**Reference: https://openfoam.org/download/4-0-source/**

**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://dl.openfoam.org/source/4-0**](http://dl.openfoam.org/source/4-0)
* [**http://dl.openfoam.org/third-party/4-0**](http://dl.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:

* Ubuntu 14.04 (**trusty**), 16.04 (**xenial**)
* OpenSuSE Tumbleweed (June 2016)
* SuSE Linux Enterprise Server (SLES) 12.0

## Unpacking the Sources

The user should choose a directory location to unpack these files, which will become the installation directory of OpenFOAM. If the installation is for a single user only, or if the user does not have root access to the machine, we would recommend the installation directory is **$HOME/OpenFOAM** (i.e. a directory **OpenFOAM** in the user’s home directory). If the installer has root permissions and the installation is for more than one user, one of the ‘standard’ locations can be used, e.g. **/usr/local/OpenFOAM**, **/opt/OpenFOAM**, or just **/opt**.

After the installation directory is chosen (and, if necessary, created), open a terminal window, change into the installation directory and download an unpack the source and third-party archives by copying and pasting the following:

**wget -O - http://dl.openfoam.org/source/4-0 | tar xvz**

**wget -O - http://dl.openfoam.org/third-party/4-0 | tar xvz**

The files unpack to produce directories **OpenFOAM-4.x-version-4.0** and **ThirdParty-4.x-version-4.0**, which need to be renamed as follows:

**mv OpenFOAM-4.x-version-4.0 OpenFOAM-4.0**

**mv ThirdParty-4.x-version-4.0 ThirdParty-4.0**

## Software for Compilation

Follow the instructions for installing [**Software for Compilation**](https://openfoam.org/download/source/software-for-compilation/) of OpenFOAM for your platform.

# **OpenFOAM Repo: 1. Software for Compilation**

The following supporting software is required to download, compile and run OpenFOAM from one of the source repositories.

* Compiler: **either** [**GCC**](http://gcc.gnu.org/) version 4.5 or above; **or** [**LLVM Clang**](http://clang.llvm.org/) version 3.6 or above; **or** [**Intel ICC**](https://wikipedia.org/wiki/Intel_C%2B%2B_Compiler) version 15.0.3  or above. GCC is most commonly available and the version can be checked by typing

**gcc --version**

* [**FLEX**](http://flex.sourceforge.net/) fast lexical analyser, used by OpenFOAM for reading files of third-party formats.
* [**cmake**](http://cmake.org/) build software for compiling ***[ParaView](http://www.paraview.org/" \t "_blank)***, the third-party visualisation toolkit
* [**QT**](https://wikipedia.org/wiki/Qt_%28software%29) cross-platform application software, version 4.8 or above, used by ParaView.
* [**Git distributed version control**](http://git-scm.com/) software used for the OpenFOAM source repositories.
* [**OpenMPI**](http://www.open-mpi.org/) message passing interface for parallel computation.

#### Installing dependent packages on Ubuntu (versions 14.04 or above)

* Install general packages for OpenFOAM

**sudo apt-get install build-essential flex bison git-core cmake zlib1g-dev libboost-system-dev libboost-thread-dev libopenmpi-dev openmpi-bin gnuplot libreadline-dev libncurses-dev libxt-dev**

* Install packages for ParaView

**sudo apt-get install qt4-dev-tools libqt4-dev libqt4-opengl-dev freeglut3-dev libqtwebkit-dev**

#### Installing on SuSE (OpenSuSE/SLES v12 or above, or Tumbleweed)

## Setting Environment Variables

The environment variable settings are contained in files in an **OpenFOAM-4.0/etc** directory in the OpenFOAM release. e.g. **$HOME/OpenFOAM/OpenFOAM-4.0/etc** for the case where the installation is in **$HOME/OpenFOAM**:

The user should configure their system with permanent settings to source the environment. **If OpenFOAM is installed in the $HOME/OpenFOAM directory**, the user should:

##### EITHER

if running **bash** or **ksh** (if in doubt type **echo $SHELL**), source the **etc/bashrc** file by adding the following line to the end of your **$HOME/.bashrc** file:

**source $HOME/OpenFOAM/OpenFOAM-4.0/etc/bashrc**

then type “**source $HOME/.bashrc**” in the current terminal window

##### OR

if running **tcsh** or **csh**, source the **etc/cshrc** file by adding the following line to the end of your **$HOME/.cshrc** file:

**source $HOME/OpenFOAM/OpenFOAM-4.0/etc/cshrc**

then type “**source $HOME/.cshrc**” in the current terminal window.

