

HyperCFD
Fertinaz Yazılım Ltd.

OpenFOAM Workshop Teknopark İstanbul

01 Introduction
January 2017



HyperCFD – About The Tutor

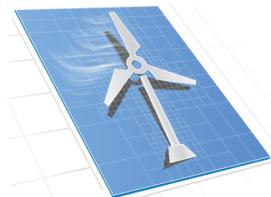


Fatih Ertinaz

BSc at ITU in Engineering Physics

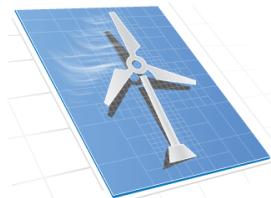
MSc at KTH in Scientific Computing

Has been working with OpenFOAM since late 2010



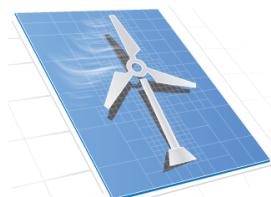
HyperCFD – Disclaimer

This offering is not approved or endorsed by OpenCFD Limited, the producer of the OpenFOAM software and owner of the OpenFOAM and OpenCFD trademarks.



HyperCFD – Workshop Contents

- OpenFOAM installation and library outlook
- Basic meshing
- Mesh conversion and manipulation
- Boundary conditions
- Field initialization and turbulence models
- Solvers and numerical methods
- Parallel processing
- Tutorial samples
- CFD basics and finite volume method
- OpenFOAM programming
- Post-processing
- Third-party OpenFOAM libraries



HyperCFD – OpenFOAM: General Description

- Open source CFD toolbox: Field Operation And Manipulation
- Linux based C++ library
- No GUI
- Object oriented programming paradigm
- Built-in meshing tools, solvers, turbulence models and IC & BC's
- Free to use, modify and distribute

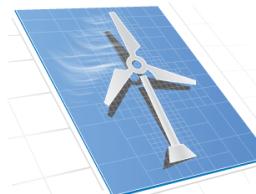
```
Fertinaz@fertinaz-M4800:~/OpenFOAM/fertinaz-3.0.x/run/turbineSiting.tests/turbineSiting.8turbines.finer
fertinaz@fertinaz-M4800:~$ tail .bashrc
alias FE31=' . $HOME/foam/foam-extend-3.1/etc/bashrc'
alias OF20x='source $HOME/OpenFOAM/OpenFOAM-2.0.x/etc/myBashrc'
alias OF16ext='source $HOME/OpenFOAM/OpenFOAM-1.6-ext/etc/bashrc'
alias OF21x=' . /home/fertinaz/OpenFOAM/OpenFOAM-2.1.x/etc/bashrc'
alias OF24x=' . /home/fertinaz/OpenFOAM/OpenFOAM-2.4.x/etc/bashrc'
alias OF30x=' . /home/fertinaz/OpenFOAM/OpenFOAM-3.0.x/etc/bashrc'
alias OF30xDebug=' . /home/fertinaz/OpenFOAM/OpenFOAM-3.0.x/etc/bashrc_dbg'
alias FE32='source $HOME/foam/foam-extend-3.2/etc/bashrc'

export NCARG_ROOT=/usr/bin/
fertinaz@fertinaz-M4800:~$ OF30x → Source OpenFOAM-3.0.x
fertinaz@fertinaz-M4800:~$ run
fertinaz@fertinaz-M4800:~/OpenFOAM/fertinaz-3.0.x/run$ cd turbineSiting.tests/turbineSiting.8turbines.finer
fertinaz@fertinaz-M4800:~/OpenFOAM/fertinaz-3.0.x/run/turbineSiting.tests/turbineSiting.8turbines.finer$ ls
0  Allclean  constant  log.decomposePar  processor0  processor2  system
0.org  Allrun  log.blockMesh  log.snapyHexMesh  processor1  processor3  tmp.foam
fertinaz@fertinaz-M4800:~/OpenFOAM/fertinaz-3.0.x/run/turbineSiting.tests/turbineSiting.8turbines.finer$ simpleFoam
/*-----*/
| ====== | F ield | OpenFOAM: The Open Source CFD Toolbox |
| \ \ / O peration | Version: 3.0.x |
| \ \ / A nd | Web: www.OpenFOAM.org |
| \ \ M anipulation | |
\*-----*/
Build : 3.0.x-195caf7479b2
Exec  : simpleFoam
Date  : Oct 28 2016
Time  : 11:59:24
Host  : "fertinaz-M4800"
PID   : 8194
Case  : /home/fertinaz/OpenFOAM/fertinaz-3.0.x/run/turbineSiting.tests/turbineSiting.8turbines.finer
nProcs : 1
sigFpe : Enabling floating point exception trapping (FOAM_SIGFPE).
fileModificationChecking : Monitoring run-time modified files using timeStampMaster
allowSystemOperations : Allowing user-supplied system call operations

// * * * * *
Create time
Create mesh for time = 0

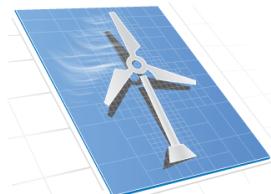
SIMPLE: no convergence criteria found. Calculations will run for 600 steps.
```

