Disclaimer

"This offering is not approved or endorsed by OpenCFD Limited, the producer of the OpenFOAM software and owner of the OPENFOAM® and OpenCFD® trade marks."

Introductory OpenFOAM® Course

From 8th to 12th July, 2013

University of Genoa, DICCA

Dipartimento di **I**ngegneria **C**ivile, **C**himica e **A**mbientale

UNIVERSITÀ DEGLI STUDI
DI GENOVA

Your Lecturer

Joel GUERRERO

joel.guerrero@unige.it

UNIVERSITÀ DEGLI STUDI
DI GENOVA

guerrero@wolfdynamics.com

wolf dynamics

Acknowledgements

These slides and the tutorials presented are based upon personal experience, OpenFOAM® source code, OpenFOAM® user guide, OpenFOAM® programmer's guide, and presentations from previous OpenFOAM® training sessions and OpenFOAM® workshops.

We gratefully acknowledge the following OpenFOAM® users for their consent to use their material:

- Hrvoje Jasak. Wikki Ltd.
- Hakan Nilsson. Department of Applied Mechanics, Chalmers University of Technology.
- Eric Paterson. Applied Research Laboratory Professor of Mechanical Engineering, Pennsylvania State University.

Today's lecture

1. OpenFOAM® installation
2. Additional utilities installation
3. CAD and Meshing applications

Today's lecture

1. **OpenFOAM® installation**
2. Additional utilities installation
3. CAD and Meshing applications

Today's lecture

Before we begin, I want to let you know that

I tested these instructions on 08/05/2013 and using OpenSUSE 11.4 and 12.3.

OpenFOAM® installation

- The easiest way to install OpenFOAM® is by downloading a precompiled binary. Ubuntu Deb Pack and Suse RPM pack are available.
- To install OpenFOAM® precompiled binaries, just follow the instructions given in http://www.openfoam.org/download/ubuntu.php or http://www.openfoam.org/download/suse.php.
- It is possible to natively install OpenFOAM® in Windows and MacOS, but the installation process is quite complicated and I will not address it.
- However, if you still want to use OpenFOAM® in Windows and/or MacOS, the easiest way is by using a virtual machine (such as VirtualBox https://www.virtualbox.org/). I will briefly address this issue.

OpenFOAM® installation

- For Windows and Mac users, you can also check the following websites:

http://www.paratools.com/OpenFOAM

http://www.ifd.mavt.ethz.ch/education/Lectures/openfoam/of-macosx

http://sourceforge.net/projects/macopenfoam/files/

http://openfoamwiki.net/index.php/Howto_install_OpenFOAM_v21_Mac

http://openfoamwiki.net/index.php/Tip_Using_Cygwin_for_cross-compiling_OpenFOAM

OpenFOAM® installation

- The ideal way is to compile the latest source code release from the Git repository.

- Those users willing to compile the source code can receive regular updates by downloading the Git repository distribution of OpenFOAM®. Known as 2.2.x, this version is managed and updated daily by the OpenFOAM® Foundation and can be conveniently **pulled** (updated) using the Git revision control system.

- To install OpenFOAM 2.2.x you will need internet access and administrative privileges.

OpenFOAM® installation

- During this installation, I am assuming that you have OpenSUSE 11.4 (or a newer version), you are using bash shell and that you have the newest gcc compiler and libraries.
- If during the installation you get an error referring to missing libraries or header files, find out what library you are missing and install it and continue with the installation.

Installing OpenFOAM® under Windows or MacOS

The poor man's way

OpenFOAM® installation under Windows or MacOS

- OpenFOAM® can be natively installed under Windows and MacOS, however the installation process is quite complex and I will not address it.

- You can also repartition your hard drive and install Linux in your computer. In this way you will have a dedicated partition for Windows and/or MacOS and a dedicated partition for Linux. This installation process is lengthy and you might want to backup all your data before.

- If you decide to repartition your hard drive, I recommend you OpenSUSE Linux distribution.

