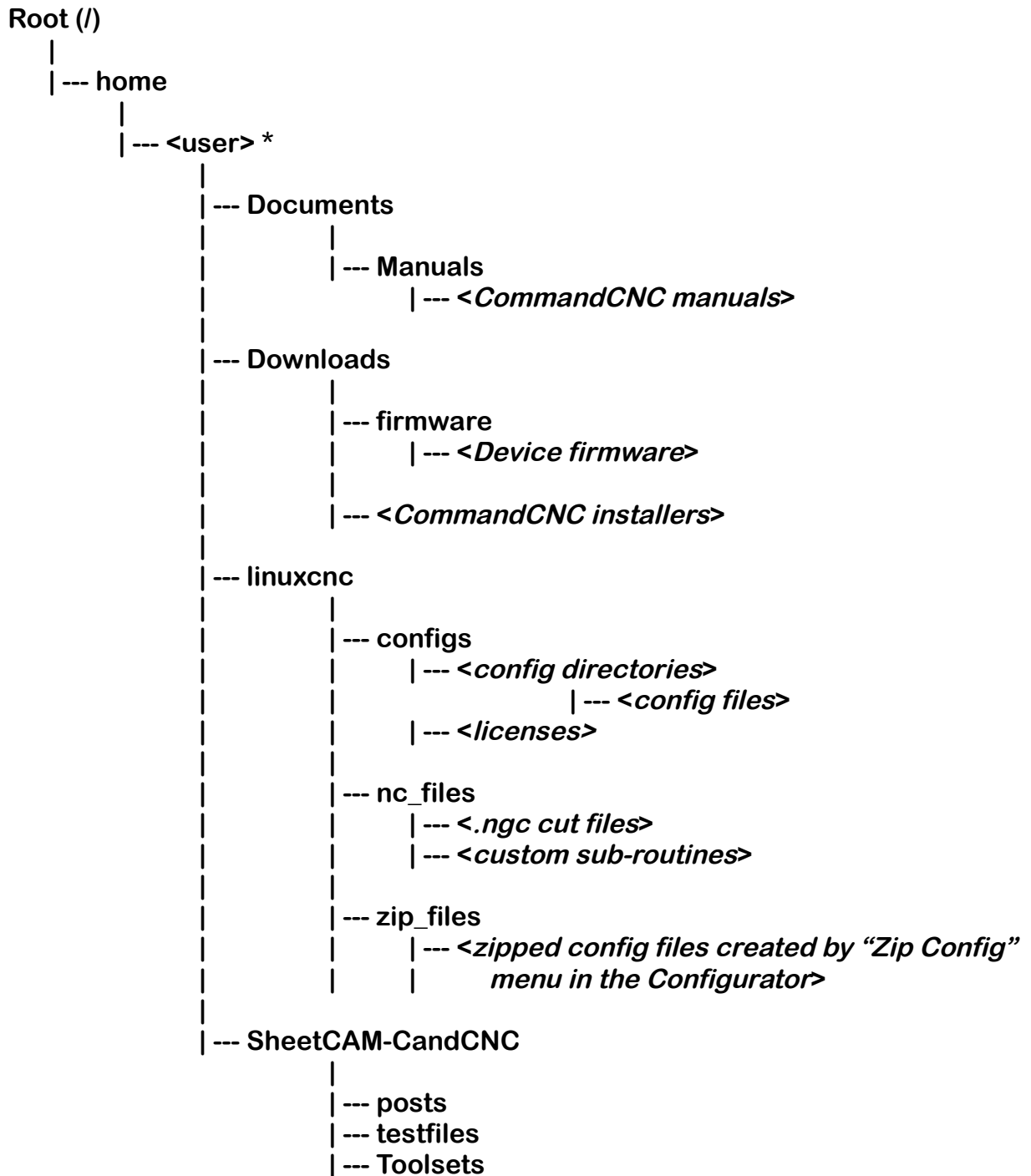


COMMANDCNC® UPDATING INSTRUCTIONS AND NOTES

REV 1.2.2



FILE LOCATIONS FOR COMMANDCNC®

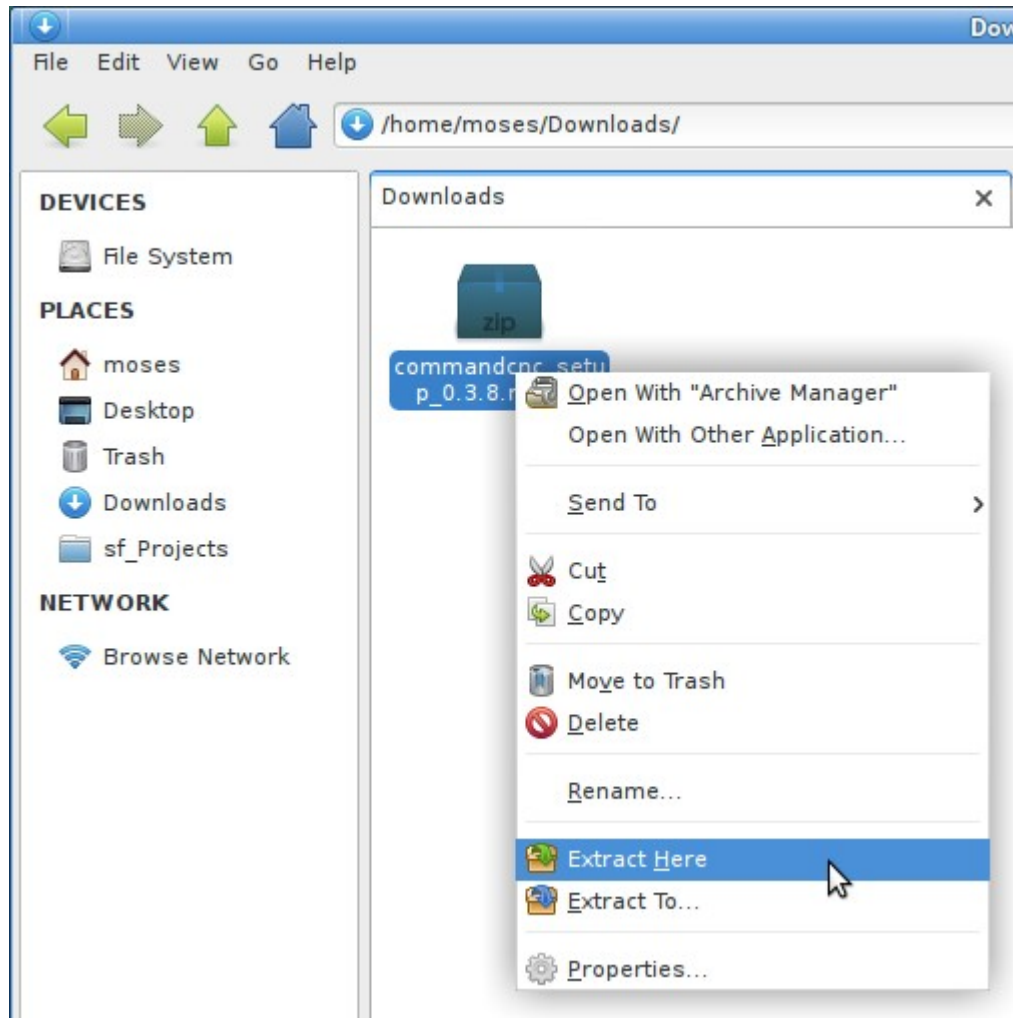


* *<user>* will be the same as user name you logged in with. In the file manager this directory will be the top link under "Places" in the left panel.

