

# OpenFOAM

A free, open source CFD software package

Introduction\*\* —————

OpenFOAM is a free, open source CFD software package developed by **OpenCFD Ltd** at **ESI Group** and distributed by the **OpenFOAM Foundation** . It has a large user base across most areas of engineering and science, from both commercial and academic organisations.

Homepage: <http://www.openfoam.com/>

## Installed version\*\*

Currently, several version compiled by GCC/ICC compilers in single/double precision with several version of openmpi are available on Anselm.

For example syntax of available OpenFOAM module is:

```
< openfoam/2.2.1-icc-openmpi1.6.5-DP >
```

this means openfoam version >2.2.1 compiled by >ICC compiler with >openmpi1.6.5 in double precision.

Naming convection of the installed versions is following:

```
openfoam/<>VERSION>>-<COMPILER>-<openmpiVERSION>-<PRECISION>
```

- <>VERSION> - version of openfoam
- <>COMPILER> - version of used compiler
- <>openmpiVERSION> - version of used openmpi/impi
- <>PRECISION> - DP/SP – double/single precision

## Available OpenFOAM modules\*\*

To check available modules use

```
$ module avail
```

In /opt/modules/modulefiles/engineering you can see installed engineering softwares:

```
----- /opt/modules/modulefiles/engineering -----
ansys/14.5.x          matlab/R2013a-COM    openfoam/2.2.1-i
comsol/43b-COM        matlab/R2013a-EDU    openfoam/2.2.1-i
comsol/43b-EDU        openfoam/2.2.1-gcc481-openmpi1.6.5-DP  paraview/4.0.1-g
```

lsdyna/7.x.x                      openfoam/2.2.1-gcc481-openmpi1.6.5-SP

For information how to use modules please look here.

Getting Started\*\* \_\_\_\_\_

To create OpenFOAM environment on ANSELM give the commands:

```
$ module load openfoam/2.2.1-icc-openmpi1.6.5-DP
```

```
$ source $FOAM_BASHRC
```

Please load correct module with your requirements "compiler - GCC/ICC, precision - DP/SP".

Create a project directory within the \$HOME/OpenFOAM directory named ><USER>-<OFversion> and create a directory named run within it, e.g. by typing:

```
$ mkdir -p $FOAM_RUN
```

Project directory is now available by typing:

```
$ cd /home/<USER>/OpenFOAM/<USER>-<OFversion>/run
```

<OFversion> - for example <2.2.1>

or

```
$ cd $FOAM_RUN
```

Copy the tutorial examples directory in the OpenFOAM distribution to the run directory:

```
$ cp -r $FOAM_TUTORIALS $FOAM_RUN
```

Now you can run the first case for example incompressible laminar flow in a cavity.

Running Serial Applications\*\* \_\_\_\_\_

Create a Bash script >test.sh

## !/bin/bash

```
module load openfoam/2.2.1-icc-openmpi1.6.5-DP source $FOAM_BASHRC
```

```
# source to run functions
```

```
. $WM_PROJECT_DIR/bin/tools/RunFunctions
```

```
cd $FOAM_RUN/tutorials/incompressible/icoFoam/cavity
```

```
runApplication blockMesh
runApplication icoFoam
```

Job submission

```
$ qsub -A OPEN-0-0 -q qprod -l select=1:ncpus=16,walltime=03:00:00
test.sh
```

For information about job submission please look here.

Running applications in parallel\*\* \_\_\_\_\_  
\_\_\_\_\_

Run the second case for example external incompressible turbulent flow - case - motorBike.

First we must run serial application blockMesh and decomposePar for preparation of parallel computation.

Create a Bash scrip test.sh:

**!/bin/bash**

```
module load openfoam/2.2.1-icc-openmpi1.6.5-DP source $FOAM_BASHRC
# source to run functions
. $WM_PROJECT_DIR/bin/tools/RunFunctions

cd $FOAM_RUN/tutorials/incompressible/simpleFoam/motorBike

runApplication blockMesh
runApplication decomposePar
```

Job submission

```
$ qsub -A OPEN-0-0 -q qprod -l select=1:ncpus=16,walltime=03:00:00
test.sh
```

This job create simple block mesh and domain decomposition. Check your decomposition, and submit parallel computation:

Create a PBS script testParallel.pbs:

**#!/bin/bash**

**PBS -N motorBike**

**PBS -l select=2:ncpus=16**

**PBS -l walltime=01:00:00**

**PBS -q qprod**

**PBS -A OPEN-0-0**

```
module load openfoam/2.2.1-icc-openmpi1.6.5-DP
source $FOAM_BASHRC
```

```
cd $FOAM_RUN/tutorials/incompressible/simpleFoam/motorBike
```

```
nproc = 32
```

```
mpirun -hostfile ${PBS_NODEFILE} -np $nproc snappyHexMesh -overwrite -parallel | tee snappyHexM
```

```
mpirun -hostfile ${PBS_NODEFILE} -np $nproc potentialFoam -noFunctionObject-writep -parallel |
```

```
mpirun -hostfile ${PBS_NODEFILE} -np $nproc simpleFoam -parallel | tee simpleFoam.log
```

nproc – number of subdomains

Job submission

```
$ qsub testParallel.pbs
```

Compile your own solver\*\* \_\_\_\_\_

Initialize OpenFOAM environment before compiling your solver

```
$ module load openfoam/2.2.1-icc-openmpi1.6.5-DP $ source
$FOAM_BASHRC $ cd $FOAM_RUN/
```

Create directory applications/solvers in user directory

```
$ mkdir -p applications/solvers $ cd applications/solvers
```

Copy icoFoam solver's source files

```
$ cp -r $FOAM_SOLVERS/incompressible/icoFoam/ My_icoFoam
$ cd My_icoFoam
```

Rename icoFoam.C to My\_icoFOAM.C

```
$ mv icoFoam.C My_icoFoam.C
```

Edit *>files* file in *Make* directory:

```
icoFoam.C EXE = $(FOAM_APPBIN)/icoFoam
```

and change to:

```
My_icoFoam.C
EXE = $(FOAM_USER_APPBIN)/My_icoFoam
```

In directory My\_icoFoam give the compilation command:

```
$ wmake
```

---

Have a fun with OpenFOAM :)\*\*

id="\_\_\_caret">