Introduction to OpenFOAM

Beginner's Course

João Castro

joaoctcastro@gmail.com

FOAM@Iberia
2023 5th edition
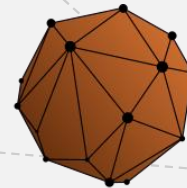
Computational Rheology @IPC

University of Minho
School of Engineering

iPC
INSTITUTE FOR POLYMERS AND COMPOSITES

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.

CRheo@IPC

Contents

1

Software Presentation

What is it?

History

Why use it?

Alternatives

Who uses it?

Where to Find?

CRheo@IPC
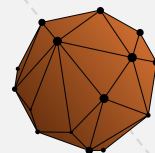
What is it?

Open-source **F**ield **O**peration **A**nd **M**anipulation

      Free software

      Distributed under GPL

      FVM based

      CFD problems

      C++ library

      Growing community

Open▽FOAM®

History

Open▽CFD

Prior to the sale of OpenCFD to SGI Corp, OpenFOAM copyright was transfered to OpenFOAM Foundation

esi get it right

Through the OF Foundation, more OpenFOAM based distributions are released by other companies and communities

Release

Secure Open-source

OpenFOAM was released in 2004 through OpenCFD

OpenFOAM The OpenFOAM Foundation

ESI Group

Other Distributions
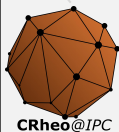
ESI Group acquires OpenCFD in 2012 and maintains OpenFOAM development until today, with two major releases per year

CFD Direct

FOAM-Extend

CRheo@IPC

Why use it?

Free and open-source

Data analysis

Parallel computing

Vast range of features

- Fluid Dynamics

  Compressible and incompressible

  Turbulent and laminar flows

  Single and multiphase flows

  Combustion and chemical reactions

  Heat and mass transfer

- Geometry and meshing

  Mesh generation for complex and simple geometries

  Mesh conversion and manipulation tools

CRheo@IPC

Alternatives

✅ More stable and user-friendly

❌ Cannot assess accuracy
Cannot adapt/modify
Expensive license

Ansys

COMSOL

Moldex3D

SIMULIA

STAR-CCM+

CRheo@IPC

Who uses it?

source: https://discovery.hgdata.com/product/openfoam

Introduction to OpenFOAM

CRheo@IPC

Where to Find?

