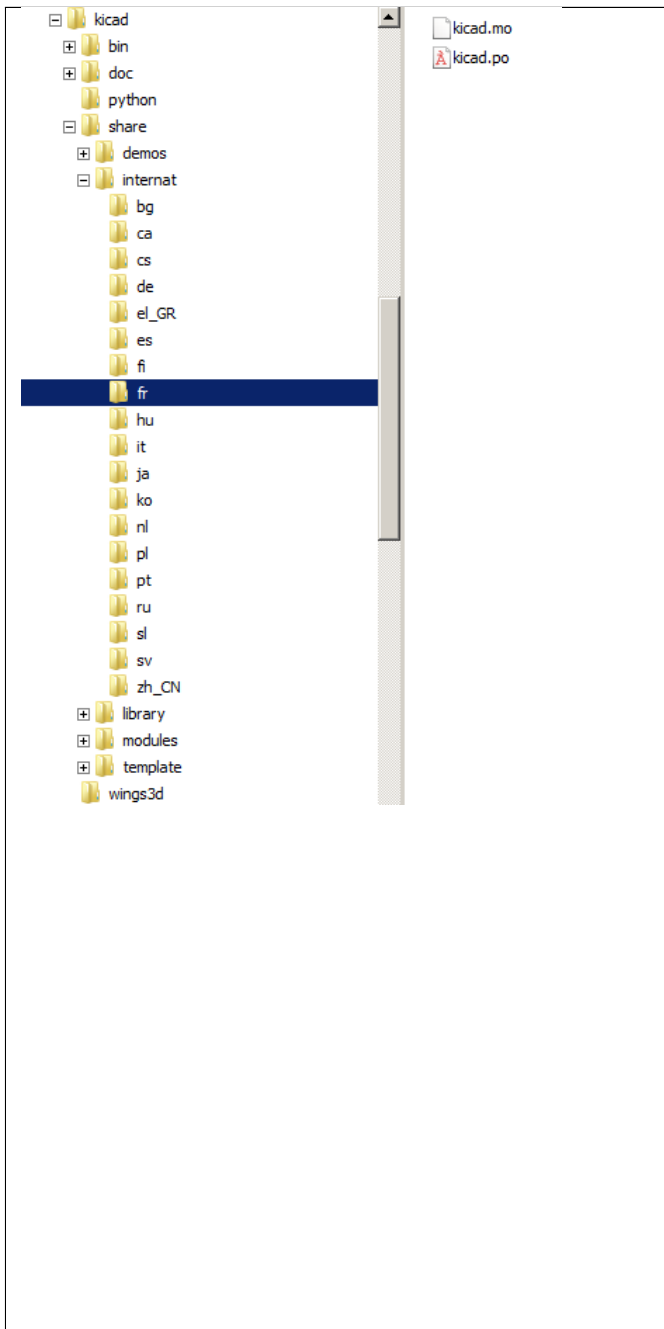
The easier way to create and maintain the dictionary English→locale is to use, **poedit**. Poedit scans KiCad sources and allows you to enter translations. You must download KiCad sources and set poedit in order to create translations.

## 3 KiCad tree for translations

### 3.1 Dictionary tree

The dictionary will be found by KiCad only if it is in a suitable path:

---

 <p>The screenshot shows a file explorer window for the Kicad project. The 'internat' directory is expanded, showing a list of locale folders: bg, ca, cs, de, el_GR, es, fi, fr (highlighted), hu, it, ja, ko, nl, pl, pt, ru, sl, sv, and zh_CN. Other visible directories include bin, doc, python, share, demos, library, modules, template, and wings3d. Files kicad.mo and kicad.po are visible in the right pane.</p>	<p>The suitable path is <b>kicad/internat/xx</b>, or <b>kicad/internat/xx_yy</b> with: <b>xx</b> = normalised locale indicator (short form) like:</p> <ul style="list-style-type: none"><li>• fr = france</li><li>• en = english</li><li>• es = spanish</li><li>• pt = portuguese</li></ul> <p>or: <b>xx_yy</b> = normalized locale indicator (long form) like:</p> <ul style="list-style-type: none"><li>• fr_FR</li><li>• en_GB</li><li>• en_US</li></ul>
--	---

## 3.2 Search path

Dictionaries and on-line help files are searched in this order:

- In the path in normalized locale indicator (long form) (kicad/internat/xx\_yy)
- In the path in normalized locale indicator (short form) (kicad/internat/xx)

And for on-line help files search is made in:

- In the path in normalized locale indicator (long form) (kicad/help/xx\_yy)
- In the path in normalized locale indicator (short form) (kicad/help/xx)

- kicad/help/en
- kicad/help/fr

---

**Note**

The main KiCad path is retrieved from the binary path, or (if not found):

UNDER WINDOWS:

- c:\kicad
- d:\kicad
- c:\Program Files\kicad

UNDER LINUX:

- /usr/share/kicad
  - /usr/local/share/kicad
  - /usr/local/kicad/share/kicad
  - /usr/local/kicad
- 

### 3.3 Files

In each directory there are 2 files **kicad/internat/xx**:

- internat.po (the dictionary file)
- internat.mo (the poedit work file)

## 4 Using poedit

### 4.1 Installation

Download and install poedit (<http://www.poedit.net>). Poedit exists on Windows, Linux and Mac OS X.

Download and unzip KiCad sources.

### 4.2 KiCad preparation

KiCad sources: in this example files are in f:/kicad/. All the strings to translate are tagged like \_("string to translate").

poedit must search the \_ (underscore) symbol to locate these strings.

One must add in KiCad the suitable directory for the dictionary (**kicad/share/internat/xx**). In this example, the directory is **kicad/share/internat/fr**.

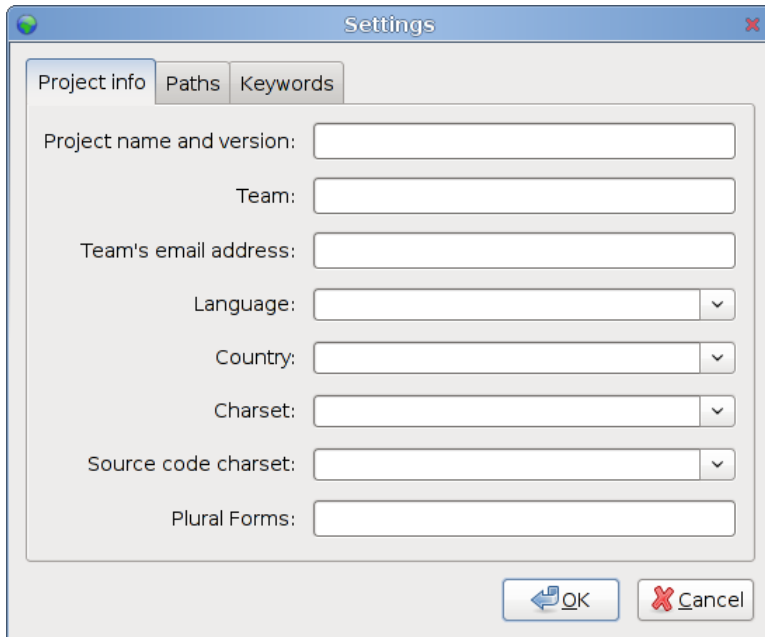
### 4.3 Poedit Configuration

Run poedit.

Run File/New catalog...

You should see something like:

---

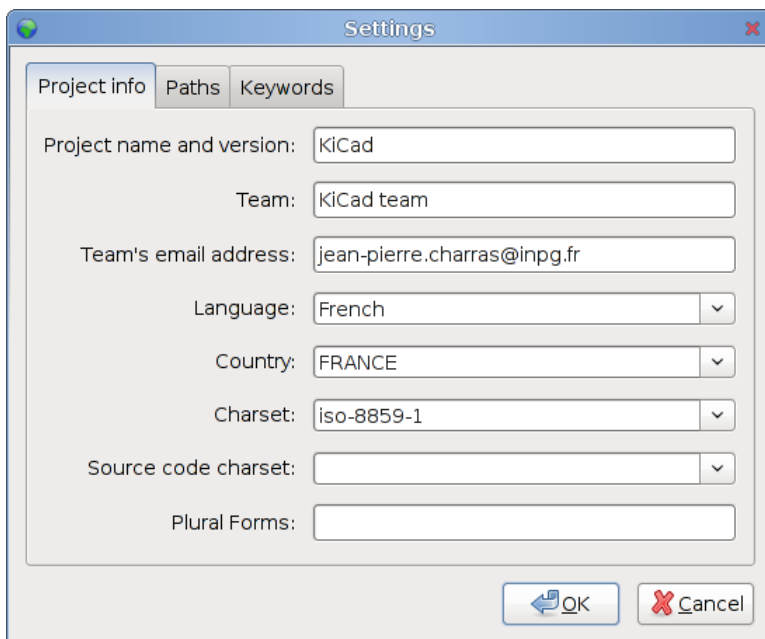


The screenshot shows a 'Settings' dialog box with three tabs: 'Project info', 'Paths', and 'Keywords'. The 'Project info' tab is selected. It contains the following fields:

- Project name and version:
- Team:
- Team's email address:
- Language:  (dropdown)
- Country:  (dropdown)
- Charset:  (dropdown)
- Source code charset:
- Plural Forms:

At the bottom right, there are two buttons: 'OK' (with a blue arrow icon) and 'Cancel' (with a red X icon).

#### 4.4 Project Configuration



The screenshot shows the same 'Settings' dialog box, but now the fields are filled with project information:

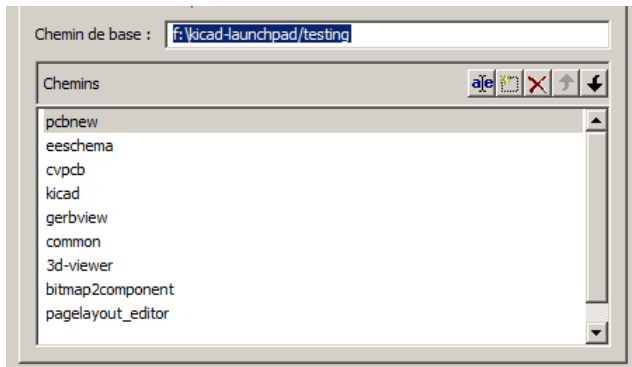
- Project name and version:
- Team:
- Team's email address:
- Language:  (dropdown)
- Country:  (dropdown)
- Charset:  (dropdown)
- Source code charset:
- Plural Forms:

The 'OK' and 'Cancel' buttons are still present at the bottom right.

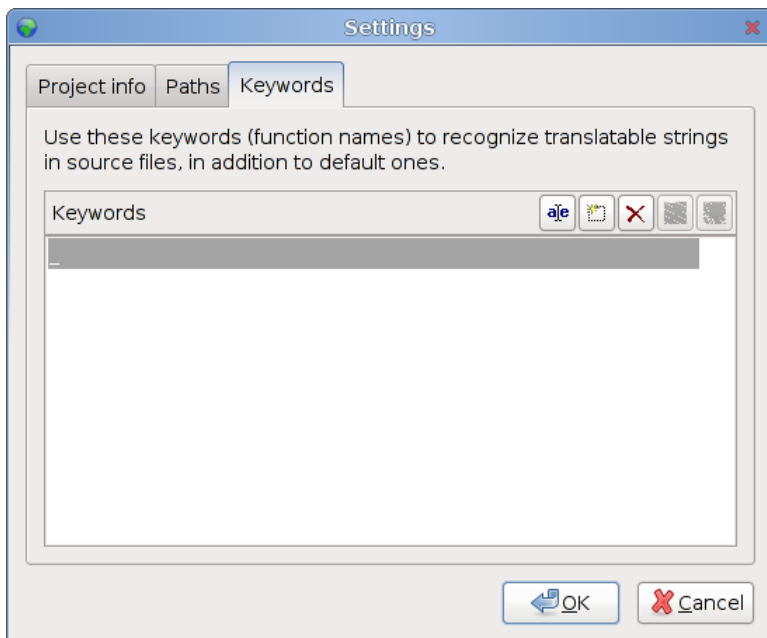
The source files are in English, so no need to choose something for source code.