Various OF versions



HyperCFD – OpenFOAM: General Description

- Based on Finite Volume Method
- Runs on unstructured meshes
 - Built-in mesh generators: blockMesh, snappyHexMesh
- Wide variety of flow solvers, turbulence models, discretization schemes, boundary conditions, linear solvers etc.
- OOP – Reusable and readable
 - Easy to implement top-level code
- Able to run in parallel machines
 - MPI based parallelism
 - Does not support CUDA by default however there are 2-3 libraries developed (afaik)



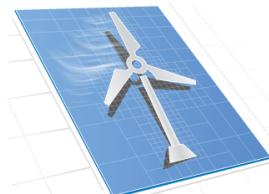
HyperCFD – OpenFOAM Installation

Binary installation for Linux and Mac

- Fairly new Docker technology
- First install Docker then OpenFoam
- For detailed explanation:
<http://openfoam.com/download/install-binary.php>

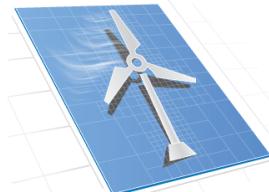
Windows installation

- Download the installer file
<http://openfoam.com/download/install-windows.php>
- Follow the instructions
- Very easy to install
- However not very practical afterwards
 - Windows file read/write permissions
 - Max. characters supported by Windows
 - Case insensitive structure of Windows



HyperCFD – OpenFOAM Installation on Linux

- Source code installation
- Best way to install OpenFoam on your (Linux) laptop is the nerd way :)
- Create OpenFOAM directory:
`mkdir -p $HOME/OpenFOAM`
- Go to that folder:
`cd $HOME/OpenFOAM`
- Install prerequisites:
`sudo apt-get install build-essential flex bison git-core`
- These libraries depend on the Linux distro you use and OpenFOAM version you want to install



HyperCFD – OpenFOAM Installation on Linux

- Clone the git repo -- OpenFOAM:

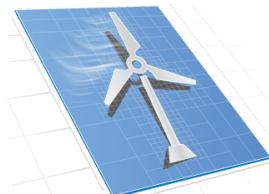
```
git clone git://github.com/OpenFOAM/OpenFOAM-3.0.x.git
```

- Clone the git repo – External libraries needed by OpenFOAM:

```
git clone git://github.com/OpenFOAM/ThirdParty-3.0.x.git
```

- Create alias in your .bashrc – don't do this if you want OF loaded by default:

```
echo "alias OF30x='.` $HOME/OpenFOAM/OpenFOAM-3.0.x/etc/bashrc'" >>
$HOME/.bashrc
```



HyperCFD – OpenFOAM Installation

- Go to OpenFOAM-3.0.x folder:

```
cd $HOME/OpenFOAM/OpenFOAM-3.0.x
```

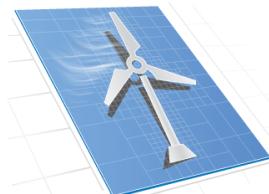
- Start compiling:

```
./Allwmake 2>&1 | tee log.Allwmake
```

- You can do it again to see if there are errors – will be much faster:

```
./Allwmake 2>&1 | tee log.Allwmake
```

- If you do not see any errors in this log file, installation should be finished successfully

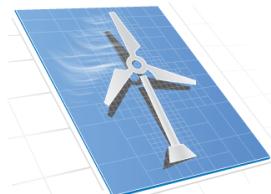


HyperCFD – OpenFOAM Installation

- Test if everything went well:

```
# Source OF-3.0  
OF30  
  
# Create run directory under $HOME/OpenFOAM/$USER-3.0  
mkdir -p $FOAM_RUN  
cd $FOAM_RUN  
  
# Copy tutorial case  
cp -r $FOAM_TUTORIALS/incompressible/icoFoam/cavity .  
cd cavity  
  
# Run case  
.Allrun
```

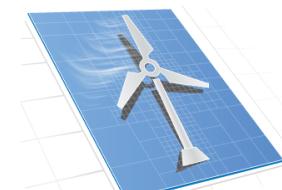
- Therefore we run our first OpenFOAM simulation. It is crucial to become proficient in the command line to make your life easier.



