Source pack installation of OpenFOAM. 4.0 into RHL

Open CAE Local Study Meeting

Y. Takagi

July 2, 2016

Download the packages from the official website

OpenFOAM

The OpenFOAM Foundation

Home | Download | Development | Resources | Contact Us

28th June 2016

Download v4.0 | Source Pack

The source code of OpenFOAM v4.0 and related third-party software can be downloaded as `tar.gz` compressed archive files:

- http://download.openfoam.org/source/4-0
- http://download.openfoam.org/third-party/4-0

The archive files download with inconvenient file names, so we recommend following the instructions below where they are unpacked immediately into the source directories, which avoids storing the files themselves.

Tested Platforms

OpenFOAM is developed and tested on Linux, but should work with other POSIX systems. OpenFOAM-4.0 and ThirdParty-4.0 have been tested on the following Linux distributions:

LATEST NEWS

OpenFOAM 4.0 Released
28th June 2016

Download v4.0 | Ubuntu
28th June 2016

Download v4.0 | Source Pack
28th June 2016

New Website Launched
5th May 2016

OpenFOAM 3.0.1 Released
15th December 2015

Where should we install?

- If you have a machine only for OpenFOAM, you can upgrade the libraries of the system.

  “Dependent packages required for RHEL 6.5 need updating to newer versions, e.g. Gcc needs upgrading from v4.4 to v4.5+. These upgrades may be available under a RHEL support subscription. If not, then upgrades can be obtained from unofficial repositories using the following instructions: ”

- If you do not want to upgrade the system casually, the compatible softwares/libraries such as GCC and MPI are built in the local ThirdParty directory.

Our machine environment

- OS: CentOS 6.5 (64 bit)
- System libraries:
  - GCC 4.4.7
  - cmake 2.6-patch 4
  - Qt 4.6.2
- Target configuration of OpenFOAM
  - Version 4.0
  - Main solvers, parallel processing, paraFoam (ParaView reader)
  - Required later version libraries are built in the project directory (ThirdParty directory).

Procedure of installation

1. Install the necessary rpm packages.
2. Download the source packages and extract them.
3. Edit the configuration/setting files.
4. Compile the tools/libraries.
5. `Allwmake`
6. Test (`foamInstallationTest`)
7. Compile ParaView and its reader.

Install the rpm packages with yum

- Necessary packages:
  - git, gcc-c++, bison, flex, m4, glibc-devel, glibc-devel.i686, zlib-devel

- As a root, type the yum command:

```
$ yum install gcc gcc-c++ bison flex m4 glibc-devel \
  glibc-devel.i686 zlib-devel
```

Directory configuration

Red: make by yourself
Blue: generated later

- Under OpenFOAM-4.0

Allwmake	applications/	doc/	platforms/	tutorials/	
COPYING	README.org	bin/	etc/	src/	wmake/

- Under ThirdParty-4.0

Allclean	cmake-3.2.1/	makeParaView3
Allwmake	etc/	makeQt
AllwmakeLibccmio	gcc-4.8.5/	mpc-1.0.1/
CGAL-4.8/	gmp-5.1.2/	mpfr-3.1.2/
COPYING	makeCGAL	openmpi-1.10.2/
ParaView-5.0.1/	makeCmake	platforms/
README.html	makeGcc	qt-everywhere-opensource-src-4.8.6/
README.org	makeGperftools	scotch_6.0.3/
boost_1_58_0/	makeLLVM	
build/	makeParaView	

Download the source packages and extract them

- OpenFOAM main source code:

```
$ git clone git://github.com/OpenFOAM/OpenFOAM-4.x.git
$ mv OpenFOAM-4.x OpenFOAM-4.0
```

- OpenFOAM ThirdParty:

```
$ git clone git://github.com/OpenFOAM/ThirdParty-4.x.git
$ mv ThirdParty-4.x ThirdParty-4.0
```

- Compiler (GCC):

```
$ cd ThirdParty-4.0.x
$ wget https://ftp.gnu.org/gnu/gcc/gcc-4.8.5/ \
gcc-4.8.5.tar.gz
$ tar zxvf gcc-4.8.5.tar.gz
```

