

# Installing OpenFOAM®

- To install OpenFOAM®, you will need a working Linux distribution.
- The easiest way to install OpenFOAM® is by downloading a precompiled binary (Ubuntu Deb Pack, Red Hat RPM Pack and Suse RPM pack are available).
- By the way, you will need internet access and administrative privileges.
- To install OpenFOAM®, just follow the instructions given in <http://www.openfoam.org/>
- If for any reason the instructions given in <http://www.openfoam.org/> do not work for you, we are going to give you our instructions.
- The instructions are exactly the same as the one you will find in <http://www.openfoam.org/>, we only added a few tips and additional comments.

# Installing OpenFOAM®

- The way we do it, is by compiling the latest source code release from the Git repository.
- Those willing to compile the source code can receive regular updates by downloading the Git repository distribution of OpenFOAM®. Known as 3.0.x, this version is managed and updated daily by the OpenFOAM® Foundation and can be conveniently **pulled** (updated) using the Git revision control system.
- It is possible to do a native installation of OpenFOAM® in Windows and Mac OS X, but we will not discuss it.
- If you work under Windows or Mac OS X, and do not want to go through the hustle and bustle of partitioning your hard drive to install a Linux distribution, you can install a virtual machine.

# Installing OpenFOAM®

- If you want to install a virtual machine, you can check these versions:
  - VirtualBox (<https://www.virtualbox.org/>). Open source.
  - VMware Workstation Player ([https://my.vmware.com/en/web/vmware/free#desktop\\_end\\_user\\_computing/vmware\\_workstation\\_player/12\\_0](https://my.vmware.com/en/web/vmware/free#desktop_end_user_computing/vmware_workstation_player/12_0)). Free.
- We recommend VMware Workstation Player.
- After installing the virtual machine, simply install any Linux version.
- Then install OpenFOAM®.
- The following instructions should work with Linux OpenSUSE 42.1 (and maybe with older versions).
- During this installation, we are assuming that you are using bash shell and that you have the newest gcc compiler and libraries.

# Installing OpenFOAM®



- **We tested these instructions on 5/7/2016 and using **OpenSUSE** 42.1**
- **We strongly suggest you not to copy and paste these instructions in the terminal, as it might experience some difficulties interpreting the end of line character.**

# OpenFOAM® installation – Linux option

## Installation prerequisites

Remember, in these installation instructions we are assuming that you have **OpenSUSE 42.1** (or a newer version), you are using bash shell and that you have the newest gcc compiler and system libraries.



- Using the graphical interface (the easy way), launch yast and search for the following packages:
  - git, cgal-devel, gmp-devel, python-devel, lapack-devel, gsl-devel, cmake, boost-devel, gnuplot, mpfr-devel, openmpi-devel, glu-devel, libQtWebKit-devel, libqt4-devel, qt4-assistant-adp-devel, qt4-x11-tools, devel\_C\_C++
- You will need administrative privileges.

# OpenFOAM® installation – Linux option

## Installation prerequisites

Remember, in these installation instructions we are assuming that you have **OpenSUSE 42.1** (or a newer version), you are using bash shell and that you have the newest gcc compiler and system libraries.



- Using the terminal (the hard way) :
  - `$> zypper install -t pattern devel_C_C++`
  - `$> zypper install git cgal-devel gmp-devel python-devel lapack-devel gsl-devel metis-devel`
  - `$> zypper install cmake boost-devel gnuplot mpfr-devel openmpi-devel glu-devel`
  - `$> zypper install libQtWebKit-devel libqt4-devel qt4-assistant-adp-devel qt4-x11-tools`
- You will need administrative privileges.