HyperCFD – OpenFOAM Overview

- When you install OF, you'll see following directories:
 - OpenFoam-4.0/applications
 - OpenFoam-4.0/bin
 - OpenFoam-4.0/etc
 - OpenFoam-4.0/src
 - OpenFoam-4.0/tutorials

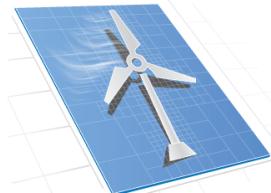
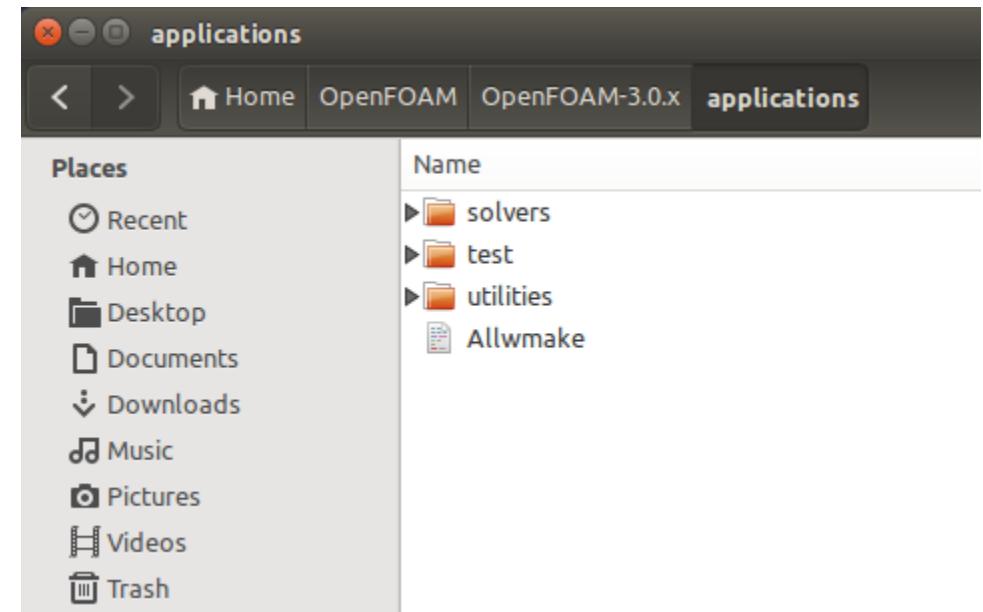
Name	Size	Type	Modified	Permissions
applications	4 items	Folder	Haz 25 2016	drwxrwxr-x
bin	77 items	Folder	Kas 10 2016	drwxrwxr-x
doc	5 items	Folder	Haz 25 2016	drwxrwxr-x
etc	11 items	Folder	Haz 25 2016	drwxrwxr-x
platforms	2 items	Folder	Kas 10 2016	drwxrwxr-x
src	37 items	Folder	Haz 25 2016	drwxrwxr-x
tutorials	17 items	Folder	Haz 25 2016	drwxrwxr-x
wmake	19 items	Folder	Kas 10 2016	drwxrwxr-x
Allwmake	1,3 kB	Program	Haz 25 2016	-rwxrwxr-x
COPYING	35,6 kB	Text	Haz 25 2016	-rw-rw-r-
log.make	142,3 kB	Text	Kas 10 2016	-rw-rw-r-
README.org	1,6 kB	Text	Haz 25 2016	-rw-rw-r-



HyperCFD – OpenFOAM Directories: applications

- OpenFoam-4.0/applications
- Type one of the following in your terminal:

```
app
cd $WM_PROJECT_DIR/applications
```
- These folders contain
 - solvers: Source code of the solvers
 - test: Contains the source code of several test cases that show the usage of some of the OpenFOAM classes
 - utilities: Source code for OpenFOAM utilities
- In each of these folders, you can find lots of subfolders



