

Dear FOAMers

Here is an installation procedure for OpenFOAM-1.6-ext on openSUSE 11.4 64-bit

There is a possibility that this procedure is not complete, as it worked on my laptop computer, but not on my desktop computer, both running 64 bit openSUSE 11.4. Nevertheless, I decide to post this procedure, since this might work on your computer, or better yet, you may come with a better solution.

Note that much of the info here is attained from cfd-online. The problem has however been that the information is scattered across the board,... really fragmented and hard to read.

My objective here is try to give one coherent one page procedure of how to install the 1.6-ext of OpenFOAM on openSUSE 11.4 64-bit, so you can enjoy for example the fun of grid-rotation with ggi or sliding-interface.

Note, some of the steps below are really redundant, but still I recommend that you follow these steps to the letter. Again, this installation procedure may not be complete, so you do this at your own risk.

I will change this file as new information becomes available! (step 19 is the newest one, but have not tried it yet)

I hope this is of help.

(0)

Make sure that you have internet connection during the whole installation procedure. A lot will be automatically downloaded during installation.

The whole installation procedure of 1.6-ext can easily take up to 8 hours (even more), and it is not fully automated, so make sure that you have the whole day for your self.

(1)

Install OpenFOAM 1.7.x as is well explained by Alberto at <http://albertopassalacqua.com/?p=920> (or at least install the programs and libraries mentioned there, like the "sudo zypper install -t pattern devel_C_C++ ")

...

or install OpenFOAM 2.0.x, as explained at <http://www.openfoam.com/download/source.php>

If your are a newcomer to OpenFOAM, you should really do this step as a training.

Installing the 1.6-ext version is more difficult,.. OK

(in fact if you are really a newcomer, you should perhaps leave 1.6-ext alone for now)

(2)

sudo zypper update

...reboot your computer (just in case), and then install some libraries,...

```
sudo zypper install flex rpm alien bison byacc texinfo m4 libglib2.0-dev libglu1-mesa-dev libiodbc2-  
dev libmpfr1 libmpfr1ldbl libmysqlclient15-dev libpng12-dev libpq-dev libsqlite0-dev libtiff4-dev  
mesa-common-dev xorg-dev zlib1g-dev libstdc++5 ibglib2.0-dev libQtWebKit4-32bit libdrm-32bit
```

liblcms1-32bit libmng-32bit libmysqlclient_r16-32bit libqt4-32bit libqt4-qt3support-32bit libqt4-sql-32bit libqt4-sql-mysql-32bit libqt4-sql-sqlite-32bit libqt4-x11-32bit libQtWebKit4-32bit libstdc++33 Mesa-32bit cmake libqt4-devel gnuplot

... and reboot your computer once more after this (just in case)

Note some of these packages WILL NOT be found,... don't worry about that. I just haven't had time to clean the above.

(3)

edit your ~/.bashrc
and add the two lines

```
export QT_BIN_DIR=/usr/bin
. $HOME/OpenFOAM/OpenFOAM-1.6-ext/etc/bashrc
```

and comment you 1.7.x line with "#", i.e. have
. \$HOME/OpenFOAM/OpenFOAM-1.7.x/etc/bashrc

... thereafter, save your .bashrc file and exit!

(4)

```
cd ~/OpenFOAM
git clone git://openfoam-extend.git.sourceforge.net/gitroot/openfoam-extend/OpenFOAM-1.6-ext
```

NOTE, IMPORTANT: log out and log in again (or reboot your system), if the 1.7.x paths were active in step (3), above. Otherwise your 1.7.x and 1.6-ext paths will be mixed, which will be fatal.

(5)

make sure that you have ...

```
compilerInstall=System
```

...in the file...

```
~/OpenFOAM/OpenFOAM-1.6-ext/etc/settings.sh
```

(...this should be the default)

(6)

```
cd $WM_PROJECT_DIR/ThirdParty/
```

```
./AllMake.stage1
```

NOTE: Do NOT [as in NEVER] run AllMake.stage0,... leave it alone!!!