- After repartitioning your hard drive and installing Linux, you can install OpenFOAM® by following the instructions that I will give later (basically the same instructions you will find in http://www.openfoam.org/).

- For those not willing to partition their hard drives and install Linux on their laptops, you can install a virtual machine by using VirtualBox (https://www.virtualbox.org/). Mac and Windows versions are available.

- By the way, did I forget to tell you that VirtualBox is open source?.

OpenFOAM® installation under Windows or MacOS

- First at all, install VirtualBox. Windows and MacOS versions are available.

- After installing VirtualBox (in Windows or MacOS), you will need to install your guest operating system (the Linux distribution of your preference).

- I highly recommend to install GeekoCFD.

"GeekoCFD is a live distribution based on OpenSUSE – 64 bits, whose purpose is to provide easy and immediate access to open-source Computational Fluid Dynamics tools. It includes cantor, gmsh, grace, gsl, wxMaxima, Octave with an almost complete selection of octave-forge packages, OpenFOAM, Paraview, pyFoam, R. Additionally, gcc, DDD, Eclipse Helios with CDT, Emacs, git, kate, vim and a complete openSUSE KDE installation are provided, including Gimp, LibreOffice and Blender."

- If you choose to install GeekoCFD, first proceed to download it (just google it). Then extract the image in a directory of your convenience and launch VirtualBox. Remember, do not erase the image.

OpenFOAM® installation under Windows or MacOS

- In VirtualBox create a new virtual machine, follow the instructions and install the GeekoCFD image you just extracted (.vmdk).

- While installing your virtual machine, do not forget to change the settings according to your hardware. Also, remember to enable 3D acceleration.

- When installing GeekoCFD; by default the drive size is set to 15 gigs and it is dynamically allocated (it will use space on your physical hard drive as it fills up, up to a maximum size).

- After finishing the installation, run the virtual machine (the first time it boots it will take some time) and install the guest additions tools (you will find them in the menu bar under Devices).

- Reboot and voila you are done.

OpenFOAM® installation under Windows or MacOS

- After login in, you will find the OpenFOAM® installation in the directory /opt.
- The default user is geeko. All the environments variables have been already setup.
- Add this point you can start enjoying OpenFOAM® (and OpenSUSE Linux distribution).
- Remember:

Accounts

Default User:	geeko
Password:	geekoPassword
Root User:	root
Password:	rootPassword

OpenFOAM® installation under Windows or MacOS

- If you choose to install another Linux distribution, just follow the default installation instructions.

- I recommend you OpenSUSE Linux distribution, but it is up to you.

- While installing your 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 image of the Linux distribution you chose.

- 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®, I recommend you to use 15 gigs. In any case, you can set any size you want according to your hardware limitations.

OpenFOAM® installation under Windows or MacOS

- After finishing the Linux distribution installation, run the virtual machine (the first time it boots it will take some time) and install the guest additions tools (you will find them in the menu bar under Devices).

- 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 I will give in the next slides.

OpenFOAM-2.2.x installation

OpenFOAM® installation (version 2.2.x)

ATTENTION

If for any reason the installation instructions given in http://www.openfoam.org/ do not work for you, try to follow mine instructions, they work fine in OpenSUSE 11.4 or newer version.

OpenFOAM® installation (version 2.2.x)

ATTENTION

I strongly suggest you not to copy and paste these instructions in the terminal, as it may experience some difficulties interpreting the characters.

Installation prerequisites

Remember, in these installation instructions I am assuming that you have OpenSUSE 11.4 (or a newer version), you are using bash shell and that you have the newest gcc compiler and system libraries.

