



KiCad:

"Defaults" button for the Hotkeys Editor dialog

Bc. Jan Doležal (dolezj21@fel.cvut.cz)

Summer 2015/16

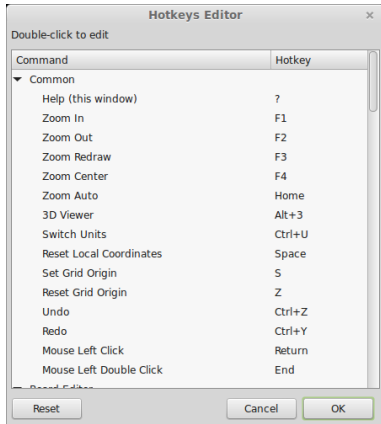
■ Kicad Project

- ▶ Electronics design automation
 - ▶ scheme
 - ▶ footprint
 - ▶ printed circuit board (PCB)
 - ▶ 3D viewer
- ▶ GUI built using multiplatform wxWidgets framework
- ▶ 25 yrs of development



Task

- ▶ Add button to 'Hotkeys editor' allowing to set hotkeys to the default key shortcut
- ▶ Modification was on the Wishlist in the bugtracker under the Bug number #1395442



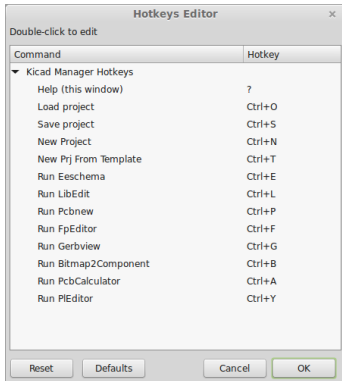
■ Description of Changes made

▶ Form

- ▶ added 'Defaults' button to the form template in XML
- ▶ regenerate .cpp and .h for the form from modified template using wxFormBuilder

▶ Functionality

- ▶ modified HOTKEY class
 - ▶ variable with default value
 - ▶ method ResetKeyCodeToDefault() replacing current hotkey value with default
- ▶ on 'Defaults' clicked loop through HOTKEYs in the dialog and call ResetKeyCodeToDefault() for each of them



■ Results Evaluation

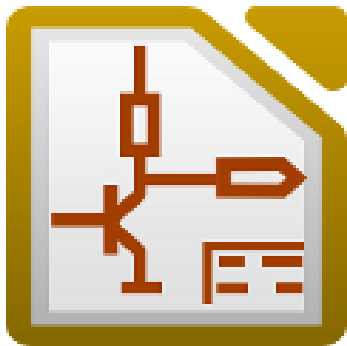
- ▶ First implementation not accepted
 - ▶ non-invasive changes
 - ▶ some compromises decreasing maintainability
- ▶ Project developers proposed changes I should make
- ▶ Second implementation included proposed changes and it was accepted to the mainline

■ Community Collaboration

- ▶ First approach
 - ▶ very friendly
- ▶ What do I appreciate
 - ▶ very fast responses
 - ▶ developers were helpful in resolving issues
 - ▶ constructive objections
- ▶ Anything to be improved?



■ Conclusion





KiCad:

"Defaults" button for the Hotkeys Editor dialog

Bc. Jan Doležal (dolezj21@fel.cvut.cz)

Summer 2015/16