

Compiling software on the user profile

This section aims at helping the users compile the software they need. They are mere guidelines!

Incompact3d

Following instructions from <https://github.com/xcompact3d/Incompact3d>

```
salloc
```

```
git clone https://github.com/xcompact3d/Incompact3d
cd Incompact3d/
git checkout v3.0
module load mpi/openmpi-x86_64
make
```

```
# once compiled
cd examples/Cylinder
pwd
/homes/<user>/Incompact3d/examples/Cylinder
```

```
mpirun -n 1 ~/Incompact3d/xcompact3d
```

```
Xcompact3d is run with the default file -->input.i3d
=====
=====Xcompact3D=====
===Copyright (c) 2018 Eric Lamballais and Sylvain Laizet===
===Modified by Felipe Schuch and Ricardo Frantz=====
===Modified by Paul Bartholomew, Georgios Deskos and=====
===Sylvain Laizet -- 2018- =====
=====
Git version          : v3.0-0-g9015765

Simulating cylinder
(lx,ly,lz)= 20.000000000000000          12.000000000000000
6.000000000000000
(nx,ny,nz)=          257          128          64
(dx,dy,dz)= 7.812500000000000E-002    9.375000000000000E-002
9.375000000000000E-002
(nx*ny*nz)= 2105344
(p_row,p_col)=          0          0

Numerical precision: Double
Boundary condition : (nclx1 ,nclxn )=(2,2)
                   (ncly1 ,nclyn )=(0,0)
                   (nclz1 ,nclzn )=(0,0)
High and low speed : u1= 1.00 and u2= 1.00
Reynolds number Re : 300.00000000
```

```
Gravity vector      : (gx, gy, gz)=(      0.00000000,      0.00000000,  
0.00000000)  
Time step dt       :      0.00250000  
Spatial scheme     :      0.69477383  
Temporal scheme    : Adams-bashforth 3  
Scalar             : off  
Immersed boundary : on with Lagrangian Poly
```

In auto-tuning mode.....

```
factors:          1  
processor grid    1 by 1 time=
```

8.6869291961193085E-002

```
the best processor grid is probably      1 by 1
```

Initializing variables...

```
Using the hyperviscous operator with (nu_0/nu,c_nu) = (  
4.0000000000000000      , 0.4400000000000000      )
```

```
Using the hyperviscous operator with (nu_0/nu,c_nu) = (  
4.0000000000000000      , 0.4400000000000000      )
```

```
Using the hyperviscous operator with (nu_0/nu,c_nu) = (  
4.0000000000000000      , 0.4400000000000000      )
```

Generating the geometry!

step 1

step 2

step 3

mpirun noticed that process rank 0 with PID 16346 on node ava01 exited on
signal 9 (Killed).

OpenFoam7



A Distro “Scientific Linux 7” não faz parte dos SO's suportados, pelo openfoam, pelo que não podemos garantir que funcione corretamente sendo também por isso que a aplicação terá de ser compilada na conta de cada um, para evitar a introdução de instabilidade no sistema important box

infos <https://openfoam.org/download/7-source/>

```
salloc -n 16 -p big
```

```
mkdir OpenFOAM
```

```
cd OpenFOAM/
```

```
wget -O - http://dl.openfoam.org/source/7 | tar xvz
```

```
wget -O - http://dl.openfoam.org/third-party/7 | tar xvz
```

```
mv OpenFOAM-7-version-7 OpenFOAM-7
```

```
mv ThirdParty-7-version-7 ThirdParty-7
```

```
# https://openfoam.org/download/source/software-for-compilation/
source $HOME/OpenFOAM/OpenFOAM-7/etc/bashrc WM_LABEL_SIZE=64 WM_MPLIB=OPENMPI
FOAMY_HEX_MESH=yes

cd ThirdParty-7
mkdir download
wget -P download https://www.open-mpi.org/software/ompi/v2.1/downloads/openmpi-2.1.1.tar.bz2
tar -xjf download/openmpi-2.1.1.tar.bz2

# https://openfoam.org/download/source/third-party-software/
./Allwmake -j 16

# https://www.paraview.org/download/
wget
'https://www.paraview.org/paraview-downloads/download.php?submit=Download&version=v5.6&type=
binary&os=Linux&downloadFile=ParaView-5.6.0-MPI-Linux-64bit.tar.gz' -O ParaView.tar.gz
gunzip ParaView.tar.gz
tar -xf ParaView.tar
rm ParaView.tar
wmRefresh

cd ..
cd OpenFOAM-7
module load gcc/9.3.1
./Allwmake -j 16

# Getting started
mkdir -p $FOAM_RUN
cd $FOAM_RUN
cp -r $FOAM_TUTORIALS/incompressible/simpleFoam/pitzDaily .
cd pitzDaily
blockMesh
simpleFoam
# more at http://cfd.direct/openfoam/user-guide
```