- Using the graphical interface, launch yast and search for the following packages:
  - git, cmake, libqt4-devel, gnuplot, devel_C_C++

- Using a terminal (the hard way), type:
  - su root_password
  - zypper install git-core
  - zypper install -t pattern devel_C_C++
  - zypper install cmake libqt4-devel gnuplot

OpenFOAM® installation (version 2.2.x)

Downloading the source code

- Download OpenFOAM 2.2.x source code using the terminal:
  - cd **$HOME**
  - mkdir **OpenFOAM**
  - cd **OpenFOAM**
  - git clone git://github.com/OpenFOAM/OpenFOAM-2.2.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 do the following:

- git clone http://github.com/OpenFOAM/OpenFOAM-2.2.x.git

As soon as you finish downloading OpenFOAM-2.2.x, do the following:

- cd **OpenFOAM-2.2.x**
- git pull (this will update the source code installation)

OpenFOAM® installation (version 2.2.x)

Now, using firefox or your preferred web browser, download the ThirdParty package from:

http://downloads.sourceforge.net/foam/ThirdParty-2.2.0.tgz?use_mirror=mesh

http://downloads.sourceforge.net/foam/ThirdParty-2.2.0.tgz?use_mirror=mesh

This package contains additional software libraries and applications required to build and run OpenFOAM-2.2.x. In the OpenFOAM® installation root directory (**$WM_PROJECT**) unpack the ThirdParty-2.2.0.tgz file and rename the unpacked directory to **ThirdParty-2.2.x:**

- tar xzf ThirdParty-2.2.0.tgz
- mv **ThirdParty-2.2.0 ThirdParty-2.2.x**

OpenFOAM® installation (version 2.2.x)

You can also download the ThirdParty package from the terminal:

- cd **$WM_PROJECT**
- wget -c http://downloads.sourceforge.net/foam/ThirdParty-2.2.0.tgz?use_mirror=mesh -O ThirdParty-2.2.0.tgz **(this is one line)**

wget -c http://downloads.sourceforge.net/foam/ThirdParty-2.2.0.tgz?use_mirror=mesh -O ThirdParty-2.2.0.tgz

Extract the ThirdParty package in **$WM_PROJECT** and rename it as follows:

- tar xzf ThirdParty-2.2.0.tgz
- mv **ThirdParty-2.2.0 ThirdParty-2.2.x**

Environment variables

Let us now setup the environment variables.

The environment variable settings are contained in **OpenFOAM-2.2.x/etc** directory, if you followed the instructions it should be in:

- **$HOME/OpenFOAM/OpenFOAM-2.2.x/etc**

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 the end of your **$HOME/**.bashrc file:

- source **$HOME/OpenFOAM/OpenFOAM-2.2.x/etc/bashrc**

then type in the current terminal window:

- source **$HOME/**.bashrc

However, usually after modifying **$HOME/**.bashrc I prefer to logout and login, so all the changes will take effect globally, it is up to you.

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:

- foamSystemCheck

If any critical software is missing, or needs updating to a newer version, install the required software before proceeding to the building stage.

Building OpenFOAM-2.2.x

If the previous check passed, we can now compile OpenFOAM-2.2.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.

Building OpenFOAM-2.2.x

Now go to the OpenFOAM-2.2.x installation directory **$WM_PROJECT_DIR** and execute the build script Allwmake, from the terminal type:

- cd **$WM_PROJECT_DIR**
- ./Allwmake

In principle this will build everything.

After compiling OpenFOAM 2.2.x, we need to compile the Paraview module and the PV3FoamReader module.

OpenFOAM® installation (version 2.2.x)

Paraview is the third-party software used for post-processing in OpenFOAM®. Its compilation is automated using the script makeParaView located in the **ThirdParty-2.2.x** directory. To compile Paraview, execute the following commands:

- cd **$WM_THIRD_PARTY_DIR**
- ./makeParaView

Then you need to compile PV3blockMeshReader and the PV3FoamReader plugins. From the terminal:

- cd **$FOAM_UTILITIES/postProcessing/graphics/PV3Readers**
- 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 OpenFOAM-2.2.x

OpenFOAM® installation (version 2.2.x)

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-2.2.x

otherwise, go back and check that you have installed the software correctly.