Download the source packages and extract them

- Libraries (gmp, mpfr, mpc)

```
$ wget http://ftp.gnu.org/gnu/gmp/gmp-5.1.2.tar.bz2
$ tar jxvf gmp-5.1.2.tar.bz2
$ wget http://ftp.gnu.org/gnu/mpfr/mpfr-3.1.2.tar.gz
$ tar zxvf mpfr-3.1.2.tar.gz
$ wget http://ftp.gnu.org/gnu/mpc/mpc-1.0.1.tar.gz
$ tar zxvf mpc-1.0.1.tar.gz
```

Download the source packages and extract them

- OpenMPI:

```
$ wget --no-check-certificate https://www.open-mpi.org/ \
software/ompi/v1.10/downloads/openmpi-1.10.2.tar.gz
$ tar zxvf openmpi-1.10.2.tar.gz
```

- Scotch:

```
$ wget https://gforge.inria.fr/frs/download.php/ \
file/34099/scotch_6.0.3.tar.gz
$ tar zxvf scotch_6.0.3.tar.gz
```

- Boost:

```
$ wget http://sourceforge.net/projects/boost/files/ \
boost/1.58.0/boost_1_58_0.tar.gz
$ tar zxvf boost_1_58_0.tar.gz
```

- CGAL:

```
$ wget https://github.com/CGAL/cgal/releases/download/ \
releases/CGAL-4.8/CGAL-4.8.tar.xz
$ tar Jxvf CGAL-4.8.tar.xz
```

Download the source packages and extract them

- Cmake:

```
$ wget --no-check-certificate https://cmake.org/files/ \
v3.2/cmake-3.2.1.tar.gz
$ tar zxvf cmake-3.2.1.tar.gz
```

- Qt (qmake)

```
$ wget --no-check-certificate https://download.qt.io/ \
 archive/qt/4.8/4.8.6/qt-everywhere-opensource-src-4.8.6.tar.gz
$ tar zxvf qt-everywhere-opensource-src-4.8.6.tar.gz
```

If you do not need ParaView, this package is unnecessary.

Edit the configuration/setting files

- Edit OpenFOAM-4.0 environment file

```
$ vi ../OpenFOAM-4.0/etc/bashrc
```

```
export WM_PROJECT=OpenFOAM
export WM_PROJECT_VERSION=4.0

# Please set to the appropriate path if the default is not
correct.
#
[ $BASH_SOURCE ] && \
export FOAM_INST_DIR=${BASH_SOURCE%/*/*/*} || \
# export FOAM_INST_DIR=$HOME/$WM_PROJECT
# export FOAM_INST_DIR=~$WM_PROJECT
export FOAM_INST_DIR=/opt/$WM_PROJECT
# export FOAM_INST_DIR=/usr/local/$WM_PROJECT
```

Edit the configuration/setting files

```
$ vi ../OpenFOAM-4.0/etc/bashrc
```

```
#- Compiler location:
#    WM_COMPILER_TYPE= system | ThirdParty (OpenFOAM)
export WM_COMPILER_TYPE=ThirdParty

#- Label size:
#    WM_LABEL_SIZE = 32 | 64
export WM_LABEL_SIZE=64

#- MPI implementation:
#    WM_MPLIB = SYSTEMOPENMPI | OPENMPI | SYSTEMMPI | \
MPICH | MPICH-GM | HPMPI
#              | MPI | QSMPI | SGIMPI
export WM_MPLIB=OPENMPI
```

Edit the configuration/setting files

- Edit OpenFOAM-4.0 compiler/libraries setting file

```
$ vi ../OpenFOAM-4.0/etc/config.sh/compiler
```

```
case "$WM_COMPILER_TYPE" in
OpenFOAM | ThirdParty)

# Default versions of GMP, MPFR and MPC, override as
necessary
    gmp_version=gmp-5.1.2
    mpfr_version=mpfr-3.1.2
    mpc_version=mpc-1.0.1

    case "$WM_COMPILER" in
    Gcc | Gcc48)
        gcc_version=gcc-4.8.5
```

