**OpenFOAM Linux Cheat Sheet**

|  |  |
| --- | --- |
| **File & Diredtory** | |
| mkdir | [OPTION]... DIRECTORY... |
| -p | make parent directories |
| $ mkdir -p $FOAM\_RUN | |
| cp & mv | [OPTION]... [-T] SOURCE DEST  [OPTION]...SOURCE...DIRECTORY |
| -b | make a backup of each existing destination file |
| -n | do not overwrite an existing file |
| -i | prompt before overwrite |
| -r | copy(move) directories recursively |
| $ cp -r $FOAM\_TUTORIALS . | |
| rm | [OPTION]... FILE... |
| -f | never prompt |
| -i | prompt before every removal |
| -r | remove directories recursively |
| $ rm -ri 0\.[^o]\* | |
| ln | TARGET... DIRECTORY |
| -s | soft link |
| $ ln -s /opt/openfoam30/doc/Guides-a4/UserGuide.pdf ~/Desktop/UserGuide | |
| touch | [OPTION]... FILE... |
| $ touch cavity.OpenFOAM | |
| chmod | [OPTION]...MODE[,MODE]...FILE...  [OPTION]...OCTAL-MODE FILE... |
| $ chmod +x Allrun | |
| find | [OPTION] [path…] [expression] |
| $ find tutorials/ -name simpleFoam | |
| locate | [OPTION]... PATTERN... |
| $ locate interPhaseChangeFoam.C | |
| **Text** | |
| echo | [SHORT-OPTION]... [STRING]... |
| -e | enable interpretation of backslash escapes |
| $ echo -e '\n'$FOAM\_SRC'\n' | |
| less | [FILE]... |
| $ less $WM\_PROJECT\_DIR/Allwmake | |
| vi | [FILE]... |
| $ vi ~/.bashrc | |
| grep | [OPTIONS] PATTERN [FILE...] |
| -i | ignore case distinctions in both the PATTERN and the input files |
| -h | Suppress the prefixing of file names on output. |
| $ grep -ih build $WM\_PROJECT\_DIR/Allwmake | |
| **Binary & Package** | |
| whereis | [-bmsu] filename... |
| -b | Search only for binaries |
| $ whereis -b paraview | |
| whatis | name ... |
| $ whatis evince | |
| man | [OPTION] page… |
| $ man chmod | |
| apropos | [OPTION] keyword… |
| $ apropos plot | |
| apt-get |  |
| update | resynchronize the package index files |
| upgrade | install the newest versions of all packages currently installed on the system |
| install | install packages |
| remove | uninstall packages |
| $ sudo apt-get install openfoam4 | |
| apt-cache |  |
| search | perform a full text search on all available package lists |
| $ apt-cache search openfoam | |
| **Process & Environment** | |
| ps | [OPTIONS] |
| -e | see every process on the system |
| $ ps -e | |
| kill | [options] <pid> [...] |
| -9 | send SIGKILL |
| -l | list signal names |
| $ kill -9 12974 | |
| env | [OPTION]... [-] [NAME=VALUE]... |
| $ env | grep ^FOAM\_ | |
| alias | [-p] [name[=value] ... ] |
| $ alias mv='mv -i' | |
| source | FILE... |
| $ source /opt/openfoam30/etc/bashrc | |

**Redirection**

|  |  |
| --- | --- |
| command1 > file1 | execute command1, placing the output in file1 |
| command1 >> file1 | append output to the end of the file |
| command1 < file1 | execute command1, with file1 as the source of input |
| tr a-z A-Z << END\_TEXT | read from a stream literal |
| command1 | command2 | executes command1, using its output as the input for command2 |

Copyright © 2016 Zhang Qi