HyperCFD – OpenFOAM Directories: bin

- OpenFoam-4.0/bin
- Type one of the following in your terminal:

```
cd $WM_PROJECT_DIR/bin
```

- These folders contain shell scripts:

createTurbulenceFields

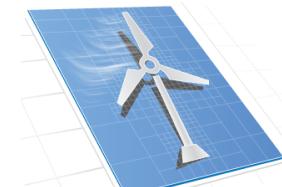
foamLog

Mach

Pe

R

Name	Type	Size	Modified	Permissions
tools	Folder	20 items	Haz 25 2016	drwxrwxr-x
Co	Program	1,6 kB	Haz 25 2016	-rwxrwxr-x
createTurbulenceFields	Program	1,6 kB	Haz 25 2016	-rwxrwxr-x
engridFoam	Program	2,1 kB	Haz 25 2016	-rwxrwxr-x
enstrophy	Link to Program	1,5 kB	Haz 25 2016	lrwxrwxr-x
execFlowFunctionObjects	Program	1,6 kB	Haz 25 2016	-rwxrwxr-x
findEmptyMake	Program	2,4 kB	Haz 25 2016	-rwxrwxr-x
flowType	Link to Program	1,5 kB	Haz 25 2016	lrwxrwxr-x
foamAllHC	Program	1,4 kB	Haz 25 2016	-rwxrwxr-x
foamCalc	Program	1,5 kB	Haz 25 2016	-rwxrwxr-x
foamCheckJobs	Program	10,1 kB	Haz 25 2016	-rwxrwxr-x
foamCleanPath	Program	4,0 kB	Haz 25 2016	-rwxrwxr-x
foamCleanPolyMesh	Program	3,7 kB	Haz 25 2016	-rwxrwxr-x
foamCleanTutorials	Program	2,1 kB	Haz 25 2016	-rwxrwxr-x
foamCloneCase	Program	2,8 kB	Haz 25 2016	-rwxrwxr-x
foamCopySettings	Program	4,1 kB	Haz 25 2016	-rwxrwxr-x
foamCreateVideo	Program	4,3 kB	Haz 25 2016	-rwxrwxr-x
foamDebugSwitches	Program	1,5 kB	Haz 25 2016	-rwxrwxr-x
foamBrowse	Program	1,9 kB	Haz 25 2016	-rwxrwxr-x
foamEndJob	Program	10,7 kB	Haz 25 2016	-rwxrwxr-x
foamEtcFile	Program	7,5 kB	Haz 25 2016	-rwxrwxr-x
foamExec	Program	4,1 kB	Haz 25 2016	-rwxrwxr-x

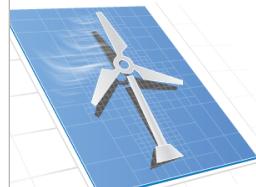


HyperCFD – OpenFOAM Directories: src

- OpenFoam-4.0/src
- Type one of the following in your terminal:

```
SRC  
cd $WM_PROJECT_DIR/src
```
- This folder contain the entire source code of OpenFOAM:

```
finiteVolume  
OpenFOAM  
turbulenceModels
```
- Note that when these are compiled, you'll have libraries (shared objects) under \$FOAM_LIBBIN