# OpenFOAM® installation – Linux option

## Downloading the source code

- Download OpenFOAM 3.0.x source code. In the terminal type:
  - `$> cd $HOME`
  - `$> mkdir OpenFOAM`
  - `$> cd OpenFOAM`
  - `$> git clone git://github.com/OpenFOAM/OpenFOAM-3.0.x.git`
- If the git protocol does not work, it means that your computer is behind a firewall that is blocking the relevant TCP port (9418). In such a case proceed as follows:
  - `git clone http://github.com/OpenFOAM/OpenFOAM-3.0.x.git`

# OpenFOAM® installation – Linux option

## Downloading the source code

- As soon as you finish downloading OpenFOAM-3.0.x, do the following:
  - `$> cd OpenFOAM-3.0.x`
  - `$> git pull`  
This will update the source code installation which is already up-to-date. This is done to test that git pull is working.
- Now download the ThirdParty package from the terminal:
  - `$> cd $WM_PROJECT`
  - `$> wget -c http://downloads.sourceforge.net/foam/ThirdParty-3.0.1.tgz?use_mirror=mesh -O ThirdParty-3.0.1.tgz`
- Extract the ThirdParty package in `$WM_PROJECT` and rename it as follows:
  - `$> tar xzf ThirdParty-3.0.1.tgz`
  - `$> mv ThirdParty-3.0.1 ThirdParty-3.0.x`



# OpenFOAM® installation – Linux option

## Environment variables

- Let's setup the environment variables.
- The environment variable settings are contained in the `$HOME/OpenFOAM/OpenFOAM-3.0.x/etc` directory.
- If you are running bash or ksh (if in doubt type `echo $SHELL`), source the `$WM_PROJECT_DIR/etc/bashrc` file by adding the following line to your `$HOME/.bashrc` file:
  - `source $HOME/OpenFOAM/OpenFOAM-3.0.x/etc/bashrc`
- Then type in the current terminal window:
  - `$> source $HOME/.bashrc`

# OpenFOAM® installation – Linux option

## Environment variables

- If you are planning to use the **MPI** distribution that comes with OpenFOAM®, in the file `$WM_PROJECT_DIR/etc/bashrc` look for the entry **export WM\_MPLIB** and made the following modification
  - **export WM\_MPLIB=OPENMPI**
- If you are planning to use the system MPI distribution made the following modification (default option),
  - **export WM\_MPLIB=SYSTEMOPENMPI**
- You will need to install the system **MPI** distribution.
- If your **MPI** distribution is not installed in the default location, add the **MPI** path entry at the very end of your `.bashrc` file.

# OpenFOAM® installation – Linux option

## Checking your system before compilation

- After installing the source code and all the dependencies, you need to check if your system is ready to build the sources, to do so execute in the terminal the `foamSystemCheck` script, in the terminal type:
  - `$> foamSystemCheck`
- If any critical software is missing or needs updating to a newer version, install the required software before proceeding to the building stage.

# OpenFOAM® installation – Linux option

## Building OpenFOAM-3.0.x

- If the previous check passed, we can now compile OpenFOAM-3.0.x.
- The compilation process can be quite lengthy, if your computer has more than one processor you can compile in parallel by setting the `WM_NCOMPPROCS` environment variable, in the terminal type:

- `$> export WM_NCOMPPROCS=NP`

where `NP` is the number of processors you want to use.

# OpenFOAM® installation – Linux option

## Building OpenFOAM-3.0.x

- Now go to the OpenFOAM-3.0.x installation directory `$WM_PROJECT_DIR` and execute the build script *Allwmake*, in the terminal type:
  - `$> cd $WM_PROJECT_DIR`
  - `$> ./Allwmake`
- In principle this will build everything.
- After compiling OpenFOAM-3.0.x, we need to compile the Paraview module and the PV4FoamReader module.

# OpenFOAM® installation – Linux option

## Building OpenFOAM-3.0.x

- At this point and if you did not get any error during the building process, you should have a fully working installation of OpenFOAM-3.0.x
- You can start to use OpenFOAM but you will not be able to use paraFoam (a Paraview fork).
- Paraview is the third-party software used for post-processing in OpenFOAM®. Its compilation is automated using the script *makeParaView4* located in the **ThirdParty-3.0.x** directory.
- To compile Paraview, execute the following commands:
  - `$> cd $WM_THIRD_PARTY_DIR`
  - `4> ./makeParaView4`

# OpenFOAM® installation – Linux option

## Building OpenFOAM-3.0.x

- Then you need to compile PV4blockMeshReader and the PV4FoamReader plugins. From the terminal:
  - `$> cd $FOAM_UTILITIES/postProcessing/graphics/PV4Readers`
  - `$> wmSET`
  - `./Allwclean`
  - `./Allwmake`
- At this point and if you did not get any error during the building process, you should have a fully working installation of both OpenFOAM-3.0.x and Paraview/paraFoam.