**When OpenFOAM is installed in an alternative directory**, e.g. **/opt**, the user should substitute **$HOME/OpenFOAM** with the relevant installation location in the lines above.

## Installing Third-Party Software

Follow the instructions for installing [**Third Party Software**](https://openfoam.org/download/source/third-party-software/) on your platform.

OpenFOAM relies some third-party software packages (in addition to OpenMPI) for some important tasks:

* [**Scotch and PT-Scotch**](https://www.labri.fr/perso/pelegrin/scotch/) for domain decomposition for parallel running (recommended/essential).
* [**ParaView**](http://www.paraview.org/) visualization application (essential, without an alternative, compatible visualisation tool)
* [**CGAL Computational Geometry Algorithms Library**](http://www.cgal.org/) used by the experimental mesher, foamyHexMesh (not essential).

The **ThirdParty** repository contains scripts for compiling these software packages.  The **README** file contains some information about the compiling the software, but we will provide step-by-step instructions for compiling Scotch/PT-Scotch and ParaView below.

#### Installing Scotch/PT-Scotch

OpenFOAM requires Scotch/PT-Scotch version 6 and higher since it includes a fix to allow both the **libscotch** and **libptscotch** libraries to be linked to the same application.  Version 6 can be installed as a package for most recent versions of Linux, but not Ubuntu, since the Debian package maintainers have failed to upgrade to version 6 since its release in December 2012.  Note that the [**packaged version of OpenFOAM-dev for Ubuntu**](https://openfoam.org/download/dev-ubuntu/) contains the object libraries for Scotch/PT-Scotch v6.0.3 built from **ThirdParty-dev**, as described next.

If a packaged version is not available, Scotch/PT-Scotch v6.0.3 can be installed simply by going into the **ThirdParty** directory (e.g. **ThirdParty-dev**) where the sources can be compiled by running the **Allwmake** script.

**./Allwmake**

Note that the **Allwmake** script is set up to be able to compile other packages, e.g. GCAL, but these packages are ignored unless the source code is downloaded to the **ThirdParty**directory.

#### Installing ParaView

Patched versions of the ParaView source code were once provided for ParaView v5.0.1.  With the current supported version of ParaView (v.5.4.0), the source code is downloaded automatically and compiles without modification.  ParaView is compiled by running the **makeParaView** script, i.e.

**./makeParaView**

Expect **ParaView to take a long time to compile**, typically several hours on a desktop computer. Following compilation, **update the environment** by sourcing the .**bashrc** (or .**cshrc**) file as described in [**3. Setting the Environment**](https://openfoam.org/download/source/setting-environment/) or by typing

**wmRefresh**

## Compiling OpenFOAM

With the Third Party software installed and environment updated, compile OpenFOAM by going into the **OpenFOAM-4.0** directory and executing the **Allwmake** script.  Type **Allwmake -help** for options, but the 2 main choices are to compile in serial with

**./Allwmake**

or compile in parallel with all available cores/hyperthreads with:

**./Allwmake -j**

Serial compilation takes several hours, whereas compilation on 8 cores/threads should take approximately one hour, possibly less, depending on the processor.

## Getting Started

Create a project directory within the **$HOME/OpenFOAM** directory named **<USER>-4.0**(e.g. **chris-4.0** for user **chris** and OpenFOAM version 4.0) and create a directory named **run**within it, e.g. by typing:

**mkdir -p $FOAM\_RUN**

Copy across the backward facing step example, generate the mesh with blockMesh and run the steady flow, incompressible solver simpleFoam

**cd $FOAM\_RUN**

**cp -r $FOAM\_TUTORIALS/incompressible/simpleFoam/pitzDaily .**

**cd pitzDaily**

**blockMesh**

**simpleFoam**

**paraFoam**

Refer to the **[OpenFOAM User Guide](http://cfd.direct/openfoam/user-guide)** to get started.

## Reporting Bugs in OpenFOAM

We appreciate that bugs in OpenFOAM are reported so we can fix them. Please refer to the **[OpenFOAM bugs](https://openfoam.org/bugs)** pages to report bugs.