3 Weeks Introduction to OpenFOAM

https://wiki.openfoam.com/Main_Page

CFD with OpenSource Software
http://www.tfd.chalmers.se/~hani/kurser/OS_CFD/

OpenFOAM Journal

https://journal.openfoam.com

OpenFOAM Governance

https://www.openfoam.com/governance/

OpenFOAM Foundation

https://openfoam.org/

OpenFOAM Wiki

https://wiki.openfoam.com/Main_Page

Unofficial OpenFOAM wiki

https://openfoamwiki.net/

CFD Online

https://www.cfd-online.com/Wiki/OpenFOAM

CFD Direct

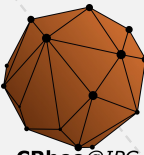https://doc.cfd.direct

CRheo@IPC

2

Finite Volume Method

Description

Domain Discretization

Equation Discretization
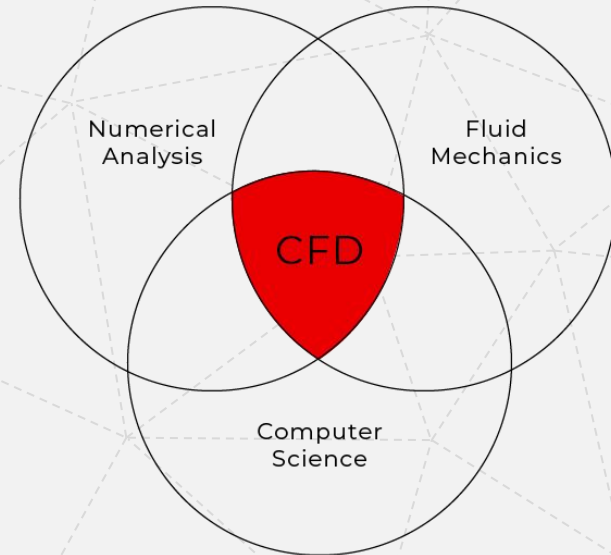
CRheo@IPC

Description

Discretization method

Partial Differential Equations (PDEs)
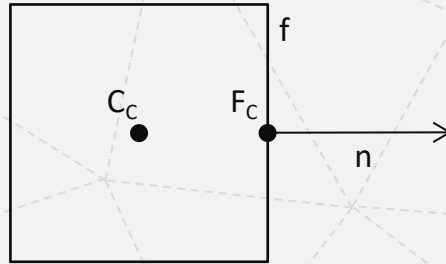
Assures conservation

For a FVM based simulation one must:
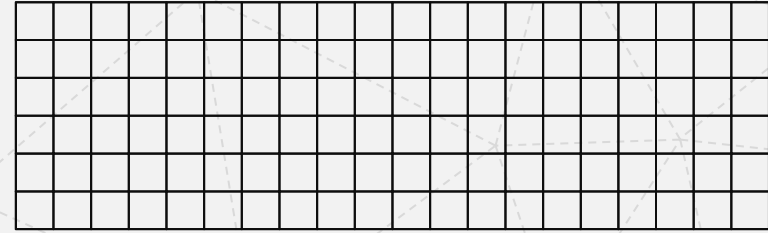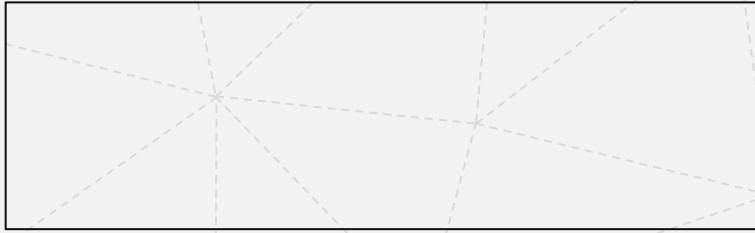
Model: domain and physical phenomena

Discretize: equations and domain

Numerical
Analysis

Fluid
Mechanics

CFD

Computer
Science

CRheo@IPC

Domain Discretization

Subdivide the domain into discrete elements

CRheo@IPC

Equation Discretization

The governing equations cannot be solved by analytical methods

Numerical methods are employed to discretize PDEs over each CV

Conservation equation	Temporal derivative	Advective term	Diffusion term	Source term
Mass	$\frac{\partial \rho}{\partial t}$	$\nabla \cdot (\rho \boldsymbol{u})$		
Momentum	$\frac{\partial (\rho \boldsymbol{u})}{\partial t}$	$\nabla \cdot (\rho \boldsymbol{u} \otimes \boldsymbol{u})$	$\nabla \cdot (\mu \nabla \boldsymbol{u})$	$-\nabla p + \rho \boldsymbol{g} + F$
Energy	$\frac{\partial (\rho C_p T)}{\partial t}$	$\nabla \cdot (\rho C_p T \boldsymbol{u})$	$\nabla \cdot (k \nabla T)$	$\dot{Q}$
General transport	$\frac{\partial (\rho \phi)}{\partial t}$	$\nabla \cdot (\rho \boldsymbol{u} \phi)$	$\nabla \cdot (\Gamma_\phi \nabla \phi)$	S_ϕ

CRheo@IPC

Equation Discretization

Integrate equation over the control volume

$$\int_{t^{n-1}}^{t^n} \left[\int_{CV} \frac{\partial(\rho\phi)}{\partial t}\, d\boldsymbol{x} + \int_{CV} \nabla \cdot (\rho\boldsymbol{u}\phi)d\boldsymbol{x} - \int_{CV} \nabla \cdot (\Gamma_\phi \nabla\phi)d\boldsymbol{x} \right] dt = \int_{t^{n-1}}^{t^n} \left(\int_{CV} S_\phi\, d\boldsymbol{x} \right) dt$$