Edit the configuration/setting files

- Edit `ThirdParty-4.0/makeCGAL`

```
$ vi makeCGAL
```

```
# Get CGAL, boost and gmp/mpfr versions
. $WM_PROJECT_DIR/etc/config.sh/functions
_foamEval SOURCE_CGAL_VERSIONS_ONLY=yes \
          $($WM_PROJECT_DIR/bin/foamEtcFile config.sh/
CGAL)
_foamSource $($WM_PROJECT_DIR/bin/foamEtcFile config.sh/
compiler)

set -x

cgalPACKAGE=${cgal_version:-CGAL-4.8}
boostPACKAGE=boost_1_58_0
gmpPACKAGE=gmp-5.1.2
mpfrPACKAGE=mpfr-3.1.2
```

Edit the configuration/setting files

- Edit CGAL

```
$ vi ../OpenFOAM-4.0/etc/config.sh/CGAL
```

```
# Description
#     Setup file for CGAL (& boost) include/libraries.
#     Sourced from OpenFOAM-<VERSION>/etc/bashrc
#------------------------------------------------------------------------------

boost_version=boost_1_58_0
#cgal_version=cgal-system
cgal_version=CGAL-4.8
```

Compile the tools/libraries

- Reload the OpenFOAM environment file and compile GCC:

```
$ . /opt/OpenFOAM/OpenFOAM-4.0/etc/bashrc
```

```
Warning in /opt/OpenFOAM/OpenFOAM-4.0/etc/config.sh/
settings:
    Cannot find /opt/OpenFOAM/ThirdParty-4.0/platforms/
linux64/gcc-4.8.5 installation.
    Please install this compiler version or if you wish to
use the system compiler,
    change the 'WM_COMPILER_TYPE' setting to 'system'
```

```
$ ./makeGcc
```

Compile the tools/libraries

- Build CMake and execute `Allwmake`:

```
$ ./makeCmake
$ . /opt/OpenFOAM/OpenFOAM-4.0/etc/bashrc
$ ./Allwmake
```

- Build OpenFOAM:

```
$ . /opt/OpenFOAM/OpenFOAM-4.0/etc/bashrc
$ cd /opt/OpenFOAM/OpenFOAM-4.0/
$ ./Allwmake –j4 # -j4 depending on your machine
```

- After the compiling, check it:

```
$ . /opt/OpenFOAM/OpenFOAM-4.0/etc/bashrc
$ foamInstallationTest
```

Compile ParaView and its reader

- Build Qt
  - Shell script for Qt compiling is not included in ThirdParty 4.0.

```
$ cd /opt/OpenFOAM/ThirdParty-4.0
$ cp /opt/OpenFOAM/ThirdParty-3.0.x/makeQt ./
$ cp /opt/OpenFOAM/ThirdParty-3.0.x/etc/tools/QtFunctions \
  ./etc/tools/
$ vi makeQt
```

```
# Description
#     Build script for qt-everywhere-opensource-src
#--------------------------------------------------------
qtVERSION=4.8.6
qtTYPE=qt-everywhere-opensource-src
```

```
$ ./makeQt
```

Compile ParaView and its reader

- Compile ParaView 5.0.1

```
$ vi makeParaView
```

```
# Set the path to the Qt-4.5 (or later) qmake if the \
system Qt is older
QMAKE_PATH="/opt/OpenFOAM/ThirdParty-4.0/ \
            platforms/linux64Gcc/qt-4.8.6/bin"

# Set the path to cmake
CMAKE_PATH="/opt/OpenFOAM/ThirdParty-4.0/ \
            platforms/linux64Gcc/cmake-3.2.1/bin"
```

```
$ ./makeParaView
```

Compile ParaView and its reader

- Compile ParaView reader

```
$ cd $FOAM_UTILITIES/postProcessing/graphics/PVReaders
$ ./Allwclean
$ ./Allwmake
```

- After the compiling, check a tutorial run (cavity):

```
$ . /opt/OpenFOAM/OpenFOAM-4.0/etc/bashrc
$ mkdir -p $FOAM_RUN
$ run
$ cp -r $FOAM_TUTORIALS/incompressible/icoFoam/ \
 cavity/cavity ./
$ cd cavity
$ blockMesh
$ icoFoam
$ paraFoam
```

Here we go! Enjoy!