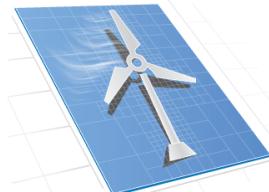
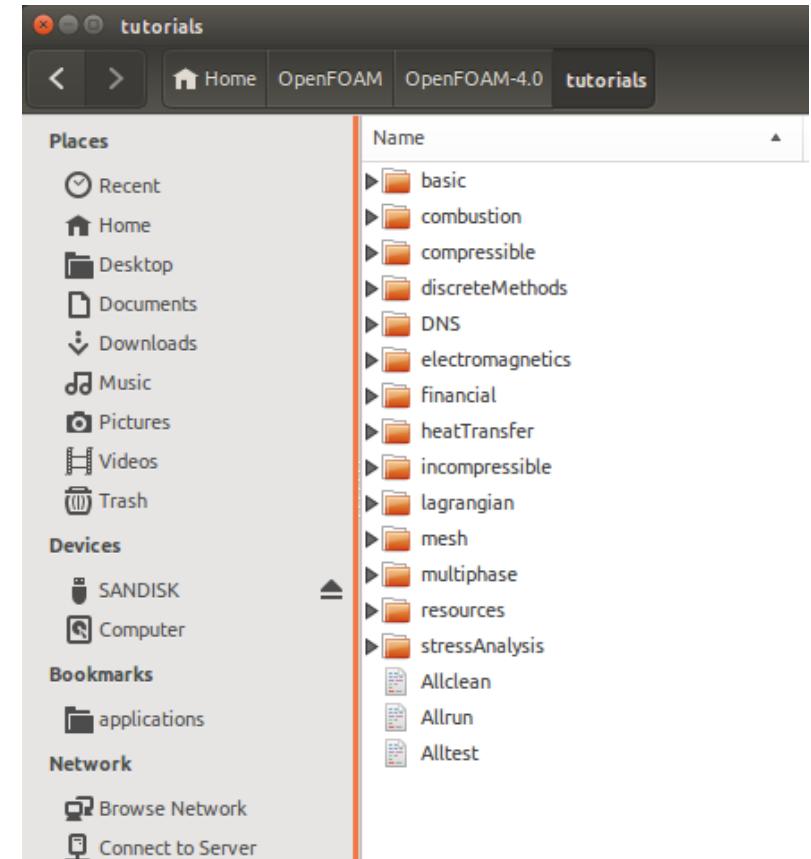
Name	Type	Size	Modified	Permissions
combustionModels	Folder	12 items	Kas 10 2016	drwxrwxr-x
conversion	Folder	8 items	Kas 10 2016	drwxrwxr-x
dummyThirdParty	Folder	5 items	Haz 25 2016	drwxrwxr-x
dynamicFvMesh	Folder	9 items	Kas 10 2016	drwxrwxr-x
dynamicMesh	Folder	22 items	Kas 10 2016	drwxrwxr-x
edgeMesh	Folder	10 items	Kas 10 2016	drwxrwxr-x
engine	Folder	8 items	Kas 10 2016	drwxrwxr-x
fileFormats	Folder	7 items	Kas 10 2016	drwxrwxr-x
finiteVolume	Folder	10 items	Kas 10 2016	drwxrwxr-x
functionObjects	Folder	6 items	Haz 25 2016	drwxrwxr-x
FvAgglomerationMethods	Folder	3 items	Haz 25 2016	drwxrwxr-x
FvMotionSolver	Folder	6 items	Kas 10 2016	drwxrwxr-x
FvOptions	Folder	7 items	Kas 10 2016	drwxrwxr-x
genericPatchFields	Folder	4 items	Kas 10 2016	drwxrwxr-x
lagrangian	Folder	10 items	Haz 25 2016	drwxrwxr-x
mesh	Folder	4 items	Haz 25 2016	drwxrwxr-x
meshTools	Folder	26 items	Kas 10 2016	drwxrwxr-x
ODE	Folder	4 items	Kas 10 2016	drwxrwxr-x
OpenFOAM	Folder	16 items	Kas 10 2016	drwxrwxr-x
OSspecific	Folder	1 item	Haz 25 2016	drwxrwxr-x
parallel	Folder	4 items	Haz 25 2016	drwxrwxr-x
Pstream	Folder	3 items	Haz 25 2016	drwxrwxr-x
randomProcesses	Folder	7 items	Kas 10 2016	drwxrwxr-x
regionCoupled	Folder	3 items	Kas 10 2016	drwxrwxr-x
regionModels	Folder	8 items	Haz 25 2016	drwxrwxr-x
renumber	Folder	4 items	Haz 25 2016	drwxrwxr-x
rigidBodyDynamics	Folder	10 items	Kas 10 2016	drwxrwxr-x
rigidBodyMeshMotion	Folder	4 items	Kas 10 2016	drwxrwxr-x
sampling	Folder	9 items	Kas 10 2016	drwxrwxr-x
sixDoFRigidBodyMotion	Folder	6 items	Kas 10 2016	drwxrwxr-x
surfMesh	Folder	11 items	Kas 10 2016	drwxrwxr-x
thermophysicalModels	Folder	14 items	Haz 25 2016	drwxrwxr-x
topoChangerFvMesh	Folder	8 items	Kas 10 2016	drwxrwxr-x
transportModels	Folder	7 items	Haz 25 2016	drwxrwxr-x
triSurface	Folder	7 items	Kas 10 2016	drwxrwxr-x
TurbulenceModels	Folder	6 items	Haz 25 2016	drwxrwxr-x
Allwmake	Program	2,5 kB	Haz 25 2016	-rwxrwxr-x