Gauss Divergence Theorem

$$\int_{CV} \nabla \cdot \boldsymbol{a}\, d\boldsymbol{x} = \oint_{\partial CV} \boldsymbol{n} \cdot \boldsymbol{a}\, d\boldsymbol{x}$$

Because the cell surface area is the sum of its faces area

$$\oint_{\partial CV} \boldsymbol{n} \cdot \boldsymbol{a}\, d\boldsymbol{x} = \sum_f \left(\int_f \boldsymbol{n} \cdot \boldsymbol{a}\, d\boldsymbol{x} \right) \approx \sum_f \boldsymbol{S}_f \boldsymbol{a}$$

Equation Discretization
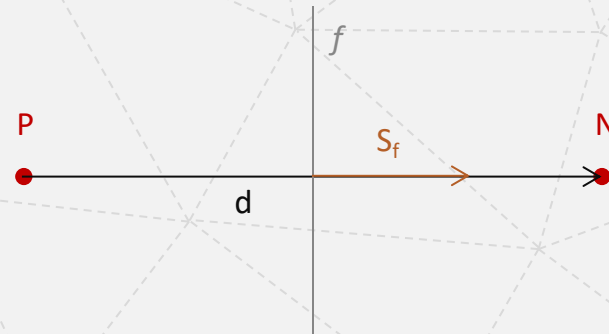
Discretize the diffusion term by applying Gauss' divergence theorem

$$\int_{CV} \nabla \cdot (\Gamma_\phi \nabla \phi) \, d\boldsymbol{x} \qquad \int_{CV} \nabla \cdot \boldsymbol{a} \, d\boldsymbol{x}$$

$$\int_{CV} \nabla \cdot (\Gamma_\phi \nabla \phi) \, d\boldsymbol{x} = \oint_{\partial CV} \boldsymbol{n} \cdot (\Gamma_\phi \nabla \phi) \, d\boldsymbol{x} \approx \sum_f \boldsymbol{S}_f \cdot (\Gamma_\phi \nabla \phi)_f = \sum_f (\Gamma_\phi)_f \boldsymbol{S}_f \cdot (\nabla \phi)_f$$

Gradient of ϕ at the face calculation

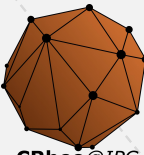$$\boldsymbol{S}_f \cdot (\nabla \phi)_f = |\boldsymbol{S}_f| \frac{\phi_N - \phi_P}{|\boldsymbol{d}|}$$

P S_f N

d

CRheo@IPC

3

How to use OpenFOAM

Pre-requisites

WSL Environment

Case Structure

Running the 1st case

CRheo@IPC

Pre-requisites

OS

Knowledge

Sofware

macOS

CFD

CRheo@IPC
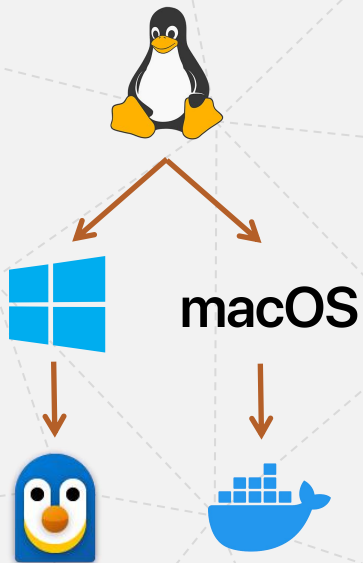
WSL Environment

Download FoamIberia folder to your desktop

Navigate via Linux   > cd path/to/folder | > ls | > pwd

Navigate via Windows   > explorer.exe .