OpenFoam6

```
# A different approach! Just do:
```

```
salloc -n 6
source /soft/no_support/OpenFOAM/OpenFOAM-6/etc/bashrc WM_LABEL_SIZE=64
WM_MPLIB=OPENMPI FOAMY_HEX_MESH=yes
```

```
# Then You can run OpenFoam cmds like:
icoFoam -help
```

```
# Next, the compilation of this software and sources
# NOTE: You do not need to compile if You use the method "source
/soft/no_support/..." explained before!
```

```
https://openfoamwiki.net/index.php/Installation/Linux/OpenFOAM-6/CentOS_SL_RHEL
2.3 CentOS 7.5 (1804)

cd ~
mkdir OpenFOAM
cd OpenFOAM
git clone https://github.com/OpenFOAM/OpenFOAM-6.git
git clone https://github.com/OpenFOAM/ThirdParty-6.git

cd ThirdParty-6
mkdir download

wget -P download https://www.cmake.org/files/v3.9/cmake-3.9.0.tar.gz
wget -P download
https://github.com/CGAL/cgal/releases/download/releases%2FCGAL-4.10/CGAL-4.10.tar.xz
wget -P download
https://sourceforge.net/projects/boost/files/boost/1.55.0/boost_1_55_0.tar.bz2
wget -P download
https://www.open-mpi.org/software/mpi/v2.1/downloads/openmpi-2.1.1.tar.bz2
wget -P download http://www.paraview.org/files/v5.4/ParaView-v5.4.0.tar.gz

tar -xzf download/cmake-3.9.0.tar.gz
tar -xJf download/CGAL-4.10.tar.xz
tar -xjf download/boost_1_55_0.tar.bz2
tar -xjf download/openmpi-2.1.1.tar.bz2
tar -xzf download/ParaView-v5.4.0.tar.gz --transform='s/ParaView-v5.4.0/ParaView-5.4.0/'

cd ..
sed -i -e 's/(\(boost_version=\)boost-system/\1boost_1_55_0/'
OpenFOAM-6/etc/config.sh/CGAL
sed -i -e 's/(\(cgal_version=\)cgal-system/\1CGAL-4.10/'
OpenFOAM-6/etc/config.sh/CGAL

source $HOME/OpenFOAM/OpenFOAM-6/etc/bashrc WM_LABEL_SIZE=64
WM_MPLIB=OPENMPI FOAMY_HEX_MESH=yes

cd $WM_THIRD_PARTY_DIR
./makeCmake > log.makeCmake 2>&1
wmRefresh

cd $WM_THIRD_PARTY_DIR
./Allwmake > log.make 2>&1
wmRefresh

cd $WM_THIRD_PARTY_DIR
./makeParaView -mpi -python -qmake $(which qmake-qt4) > log.makePV 2>&1
```

wmRefresh

```
cd $WM_PROJECT_DIR
./Allwmake -j 8 > log.make 2>&1

# Check if icoFoam is working, by running this command:
icoFoam -help

# https://openfoamwiki.net/index.php/FAQ/Installation_and_Running

# manual
# https://cfd.direct/openfoam/user-guide/
ls -A1 $WM_PROJECT_DIR/doc/Guides*/*UserGuide*.pdf
```

From:

<https://grid.fe.up.pt/dokuwiki/> - **GRID FEUP**

Permanent link:

<https://grid.fe.up.pt/dokuwiki/doku.php?id=documentation:software:compiling>

Last update: **2022/01/04 13:24**