UPDATING COMMANDCNC®

Larger updates may be distributed as a combined setup package name something like this: `commandcnc_update_1.0.3_i386.run.zip`

To install this kind, save the file onto your computer – I recommend to put it in your Downloads directory. Then open the file manager (Home icon on the desktop) and find the file, then extract the installer by right-clicking on the file and selecting “Extract Here”

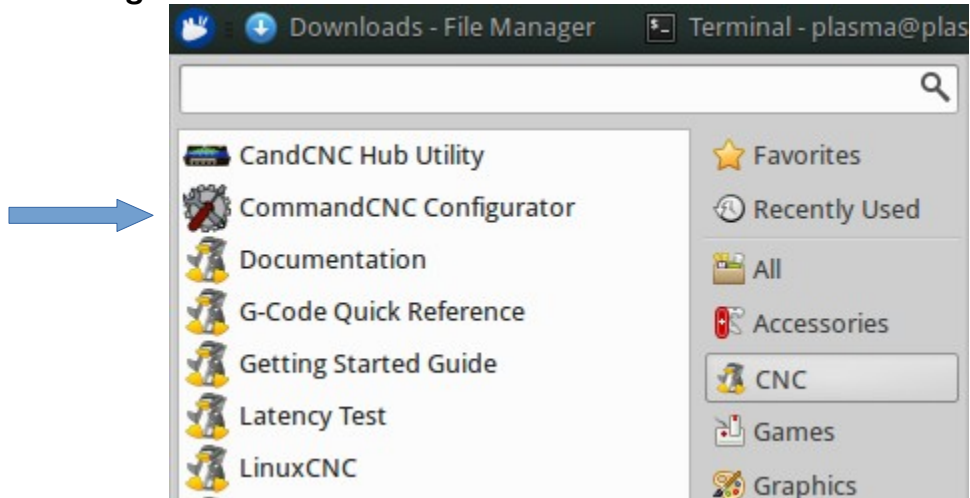


Then double-click the `commandcnc_update_1.0.3_i386.run` file to start the installation. Enter your password when prompted and when it is finished you can close the window and go to the next step.

UPDATING COMMANDCNC®

After installing an update to CommandCNC, it is imperative that you run the CommandCNC Configurator to update your config(s).

Start the configurator:



- In the configurator, click the “File->Open Config...” menu or the “Open” button on the toolbar and select the config to edit and click Ok.
- Check the notes below for this release of CommandCNC and see if any settings need to be changed in your config, and make the changes as necessary.
- Click “Save” on the toolbar (or the File→Save Config menu) to save changes and update files.
- Then exit the configurator.

COMMANDCNC® CHANGELOG

CommandCNC 1.2.2

- Fixed: ethernet port was not locked to CommandCNC only
 - Chargepump and outputs now turn off in Estop
 - Gantry now only does homing logic while in a homing move
-

CommandCNC 1.2.1

- Fixed Configurator bug in PN200 settings
 - Fixed Configurator bug in loading THC Max Velocity setting
 - Kernel and misc system config updates to fix realtime problems in new CNC-Xf 64-bit systems.
-

CommandCNC 1.2

- Moved most THC settings out of CommandCNC into the Configurator
 - Added PN200 settings tab in the Configurator
 - Now shows power supply info in diagnostics if ESPIIA is connected to hub
 - Added Tipsaver back in with enable checkbox in Configurator
 - New part runtime counter in diagnostics – shows time the current gcode has been running. It starts over each time the code is run.
 - Now checks to see if gcode exceeds machine limits on each run and stops and shows an error if it does.
-

CommandCNC 1.1.1

- Added checkbox in Advanced page to select homing method.
-

CommandCNC 1.1

- Auto-Set support for DTHC
 - Added Fixture Offsets page for setting up offsets for work-holding fixtures
 - Output buttons can now be customized
 - Support for two more axes (U and V)
 - New “Auto” setting to auto-calculate THC delay. This is on by default.
 - UI rework to reduce some clutter and better fit everything on the screen
 - Fixed: PN200 step-jogging with the joystick only moved in one direction
 - Changes to tolerate higher latency and bad packets without following errors
 - Miscellaneous other fixes and improvements.
-

CommandCNC 1.0.3

- Oxy-fuel config bug fixed.
 - “Use Hardware Enable Pin” option removed, and that option is now on permanently.
-