Edit .bashrc   > nano .bashrc or > vim .bashrc

    alias of2306='source /usr/lib/openfoam/openfoam2306/etc/bashrc'

    alias foamIberia='cd /mnt/c/Users/<yourUser>/Desktop/FoamIberia'

    shopt -s direxpand

Load your OpenFOAM   > of2306

Important folders
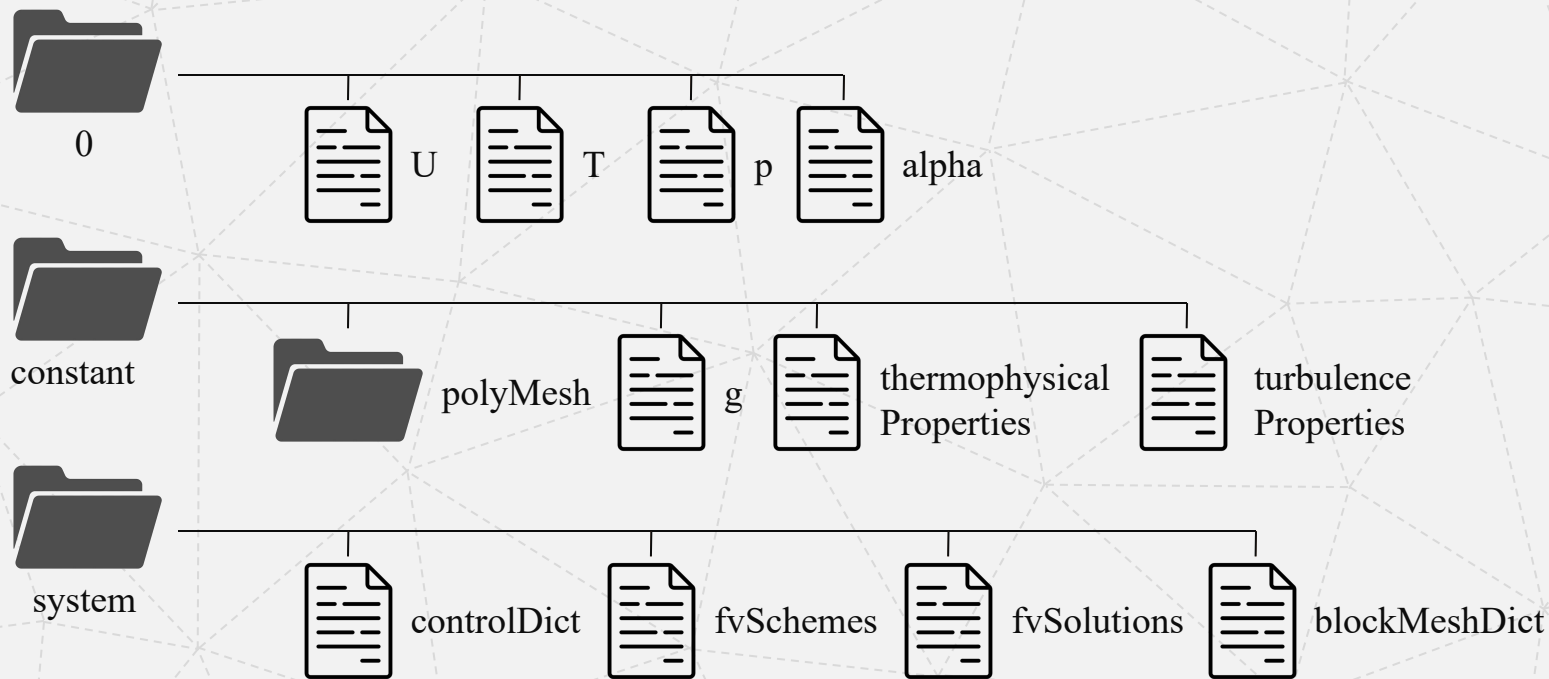
    applications   > cd $FOAM_APP

    src   > cd $FOAM_SRC

    tutorials   > cd $FOAM_TUTORIALS

CRheo@IPC

Case Structure

Running the 1st case

Copy a tutorial case

> cd $FOAM_TUTORIALS

> cp -r incompressible/icoFoam/elbow /mnt/c/Users/<yourUser>/Desktop/FoamIberia/C1

Analyze the copied case

> foamIberia

> cd C1/elbow

Execute the file   > ./Allrun

Paraview visualization   > touch open.foam

Clean case   > ./Allclean

CRheo@IPC

4

Detailed Case Setup

Case Overview

Geometry Definition

Initial Conditions

Simulation Controls

Other files

Run and Analysis

CRheo@IPC
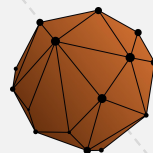
Case Overview

Q: What do we want to simulate?