# OpenFOAM® installation – Linux option

- To check your installation, execute the *foamInstallationTest* script from the terminal.
- If no problems are reported,

**Congratulations, you have a fully working installation of OpenFOAM-3.0.x and Paraview**



# OpenFOAM® installation – Linux option

## Testing you installation

- Create an user directory in the `$HOME/OpenFOAM` directory, in the terminal type:
  - `$> mkdir -p $FOAM_RUN`
- Then from the terminal:
  - `$> cd $FOAM_RUN`
  - `$> pwd`
- Then you should see your current working directory, which should be:
  - `$HOME/OpenFOAM/USERNAME-3.0.x/run`

# OpenFOAM® installation – Linux option

## Testing you installation

- Now copy OpenFOAM® tutorials to your user directory. In the terminal type:
  - `$> cp -r $FOAM_TUTORIALS $FOAM_RUN`
- Now go to the tutorials directory. In the terminal type:
  - `$> cd $FOAM_RUN/tutorials`
  - `$> ls -al`  
(just to take a look at the directory organization and to verify that the information was copied)

## Testing you installation

- Run the first example case of incompressible flow in a lid-driven square cavity:
  - `$> cd $FOAM_RUN/tutorials/incompressible/icoFoam/cavity`
  - `$> blockMesh`
  - `$> icoFoam`
  - `$> paraFoam`

**Enjoy and happy CFD**

# OpenFOAM® installation – Linux option

## Updating your OpenFOAM-3.0.x source code installation

- To update OpenFOAM 3.0.x source code installation, from the terminal:
  - `$> cd $WM_PROJECT_DIR`
  - `$> git pull`
  - `$> ./Allwmake`
- When you recompile the updated source code installation, you do not recompile the whole installation, you only compile the updated files. If you want to compile a clean installation after an update, from the terminal:
  - `$> cd $WM_PROJECT_DIR`
  - `$> git pull`
  - `$> ./wcleanAll`
  - `$> ./Allwmake`

# OpenFOAM® installation – Linux option

## To compile OpenFOAM-3.0.x doxygen documentation

- **This step is optional.**
- To compile Doxygen documentation, from the terminal:
  - `$> cd $WM_PROJECT_DIR`
  - `$> ./Allwmake doc`

**Note: You will need to install doxygen and graphviz/dot**

- Have in mind that compiling the Doxygen documentation is time consuming and it will take a lot hard drive space, so it is easier to access the information online <http://cpp.openfoam.org/v3/>

# OpenFOAM® installation – Virtual machine option

- **Installing OpenFOAM® on Windows and MacOS – Virtual machine**
  - If you work under Windows and/or Mac OS X, and do not want to go through the hustle and bustle of partitioning your hard drive to install a Linux distribution, you can install a virtual machine.
  - If you are looking for a virtual machine, you can download VirtualBox (<https://www.virtualbox.org/>) or VMware Workstation Player ([https://my.vmware.com/en/web/vmware/free#desktop\\_end\\_user\\_computing/vmware\\_workstation\\_player/12\\_0](https://my.vmware.com/en/web/vmware/free#desktop_end_user_computing/vmware_workstation_player/12_0)). Mac and Windows versions are available and they are open source/free.
  - After installing the virtual machine, simply install the guest operating system, namely, the Linux version of your preference.
  - If you want to know it, we use **OpenSUSE**.
  - To install OpenFOAM®, just follow the instructions given in <http://www.openfoam.org/> or the previous instructions.

- **Installing OpenFOAM® on Windows and MacOS – Virtual machine**

- To avoid compatibility issues during the course, we highly advise you to install our custom made Linux version. It is a fully working Linux **OpenSUSE 42.1** version (with KDE 4), that you can modify or update to fit your needs.