CommandCNC 1.0.2

- Improved simulation configs to handle more than 3 axes
 - HyT improvements to re-init and clear error codes
 - Router config bug fixed.
-

CommandCNC 1.0

- Added initial support for multiple languages
 - Added support for EBOB with more inputs and outputs
-

CommandCNC 0.9.7

- Jogging now stops if CommandCNC window loses focus.
 - Added more checks for invalid input setups in Configurator
 - The update installer installs a new linux kernel and sets some options to fix freezing and reboot issues with certain computers.
-

CommandCNC 0.9.6

- Added support for DTHC-5
 - Can now have up to 16 ESTOP inputs
-

CommandCNC 0.9.5

- Bug fixes in Hub Utility and CommandCNC
 - HyT fix to re-init after power cycle
 - Fixes to ethernet port setup script
-

CommandCNC 0.9.4

- Set [TRAJ] MAX_VELOCITY now based on combination of fastest axes
- Fix for DTHC-IV: rev 2.0 and greater firmware **MUST USE THIS RELEASE OR GREATER!**

Note:

DTHC-IV rev 2.0 firmware file is included and will be in:
“/home/plasma/Downloads/firmware”

This update fixes an intermittent bug in certain configurations where the Z would move the wrong direction. It also contains a fix for a possible mis-count of Z steps while cutting.

Use the Hub Utility to update the DTHC-IV with this file.

CommandCNC 0.9.2

- Raised limit on touchoff feedrate to support metric systems
- Installer changes to help with latency issues

CommandCNC 0.9.1

- Added support for PN200 hand pendant
- Fixed: incremental jog was not using jog speed slider setting
- Added hot key for feedhold - the 'space' key

CommandCNC 0.8.9

- Added “Disable HOLD” checkbox. Makes CommandCNC ignore the HOLD signal for things such as manual cutting or resuming a cut.
- New “Clear Messages” button in the Messages window.
- Added a light color scheme for the code window and live view. See the Advanced tab in the Configurator to change color schemes.
- Fixed line number glitch while running subroutine
- Fixed file reload error if file extension has no filter program in ini
- Add box to show and jump to line number in gcode
- Reload live view after zeroing an axis
- Fixed tolerance setting being lost on E-stop
- Tool cross-hairs now instead of circle (cone)
- Live view color improvements
- Configurator: option to require homing before running code

CommandCNC 0.8.6

- Added Save as template in Configurator
- Kernel version reported in Help→About
- More config checks in Configurator
- Installer improvements
- Misc bug fixes

- CommandCNC Ethernet Port Setup utility in Main Menu→CNC menu
-

CommandCNC 0.8.5

- New button to toggle display of tool in live-view
- Minor fixes to live-view buttons
- Tipsave enable state now saved in prefs
- Checkbuttons in Manual tab for "Skip Touchoffs" (Block Delete) and "Stop at M1" (Optional Stop)
- Now automatically adds the block delete character (/) to o<touchoff>, G38.2, and G92 lines. This is done automatically when loading a file.
- Will now show files with the following extensions in the "Open" dialog: .ngc, .NGC, .nc, .NC, .tap, .TAP
- Added button to clear MDI history.
- Added menu item to clear messages and status bar.
- More config checks in Configurator
- Homing velocity set properly when home switch is enabled on an axis. Previously the homing velocity would be set to zero if there was no home switch configured for an axis, but when a home switch was enabled in the Inputs page, the homing velocity would be left at zero which left homing disabled for that axis. Now the user entered velocity is remembered, or a default of 20 IPM (or 508mm/M) used if the velocity is zero and the home switch is enabled. A side effect is that you can no longer enter a homing velocity of 0. To disable a home switch you simply set the input the home switch is on to "Not Used".
- Configurator UI re-worked with menubar and toolbar
- Added "Zip Config" menu in Configurator, zips current config
- Configurator now loads last opened config at startup
- Fixed inaccuracy in touchoff position
- Improved dry-run support (just click "disable torch"), and made HOLD work while cutting with THC off in manual mode.
- Fix Motor step/dir timings not being read from ini file (they would be saved fine, but not read back next time and it would just show the defaults).
- Updated Sheetcam post (rev18) with fixes for peck tools and a fix to allow parking in Sheetcam to work.