    A: Liquid flow through a large cavity
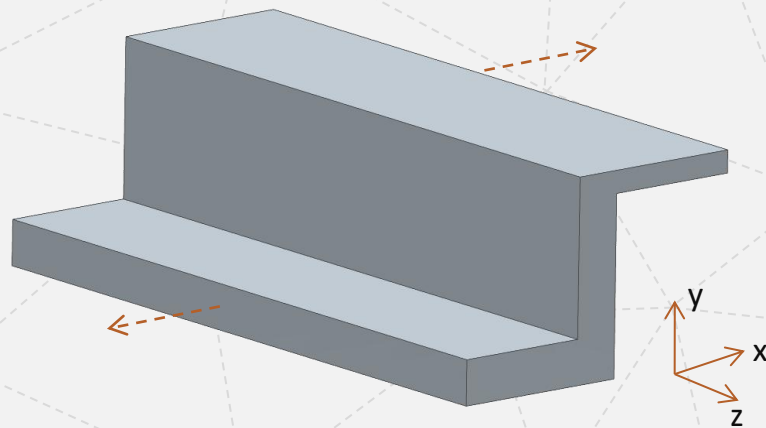
Q: What information do we need?

    A: Velocity profile

Q: What solver should we use?

    A: Single-phase/~~multi-phase~~, ~~compressible~~/incompressible, ~~steady-state~~/transient, velocity/pressure/~~temperature~~
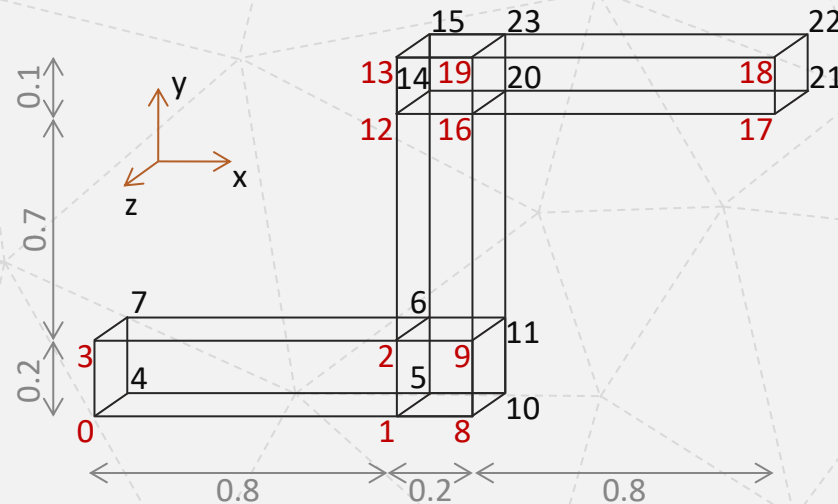
Q: How should we model the domain?

    A: ~~3D case of all domain~~/~~3D case of a portion~~/2D section

CRheo@IPC

Geometry Definition

Vertices

0   (0 0 0)
1   (0.8 0 0)
2   (0.8 0.2 0)
3   (0 0.2 0)
4   (0 0 -0.1)
5   (0.8 0 -0.1)
6   (0.8 0.2 -0.1)
7   (0 0.2 -0.1)

8   (1 0 0)
9   (1 0.2 0)
10 (1 0 -0.1)
11 (1 0.2 -0.1)

12 (0.8 0.9 0)
13 (0.8 1 0)
14 (0.8 0.9 -0.1)
15 (0.8 1 -0.1)

16 (1 0.9 0)
17 (1.8 0.9 0)
18 (1.8 1 0)
19 (1 1 0)
20 (1 0.9 -0.1)
21 (1.8 0.9 -0.1)
22 (1.8 1 -0.1)
23 (1 1 -0.1)

