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
```

Pleas 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 bockMesh 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">