IMPORTANT!

Now that the Configurator has separate "Save" and "Exit" buttons, when you load a config after updating to 0.8.5 you must first click the "Save" button to actually update the config before exiting!

CommandCNC 0.8.0

- Dual switch probing will now use floating switch if ohmic fails – *needs modification to hardware and config for this to work!*
- Probe limit now set in Manual tab in CommandCNC
- Torch off before end of cut support - using M62 P2

Note:

The features in this release require the use of our SheetCAM post rev17B or later. rev17B is included in the install and placed in the SheetCAM-CandCNC as shown in page 2 of this document.

CommandCNC 0.7.0

- Fixed g-code sometimes rewinding after hitting “Stop”
- New “simple” run-from-line function which fixes delays and other problems with the normal “complex” run-from-line function.
- New context sensitive help dialog in the Configurator. Click the “Help!” button in the top right corner to access it. Leave it open and change pages in the Configurator and the help will change for the new page.
- Added more homing options – direction, offsets, final “Move to” position. *IMPORTANT: See the new help in the Configurator, and the note just below here.*
- Feed inhibit input function should now work with non-plasma setups.

IMPORTANT NOTE:

In previous releases, the Z home switch offset always showed as a positive number. **THIS HAS CHANGED!** When you open your existing config, Z home switch offset will now be a negative number, and it should remain that way. If you are setting up a new config, the Z home switch offset number must now be entered as a negative number. *If you enter a positive number the Z will move farther down after homing instead of moving up!*

CommandCNC 0.6.1

- Fixed bug in simulator mode.

CommandCNC 0.6.0

- Velocity Anti-Dive: disables THC when motion slows too much
- New Oxy-fuel machine type.
- Added ability to change Step/Dir timing
- Stop on Fault option for THC

- Improved error reporting, with button to create zip of config and error information
 - “Override Limits” button for jogging off of limit switches
 - Eliminated popup notifications that had to be clicked to make them go away
 - Fixed Z DRO for non-plasma setups.
-

CommandCNC 0.5.3

- Template support in Configurator
 - Configurator now can create desktop shortcuts
 - Configurator does more config checking
-

CommandCNC 0.5.2

- Added Outputs tab for selecting output functions
 - Fixed startup error when gantry has no home switches
 - Load material subroutine now uses G53 machine coordinates
-

CommandCNC 0.5.1

- Fix step polarity into DTHC-IV
 - Ethernet card firmware updates now in Configurator
 - Allow floating point Z steps-per-unit
-

CommandCNC 0.5

- Improved support for Router screen and setup
 - Sheetcam post rev 14 with scriber fixes
 - Added Sheetcam tools for CommandCNC
-

CommandCNC 0.4

- Improved support for 5 axis systems
- Added support for 6-motor systems
- Handles axes with no home switch properly now
- Added support for auto-squaring with two home switches on a gantry

Note:

To use two home switches on a gantry, set two inputs to the same axis home switch. So for a Y gantry, you could set “Y home input” = “Y Home switch”, and “A home input” = “Y Home switch”

CommandCNC 0.3.9

- Fixed Z synchronization problem for cuts with DTHC OFF

CommandCNC 0.3.8 (since 0.3.3)

- Dry run support - motion should not hold for ARC OK when THC is off now.
- Step polarity settings
- Settings to stop probe errors when homing and jogging
- Torch Disable check box - good for dry run and for resuming a cut.
- B axis support
- Keymapping fixes
- Show CommandCNC version in About dialog
- Z zero button now uses G92 offset like the touchoff

Change Notes:

- The E-stop input polarity has changed since early versions. The “Invert” checkbox for E-STOP input should be cleared in version 0.3.8 and greater
- Z home input should be set to Ohmic Sensor + Z Home switch even if you do not have an Ohmic Sensor.