HyperCFD – OpenFOAM Directories: tutorials

OpenFoam Directories

- OpenFoam-4.0/tutorials
- Type one of the following in your terminal:

```
tut  
cd $WM_PROJECT_DIR/tutorials
```
- These folders contain sample cases prepared by OpenFOAM distributors
- This is the first place to check when you want to adapt your specific case to OpenFOAM



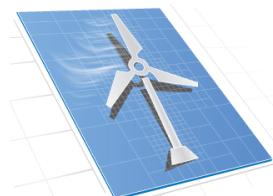
HyperCFD – OpenFOAM Solvers

Basic

- `laplacianFoam`:
Solves the Laplace equation
- `potentialFoam`:
Potential flow solver
- `scalarTransportFoam`:
Solves transport equation of a passive scalar

Combustion

- `chemFoam`:
Chemistry problems
- `engineFoam`:
Internal combustion engines
- `reactingFoam`:
Combustion with chemical reactions



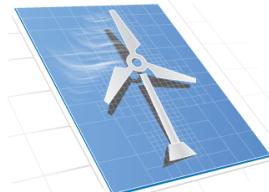
HyperCFD – OpenFOAM Solvers

Compressible

- rhoCentralFoam:
Density based solver
- rhoSimpleFoam:
Steady and turbulent
- rhoPimpleFoam:
Transient solver for turbulent compressible fluids for HVAC
- sonicFoam:
Transient and turbulent for trans-sonic/supersonic gas

Incompressible

- icoFoam:
Transient and laminar
- simpleFoam:
Steady and turbulent using SIMPLE algorithm
- pisoFoam:
Transient solver for incompressible turbulent flow using PISO algorithm
- pimpleFoam: Large time-step solver for transient and turbulent incompressible applications



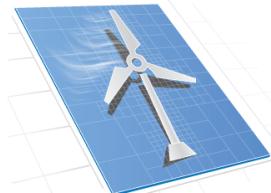
HyperCFD – OpenFOAM Solvers

Heat Transfer

- `buoyantSimpleFoam`:
Steady-state solver for buoyant, turbulent flow of compressible fluid
- `buoyantBoussinesqSimpleFoam`:
Steady-state solver for buoyant, turbulent flow of incompressible fluid
- `buoyantPimpleFoam`: Steady-state solver for buoyant, turbulent flow of compressible fluid for ventilation
- `buoyantBoussinesqPimpleFoam`:
Transient solver for buoyant, turbulent flow of incompressible fluid

Multiphase

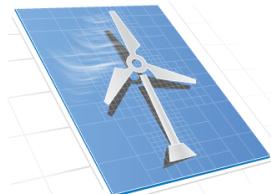
- `interFoam`:
Solver for 2 isothermal, immiscible and incompressible fluids using a VoF phase fraction based interface capturing
- `interDyMFoam`:
Same purpose with mesh motion and adaptive meshing
- `cavitatingFoam`:
Transient cavitation solver



HyperCFD – OpenFOAM Solvers

Others

- Lots of other solvers are available: Variations of the listed solvers, DNS, Molecular Dynamics, Electromagnetics, Viscoelasticity, Solid Mechanics, Stress Analysis, Lagrangian methods
- See full list:
<http://www.openfoam.com/documentation/user-guide/standard-solvers.php>
- No time to go further details in this workshop but there is another version of OpenFOAM called foam-extend
- You can download and install it in a similar fashion to OpenFOAM
- It provides additional methods and solvers. Really worth checking



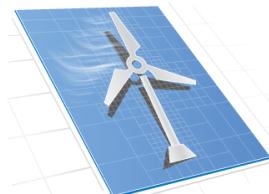
HyperCFD – OpenFOAM Solvers: potentialFoam

A Deeper Look

- **potentialFoam**: Potential flow solver
 $\nabla \cdot U = 0$
 $\nabla^2 p = 0$
- Mainly used for generating initial conditions for faster convergence in complex cases.
- **potentialFoam.C** is the main source for this solver. **Make** folder has the compilation files.
- OpenFOAM uses wmake – a makefile system designed for OF.