## 4.5 Path and files Configuration



## 4.6 Keyword Configuration



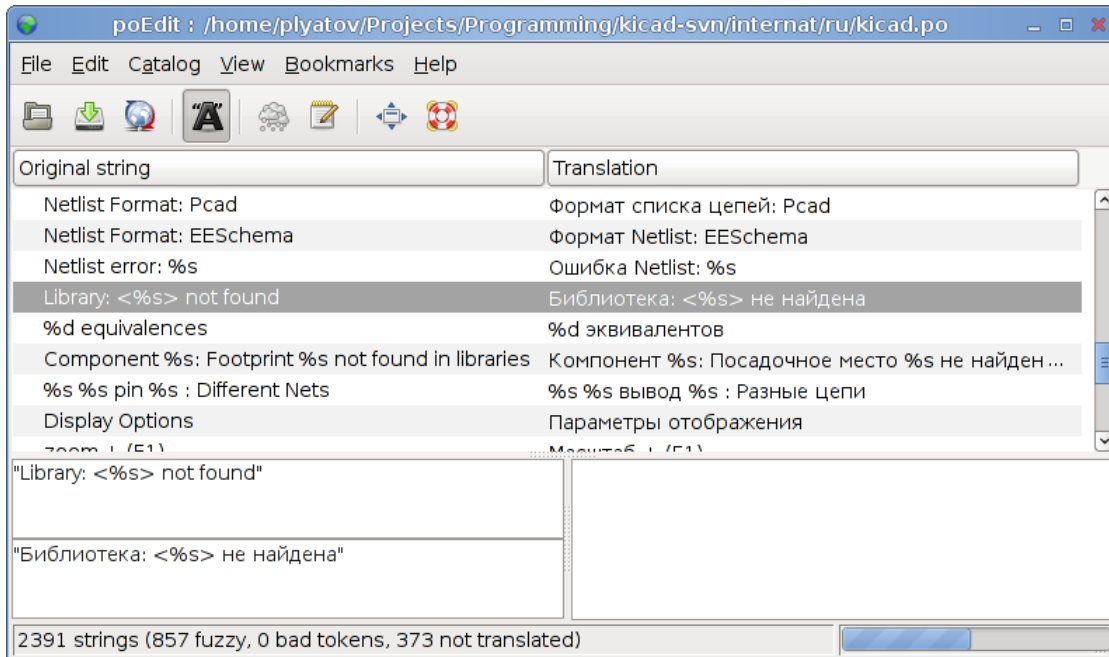
Only one keyword to enter: \_ (underscore) used as tag in source files.

## 4.7 Save the project

Save the new projet in **kicad/share/internat/xx** with the name **kicad.po**.

## 5 Create or edit a dictionary

Run poedit and load a project (here: **kicad.po**).



Run the command **Catalog/update from sources**.

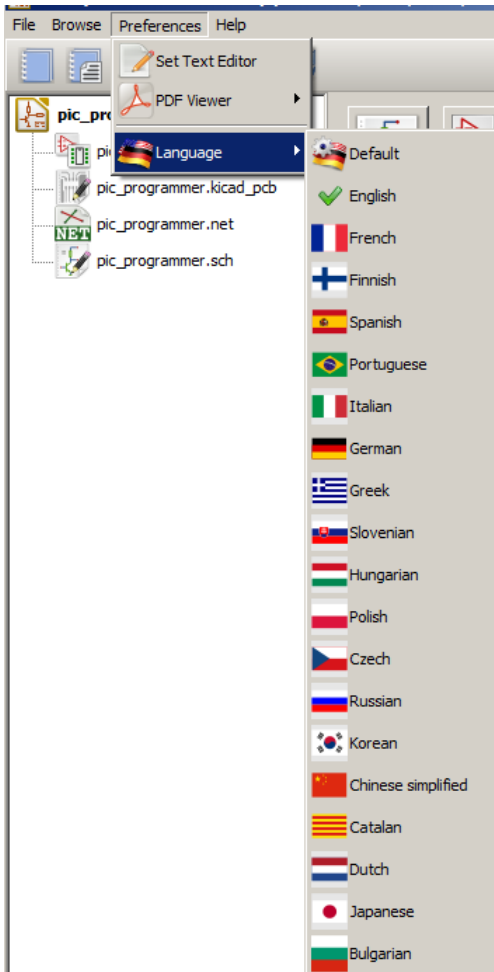
New strings (not yet translated) will be displayed on the top of the window.

## 6 Adding a new language entry in KiCad source code (devs only)

This step is NOT required. It is useful only for developers, and for testing purpose only

In KiCad we can force the used language.

It is highly recommended to use the default language.



But because developers have to test translations, a new entry in the language list can be useful for testing purposes.