- You can download the virtual machine distribution or an ISO distribution from this link

<https://susestudio.com/a/XrxjaO/joel-s-opensuse-leap-42-1-kde-4-desktop>

- We regularly update the distributions.

- **Installing OpenFOAM® on Windows and MacOS – Virtual machine**
  - This Linux version was created with CFD applications in mind for CFD users. It comes with many CFD, CAD, mesh generation, visualization, numerical libraries, optimization, plotting applications, and text editors pre-installed. It also comes with all the compilers and libraries necessary to compile most of the applications.
  - This is a light Linux distribution and is already configured to seamlessly run OpenFOAM® and all applications to be used during the course.
  - To install the virtual machine you will need at least 50 gigs of hard drive space and 2 gigs of RAM memory.
  - If you want to exploit Linux to its maximum potential, it is better to install a full distribution.



- **Installing OpenFOAM® on Windows and MacOS – Virtual machine**
  - If you choose to install our Linux distribution, extract the image in a directory of your convenience and launch VirtualBox or VMware Workstation Player.
  - In VirtualBox or VMware Workstation Player create a new virtual machine, follow the instructions and install the Linux image you just extracted (the .vmdk file).
  - After installing the virtual machine, do not forget to change the settings according to your hardware. Also, remember to enable 3D acceleration.
  - Remember, to install the virtual machine you will need at least 50 gigs of hard drive space and 2 gigs of RAM memory.

- **Installing OpenFOAM® on Windows and MacOS – Virtual machine**
  - After finishing the installation, run the virtual machine (the first time it boots it will take some time).
  - If you are using VirtualBox you will need to install the guest additions tools (you will find them in the menu bar under Devices).
  - Reboot and voila you are done.
  - There are many applications preinstalled, just to name a few: OpenFOAM®, salome, dakota, pyfoam, swak4foam, blender, fipy, gmsh, gnuplot, helyx-os, maxima, openscad, openvsp, octave, paraview, Python, valgrind, xflr5.
  - By the way, we have tested the virtual machine with vmware fusion, vmware player, parallels and VirtualBox.

- **Installing OpenFOAM® on Windows and MacOS – Virtual machine**

- After login in, you will find the OpenFOAM® installation in the following directory:

`/home/cfd/OpenFOAM`

- The default user is **cf**d.
- All the environment variables have been already setup.
- Add this point you can start enjoying OpenFOAM® (and **OpenSUSE** Linux distribution).
- By the way, do not forget to check the `.bashrc` file to know all the aliases and environment variables defined.

# OpenFOAM® installation – Virtual machine option

- **Installing OpenFOAM® on Windows and MacOS – Virtual machine**

- Remember:

## Accounts

Default User: `cfid`

Password: `cfid`

Root User: `root`

Password: `linux`



- **Installing OpenFOAM® on Windows and MacOS – Virtual machine**
  - If you choose to install another Linux distribution, just follow the default installation instructions.
  - We recommend you **OpenSUSE** Linux distribution, but it is up to you.
  - After installing the virtual machine, do not forget to change the settings according to your hardware. Also, do not forget to enable 3D acceleration.
  - By the way, during the installation of the virtual machine you will need the installation DVD or the image of the Linux distribution you chose.

- **Installing OpenFOAM® on Windows and MacOS – Virtual machine**
  - Also, you will need to manually set the size of your virtual hard drive.
  - If you are planning to do a minimum Linux installation and doing some small runs using OpenFOAM®, we recommend you to use 20 gigs.
  - In any case, you can set any size you want according to your hardware limitations.

- **Installing OpenFOAM® on Windows and MacOS – Virtual machine**
  - After finishing the installation of your Linux distribution, run the virtual machine (the first time it boots it will take some time) and if it is needed, install the guest additions tools.
  - Reboot and voila you are done with the Linux distribution installation.
  - Now proceed with the OpenFOAM® installation.
  - As you will be doing a clean OpenFOAM® installation, just follow the instructions that you will find in <http://www.openfoam.org/>.
  - If for any reason the instructions given in <http://www.openfoam.org/> do not work for you, try to follow the previous instructions.