Places	Name	Size	Type	Modified	Permissions
Recent	solvers	12 items	Folder	Kas 9 2015	drwxrwxr-x
Home	basic	3 items	Folder	Kas 9 2015	drwxrwxr-x
Desktop	► laplacianFoam	4 items	Folder	Kas 9 2015	drwxrwxr-x
Documents	▼ potentialFoam	4 items	Folder	Kas 9 2015	drwxrwxr-x
Downloads	► Make	2 items	Folder	Kas 9 2015	drwxrwxr-x
Music	► createControls.H	185 bytes	Text	Kas 9 2015	-rw-rw-r-
Pictures	► createFields.H	1,9 kB	Text	Kas 9 2015	-rw-rw-r-
Videos	► potentialFoam.C	5,4 kB	Text	Kas 9 2015	-rw-rw-r-
Trash	► scalarTransportFoam	3 items	Folder	Kas 9 2015	drwxrwxr-x
Devices	► combustion	7 items	Folder	Kas 9 2015	drwxrwxr-x
SANDISK	► compressible	4 items	Folder	Kas 9 2015	drwxrwxr-x
Computer	► discreteMethods	2 items	Folder	Kas 9 2015	drwxrwxr-x
Network	► DNS	1 item	Folder	Kas 9 2015	drwxrwxr-x
	► electromagnetics	3 items	Folder	Kas 9 2015	drwxrwxr-x
	► financial	1 item	Folder	Kas 9 2015	drwxrwxr-x
	► heatTransfer	6 items	Folder	Kas 9 2015	drwxrwxr-x
	► incompressible	8 items	Folder	Kas 9 2015	drwxrwxr-x
	► lagrangian	7 items	Folder	Kas 9 2015	drwxrwxr-x
	► multiphase	12 items	Folder	Kas 9 2015	drwxrwxr-x
	► stressAnalysis	2 items	Folder	Kas 9 2015	drwxrwxr-x
	► test	124 items	Folder	Kas 9 2015	drwxrwxr-x
	► utilities	7 items	Folder	Kas 9 2015	drwxrwxr-x
	► Allmake	789 bytes	Program	Kas 9 2015	-rwxrwxr-x

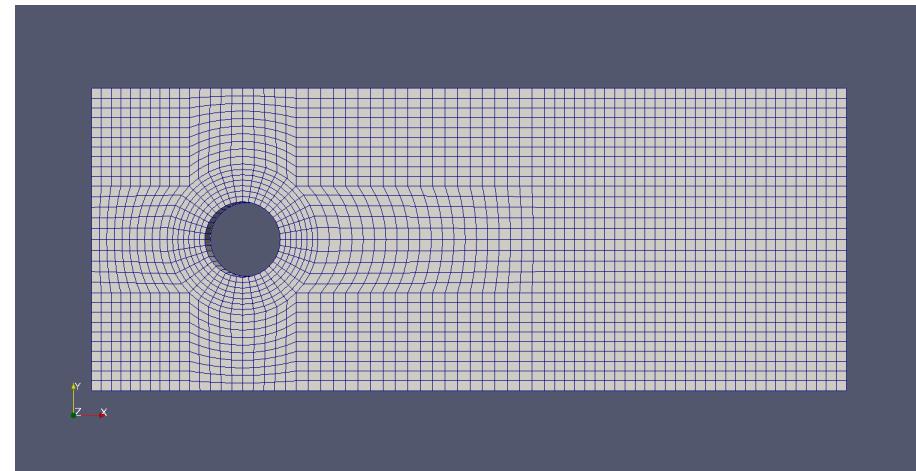
"potentialFoam" selected (containing 4 items)



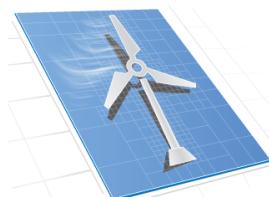
HyperCFD – The Very First Example: Flow Around a Cylinder

