Open Source CAE for Ship Design and Analysis
Introduction to OpenFOAM

Ummu Saiyidah Najihah binti Zainudin
Abu Hasan Abdullah

December 2019

Outline

1) What is OpenFOAM?
2) Solver - Case Architecture
3) Setup Case Directory - Initial Field Conditions
4) Setup Case Directory - Constant Directory
5) Setup Case Directory - Solving Parameters Directory
6) Advantages of OpenFOAM
7) Run Simple Cases

What is OpenFOAM?

- OpenFOAM Sofware :
  - a free and open source CFD toolbox. OpenFOAM stands for Open source Field Operation And Manipulation.
  - a C++ toolbox for the development of customized numerical solvers, and pre-/post-processing utilities for the solution of continuum mechanics problems, most prominently including computational fluid dynamics.

Introduction to OpenFOAM

1) Solver – Case Architecture

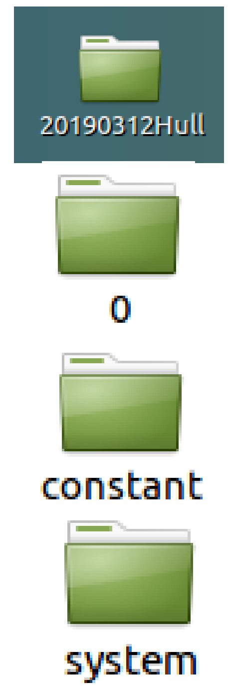
A case is usually composed of three directories. Each dedicated to a specific part of the case.

Case directory

Initial conditions directory

Constant directory

Solving parameters directory



2) Setup case directory – Initial field conditions

```
0
├── alpha.water
├── k
├── nut
├── omega
├── pointDisplacement
├── p_rgh
└── U
```

Initial Field Conditions for :

1- alpha.water

2- k (turbulent kinematics energy)

3- nut (turbulent kinematics viscosity)

4- omega

5- pointDisplacement

6- p_rgh

7- U (speed in m/s)

3) Setup case directory – Constant directory

```
constant
├── g
├── hRef
├── transportProperties
├── triSurface
│   └── DTCHull .stl
└── turbulenceProperties
```

Physical properties values

4) Setup case directory – Solving parameters directory

```
system
├── blockMeshDict
├── controlDict
├── decomposeParDict
├── fvSchemes
├── fvSolution
├── meshQualityDict
├── refineMeshDict
├── setFieldsDict
├── snappyHexMeshDict
├── surfaceFeatureExtractDict
├── topoSetDict.1
├── topoSetDict.2
├── topoSetDict.3
├── topoSetDict.4
├── topoSetDict.5
└── topoSetDict.6
```

Mesh Instruction

Parameters file

Advantages of OpenFOAM software

1) Friendly syntax for partial differential equations.
2) Fully documented source code.
3) Unstructured polyhedral grid capabilities.
4) Automatic parallelization of applications written using OpenFOAM high-level syntax.
5) Wide range of applications and models ready to use.
6) Commercial support and training provided by the developers.
7) No license costs.

Run Simple Cases

THANK YOU