OpenFOAM® installation (version 2.2.x)

Finally

Create an user directory in the **$HOME/OpenFOAM** directory named **USER-2.2.x** (e.g. **banana-2.2.x** for **USER** banana and **-2.2.x** for OpenFOAM version 2.2.x), in the terminal type:

- cd **$WM_PROJECT**
- mkdir **banana-2.2.x**

Now create a directory named **run** within it:

- cd **banana-2.2.x**
- mkdir **run**

Then from the terminal:

- cd **$FOAM_RUN**
- pwd

Then you should see your current working directory, which should be:

$HOME/OpenFOAM/banana-2.2.x/run

OpenFOAM® installation (version 2.2.x)

Finally

Run the first example case of incompressible flow in a lid-driven square cavity:

- cp -r **$FOAM_TUTORIALS $FOAM_RUN**
- cd **$FOAM_RUN/tutorials/incompressible/icoFoam/cavity**
- blockMesh
- icoFoam
- paraFoam

Enjoy and happy CFD

Updating your OpenFOAM-2.2.x source code installation

To update OpenFOAM 2.2.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

To compile OpenFOAM-2.2.x doxygen documentation

To compile Doxygen documentation, from the terminal:

- cd **$WM_PROJECT_DIR**
- ./Allwmake doc

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

OpenFOAM-1.6-ext installation

OpenFOAM® installation (version 2.2.x)

ATTENTION

If for any reason the installation instructions given in http://www.extend-project.de/ do not work for you, try to follow mine instructions, they work fine in OpenSUSE 11.4 or newer version.

OpenFOAM® installation (version 1.6-ext)

ATTENTION

I strongly suggest you not to copy and paste these instructions in the terminal, as it may experience some difficulties interpreting the characters.

OpenFOAM® installation (version 1.6-ext)

Downloading the source code

- Download OpenFOAM-1.6-ext source code using the terminal:
  - cd **$HOME**
  - mkdir **OpenFOAM** (if it does not exist)
  - cd **OpenFOAM**
  - git clone git://openfoam-extend.git.sourceforge.net/gitroot/openfoam-extend/openfoam-extend OpenFOAM-1.6-ext
- As soon as you finish downloading OpenFOAM-1.6-ext, do the following:
  - tar xzf OpenFOAM-1.6-ext.tar.gz
  - cd **OpenFOAM-1.6-ext**
- You might need to install rpmbuild

OpenFOAM® installation (version 1.6-ext)

Downloading the source code

The last time I checked (15MAY2013), the repository

git://openfoam-extend.git.sourceforge.net/gitroot/openfoam-extend/openfoam-extend OpenFOAM-1.6-ext

was offline.

If you are interested in installing the extend version let me know, I have a compress file of the directory.

You can also try the following alternate repositories

git clone git://repo.or.cz/OpenFOAM-1.6-ext.git

git clone git://github.com/ogoe/OpenFOAM-1.6-ext.git OpenFOAM-1.6-ext

git clone git://git.code.sf.net/p/openfoam-extend/OpenFOAM-1.6-ext openfoam-extend-OpenFOAM-1.6-ext

OpenFOAM® installation (version 1.6-ext)

Environment variables

The environment variable settings are contained in **OpenFOAM-1.6-ext/etc** directory, if you followed the instructions it should be in:

- **$HOME/OpenFOAM/OpenFOAM-1.6-ext/etc**

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 the end of your **$HOME/**.bashrc file:

- **source $HOME/OpenFOAM/OpenFOAM-1.6-ext/etc/bashrc**

Do not forget to comment the line pointing to the bashrc file of another OpenFOAM® installation. Type in the current terminal window:

- wmUNSET (this is to unset OpenFOAM-2.2.x environment variables)
- source **$HOME/**.bashrc

However, usually after modifying **$HOME/**.bashrc I prefer to logout and login, so all the changes will take effect globally, it is up to you.