Solving Laminar Flow in 2D using icoFoam

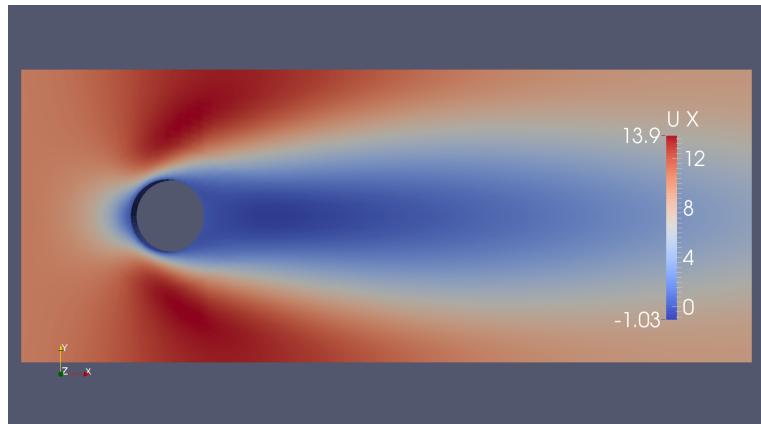
- Time dependent
- Incompressible
- Laminar
- Iso-thermal
- Navier-Stokes solver



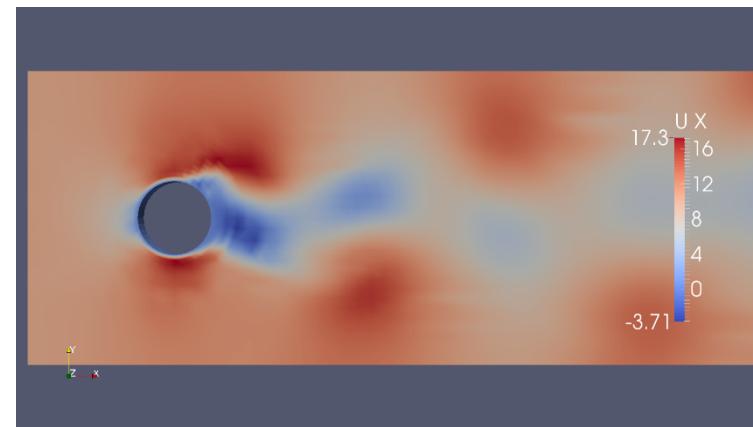
Block structured mesh is generated using
blockMesh



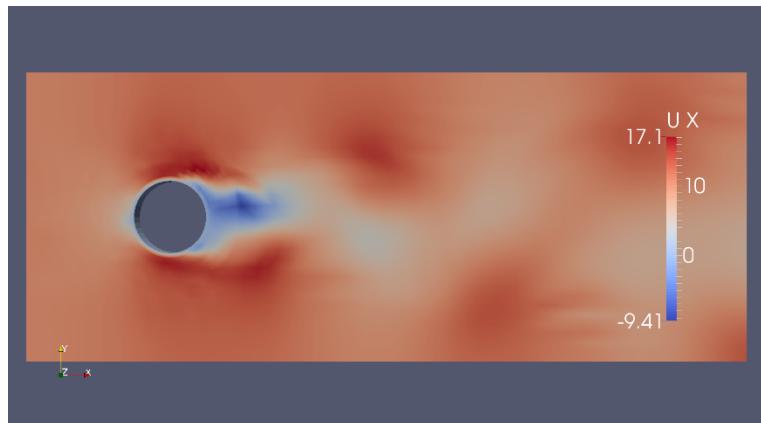
HyperCFD – The Very First Example: Flow Around a Cylinder



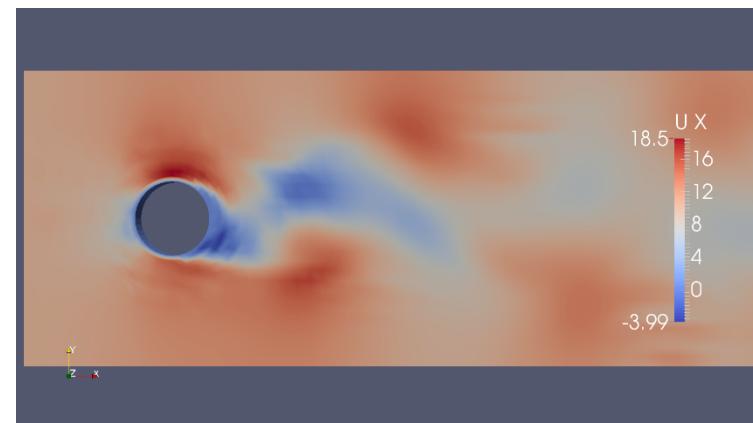
Viscosity = 1e-02



Viscosity = 1e-04



Viscosity = 1e-06



Viscosity = 1e-08