(7)

```
source your ~/.bashrc (source ~/.bashrc), or better yet open a new xterm and quit the old one.
cd $WM_PROJECT_DIR/ThirdParty/
```

This will initialize the CMAKE_DIR environment type...

```
echo $CMAKE_DIR
```

...in your xterm (or konsole/gnome-terminal) to make sure. If empty try again. Dont start the next step until you have something like this ...

```
/home/jonelvar/OpenFOAM/OpenFOAM-1.6-ext/ThirdParty/packages/cmake-2.8.3/platforms/linux64GccDPOpt
```

(8)

```
cd $WM_PROJECT_DIR/ThirdParty/  
./AllMake.stage2
```

(9)

source your ~/.bashrc (source ~/.bashrc), or better yet open a new xterm and quit the old one.

type...

```
echo $FOAM_MPI_LIBBIN
```

...to make sure that you have MPI libs env in order!

You should get something like...

```
/home/jonelvar/OpenFOAM/OpenFOAM-1.6-ext/lib/linux64GccDPOpt/openmpi-1.4.3
```

(10)

```
cd $WM_PROJECT_DIR/ThirdParty/  
./AllMake.stage3
```

(11)

source your ~/.bashrc (source ~/.bashrc), or better yet open a new xterm and quit the old one.

[optinal: you could do "env | grep Third" to see the ThirdParty env variables that is present at this stage]

(12)

```
cd $WM_PROJECT_DIR/ThirdParty/
```

edit Allwmake.stage4 and add

```
QT_THIRD_PARTY=1
```

```
QT_BIN_DIR=/usr/bin
```

...these commands are supposed to be approximately in lines 54 and 55, i.e. just before

```
echo =====
```

```
echo Starting ThirdParty AllMake: Stage4
```

```
echo =====
```

```
echo
```

(13)
source your ~/.bashrc (source ~/.bashrc), or better yet open a new xterm and quit the old one.

NOTE: I don't think this step is necessary, but just do it just in case in the space.

(14)
cd \$WM_PROJECT_DIR/ThirdParty/
./AllMake.stage4

(15)
source your ~/.bashrc (source ~/.bashrc), or better yet open a new xterm and quit the old one.

(16)
cd \$WM_PROJECT_DIR
./Allwmake 2>&1 | tee wmake_log_file_A.txt

(you could also include the "2>&1" thingi if you like, but not necessary, i.e. ./Allwmake 2>&1 | tee wmake_log_file_A.txt)

NOTE NOTE NOTE: Your compilation at this stage WILL crash at some point. That's OK (provided that all the above steps have been successful)! When it does, do the steps below...

(17)

mkdir -p \$FOAM_RUN
mkdir -p \$FOAM_USER_APPBIN
mkdir -p \$FOAM_SITE_APPBIN

source your ~/.bashrc (source ~/.bashrc), or better yet open a new xterm and quit the old one.

cd \$WM_PROJECT_DIR
./Allwmake | tee wmake_log_file_B.txt

(you will likely see some error, but don't mind it)

(18)
source your ~/.bashrc (source ~/.bashrc), or better yet open a new xterm and quit the old one.

cd \$WM_PROJECT_DIR
./Allwmake | tee wmake_log_file_C.txt

(...I did this in total 3 times, i.e. steps 16 to 18)

Now you should have a fully functional OpenFOAM-1.6-ext at your fingertip
cd \$WM_PROJECT_DIR
du -hs

and you install should be about 8.5 Gb (is at least so in my case),... anything less means not everything has been installed.

The reason for that is most likely that some libraries are missing on your computer (and not included in step (2) above).

(19)

Additional step which I found on cfd-online (do this only if the above fails)

delete ALL of the .dep files an lnInclude folders, and the compile everything again.

./Allmake again (i.e. steps 16 to 18)

several times