Environment variables

- cp **$HOME/OpenFOAM/OpenFOAM-1.6-ext/etc/**prefs.sh-EXAMPLE **$HOME/OpenFOAM/OpenFOAM-1.6-ext/etc/**prefs.sh (this is one line)

cp $HOME/OpenFOAM/OpenFOAM-1.6-ext/etc/prefs.sh-EXAMPLE $HOME/OpenFOAM/OpenFOAM-1.6-ext/etc/prefs.sh

Now use gedit or your favorite text editor

- gedit **$HOME/OpenFOAM/OpenFOAM-1.6-ext/etc/**prefs.sh

and uncomment the following lines (lines 137 and 138 on my installation)

- **export QT_DIR=/usr/lib64/qt4/**
- **export QT_BIN_DIR=$QT_DIR/bin**

You might need to add **/usr/lib/qt4** to the **QT_DIR** variable.

Take your time to read prefs.sh, probably you will need to set some other environment variables.

OpenFOAM® installation (version 1.6-ext)

Environment variables

After doing all the necessary changes in **$HOME/OpenFOAM/OpenFOAM-1.6-ext/etc/**prefs.sh type in the current terminal window:

- source **$HOME/**.bashrc

Also, do not forget to source the basic configuration

- source **$HOME**/OpenFOAM/OpenFOAM-1.6-ext/etc/bashrc

However, usually after modifying **$HOME/**.bashrc I prefer to logout and login, so all the changes will take effect globally, it is up to you.

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.

If any critical software is missing, or needs updating to a newer version, install the required software before proceeding to the building stage. If you are in doubt just ask me for some help.

Building OpenFOAM-1.6-ext

If the previous check passed, we can now compile OpenFOAM-1.6-ext.

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, from the terminal type:

- export WM_NCOMPPROCS=NP

where NP is the number of processor you want to use.

If your OpenSUSE distribution does not have gmake installed, you will need to create the following symbolic link:

- sudo ln **-s /usr/bin/**make **/usr/bin/**gmake

To do this you will need administrative privileges

Building OpenFOAM-1.6-ext

Now go to ThirdParty installation directory **$WM_PROJECT_DIR/ThirdParty** and execute the build script AllMake, from the terminal type:

- cd **$WM_PROJECT_DIR /ThirdParty**
- ./AllMake

In principle this will build all ThirdParty libraries and utilities. After you finish compiling the ThirdParty utilities

- source **$HOME/**.bashrc

On my installation, I usually get errors when compiling paraFoam (AllMake.stage4), if you also get errors, do not mind and continue the installation.

If you are on the mood, try to sort out what are the problems. In any case, this will only affect the paraFoam installation.

Building OpenFOAM-1.6-ext

If you get errors during any stage of the installation, try to sort out what are the problems and restart the building process from the stage you were getting problems:

- ./AllMake.stage1
- source **$HOME/**.bashrc
- ./AllMake.stage2
- source **$HOME/**.bashrc
- ./AllMake.stage3
- source **$HOME/**.bashrc
- ./AllMake.stage4
- source **$HOME/**.bashrc

As usual, after modifying **$HOME/**.bashrc I prefer to logout and login, so all the changes will take effect globally, it is up to you.

Building OpenFOAM-1.6-ext

After compiling the ThirdParty libraries and utilities, we can build OpenFOAM-1.6-ext, from the terminal:

- cd **$WM_PROJECT_DIR**
- ./Allwmake

At this point and if you did not get any error during the building process, you should have a fully working installation of OpenFOAM-1.6-ext

OpenFOAM® installation (version 1.6-ext)

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-1.6-ext

otherwise, go back and check that you have installed the software correctly.