## 6.1 Steps

### 6.1.1 Adding a new id in include/id.h.

→ In include/id.h, locate the sequence like:

```
ID_LANGUAGE_CHOICE,
ID_LANGUAGE_DEFAULT,
ID_LANGUAGE_ENGLISH,
ID_LANGUAGE_FRENCH,
ID_LANGUAGE_SPANISH,
ID_LANGUAGE_GERMAN,
ID_LANGUAGE_RUSSIAN,
ID_LANGUAGE_PORTUGUESE,
```

and add a new entry in list (which will be used later in menus) like:

ID\_LANGUAGE\_MY\_LANGUAGE before ID\_LANGUAGE\_CHOICE\_END.

### 6.1.2 Adding a new icon (aesthetic purpose only)

→ Create a new icon in SVG (Using Inkscape for instance) format: usually the country flag. For instance lang\_new.svg

Others language icons are in `common/bitmaps_png/source`

### 6.1.3 Editing bitmaps\_png/CMakeLists.txt

→ locate the text:

```
lang_catalan
lang_chinese
lang_bg
lang_cs
lang_def
lang_de
lang_en
lang_es
lang_fr
lang_fi
lang_gr
lang_hu
lang_it
lang_jp
lang_ko
lang_nl
lang_pl
lang_pt
lang_ru
lang_sl
```

and add the new filename (without extension): lang\_new

### 6.1.4 Editing include/bitmaps.h

→ locate the text:

```
EXTERN_BITMAP( lang_bg_xpm )
EXTERN_BITMAP( lang_catalan_xpm )
EXTERN_BITMAP( lang_chinese_xpm )
EXTERN_BITMAP( lang_cs_xpm )
EXTERN_BITMAP( lang_def_xpm )
EXTERN_BITMAP( lang_de_xpm )
EXTERN_BITMAP( lang_en_xpm )
EXTERN_BITMAP( lang_es_xpm )
EXTERN_BITMAP( lang_fr_xpm )
EXTERN_BITMAP( lang_fi_xpm )
EXTERN_BITMAP( lang_gr_xpm )
EXTERN_BITMAP( lang_hu_xpm )
EXTERN_BITMAP( lang_it_xpm )
EXTERN_BITMAP( lang_jp_xpm )
EXTERN_BITMAP( lang_ko_xpm )
EXTERN_BITMAP( lang_nl_xpm )
EXTERN_BITMAP( lang_pl_xpm )
EXTERN_BITMAP( lang_pt_xpm )
EXTERN_BITMAP( lang_ru_xpm )
EXTERN_BITMAP( lang_sl_xpm )
```

and add a line to include the new icon name called lang\_new\_xpm (\_xpm added to the filename).

### 6.1.5 Editing common/edaappl.cpp

→ Locate:

```
struct LANGUAGE_DESCR
{
```

```

    int          m_WX_Lang_Identifier;           // wxWidget locale identifier (see ↵
    wxWidget doc)
    int          m_KI_Lang_Identifier;           // kicad identifier used in menu ↵
    selection (see id.h)
    const char** m_Lang_Icon;                   // the icon used in menus
    const wxChar* m_Lang_Label;                 // Label used in menus
    bool         m_DoNotTranslate;              // set to true if the m_Lang_Label ↵
    must not be translated
};

#define LANGUAGE_DESCR_COUNT 14
static struct LANGUAGE_DESCR s_Language_List [LANGUAGE_DESCR_COUNT] =
{
    {
        wxLANGUAGE_DEFAULT,
        ID_LANGUAGE_DEFAULT,
        lang_def_xpm,
        _( "Default" )
    },
    {
        wxLANGUAGE_ENGLISH,
        ID_LANGUAGE_ENGLISH,
        lang_en_xpm,
        wxT( "English" ),
        true;
    },
    {
        wxLANGUAGE_FRENCH,
        ID_LANGUAGE_FRENCH,
        lang_fr_xpm,
        _( "French" )
    },

```

and add a new entry like:

```

    {
        wxLANGUAGE_MY_LANGUAGE,
        ID_LANGUAGE_MY_LANGUAGE,
        lang_new_xpm,
        _( "My_language" )
    },

```

`wxLANGUAGE_MY_LANGUAGE` is the wxWidgets language identifier for the country (see wxWidget doc).

### 6.1.6 Recompiling

You should be a PNG Maintainer (see `bitmaps_png/CMakeLists.txt` file), i.e compile KiCad with the option `MAINTAIN_PNGS` on Obviously, this is the next and last step.