OpenFOAM® installation (version 1.6-ext)

Finally

Create an user directory in the **$HOME/OpenFOAM** directory named **USER-1.6-ext** (e.g. **banana-1.6-ext** for **USER** banana and **-1.6-ext** for OpenFOAM® version 1.6-ext), in the terminal type:

- cd **$WM_PROJECT**
- mkdir **banana-1.6-ext**

Now create a directory named **run** within it:

- cd **banana-1.6-ext**
- mkdir **run**

Then from the terminal:

- cd **$FOAM_RUN**
- pwd

Then you should see your current working directory, which should be:

$HOME/OpenFOAM/banana-1.6-ext/run

OpenFOAM® installation (version 1.6-ext)

Finally

Run the first example case of incompressible flow in a lid-driven square cavity:

- cp -r **$FOAM_TUTORIALS $FOAM_RUN**
- cd **$FOAM_RUN/tutorials/incompressible/icoFoam/cavity**
- blockMesh
- icoFoam
- paraFoam

Enjoy and happy CFD

Updating your OpenFOAM-1.6-ext source code installation

To update OpenFOAM-1.6-ext 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 just 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

1. ~~OpenFOAM installation~~
2. **Additional utilities installation**
3. CAD and Meshing applications

The following utilities might come in handy:

PyFoam

A python library to control OpenFOAM® runs and manipulates OpenFOAM® data. Comes with a number of utilities that should make your life easier.

swak4Foam

A library that combines the functionality of groovyBC (a library to implement user defined boundary conditions without programming) and funkySetFields (an utility that sets the initial value of a scalar or a vector field without programming, it is like the setFields utility but on steroids).

setDiscreteFields

This utility can set profiles of any vector/scalar fields at specified boundary patches by using discrete profile data.

OpenFOAM® installation

For source code download and installation instructions go to:

PyFoam

http://openfoamwiki.net/index.php/Contrib_PyFoam

http://openfoamwiki.net/index.php/Contrib_PyFoam

Swak4foam

http://openfoamwiki.net/index.php/Contrib/swak4Foam

setDiscreteFields

http://openfoamwiki.net/index.php/Contrib_setDiscreteFields

http://openfoamwiki.net/index.php/Contrib_setDiscreteFields

I highly advice you to install these utilities (and those that you might consider handy) in your user directory **$WM_PROJECT_USER_DIR.**

OpenFOAM® installation

1. ~~OpenFOAM installation~~
2. ~~Additional utilities installation~~
3. CAD and Meshing applications

OpenFOAM® installation

For geometry generation, mesh generation, and visualization, the following open source applications might come in handy:

CAD - 3D modeling - Surface modeling

- **SALOME** (http://www.salome-platform.org/).
- **Google SketchUp** (http://sketchup.google.com/).
- **Free-CAD** (http://sourceforge.net/apps/mediawiki/free-cad/).
- **Blender** (http://www.blender.org/).

Note: precompiled binaries are available for all these applications.

Note: it is possible to install Google SketchUp in linux by using wine (http://www.winehq.org/).

OpenFOAM® installation

For geometry generation, mesh generation, and visualization, the following open source applications might come in handy:

Mesh Generation

- **SALOME** (http://www.salome-platform.org/). *
- **ENGRID** (http://engits.eu/en/engrid). *
- **GMSH** (http://www.geuz.org/gmsh/). *
- **Triangle** (http://www.cs.cmu.edu/~quake/triangle.html).
- **Tetgen** (http://tetgen.berlios.de/).

* Note: precompiled binaries are available for these applications.

OpenFOAM® installation

For geometry generation, mesh generation, and visualization, the following open source applications might come in handy:

Visualization - STL files manipulation

- **SALOME** (http://www.salome-platform.org/).
- **VISIT** (https://wci.llnl.gov/codes/visit/).
- **Paraview** (http://www.paraview.org/).
- **Meshlab** (http://meshlab.sourceforge.net/).
- **Netfabb** (http://www.netfabb.com/).

Note: precompiled binaries are available for all these applications.

OpenFOAM® installation

For geometry generation, mesh generation, and visualization, the following open source applications might come in handy:

OpenFOAM® GUI

- **Discretizer** (http://www.discretizer.org/).
- **Helyx-os** (http://engys.com/products/helyx-os).

Note: precompiled binaries are available for all these applications.

Thank you for your attention