CRheo@IPC

Geometry Definition

Blocks
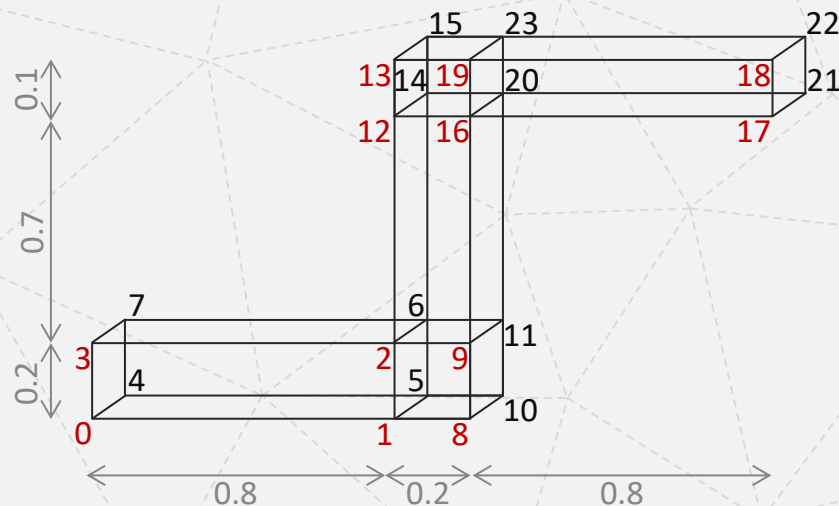
hex (0 1 2 3 4 5 6 7)   (80 20 1)

hex (1 8 9 2 5 10 11 6)   (20 20 1)

hex (2 9 16 12 6 11 20 14)   (20 70 1)
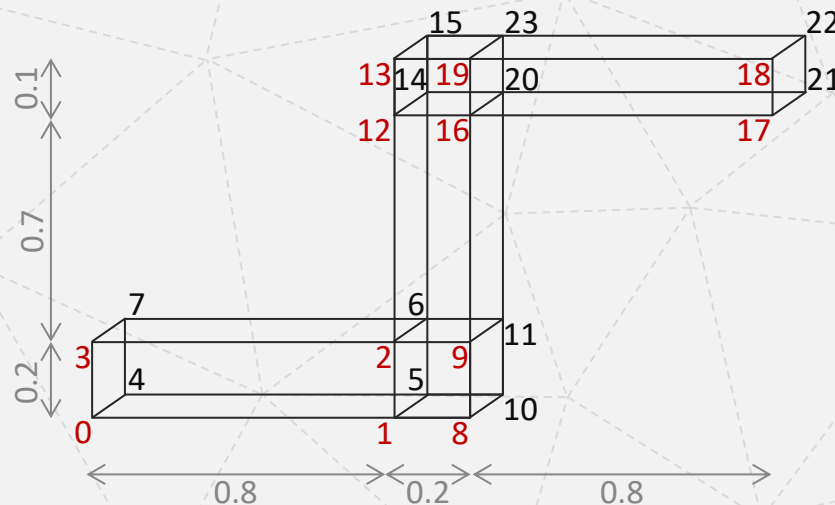
hex (12 16 19 13 14 20 23 15) (20 10 1)

hex (16 17 18 19 20 21 22 23) (80 10 1)

Geometry Definition

Boundary

inlet   (0 3 7 4)   frontAndBack
outlet (17 21 22 18)   (0 1 2 3)
walls (0 4 5 1)   (1 8 9 2)
          (1 5 10 8)   (2 9 16 12)
          (8 10 11 9)   (12 16 19 13)
          (11 20 16 9)   (16 17 18 19)
          (7 3 2 6)   (7 6 5 4)
          (6 14 12 2)   (11 10 5 6)
          (12 13 15 14)   (14 20 11 6)
          (23 15 13 19)   (15 23 20 14)
          (23 19 18 22)   (23 22 21 20)
          (17 16 20 21)

CRheo@IPC

Initial Conditions

Velocity

```
dimensions      [0 0 0 0 0 0 0];        SI [kg m s K mol A cd]
dimensions      [0 1 -1 0 0 0 0];
internalField   uniform (0 0 0);
inlet
  {
    type         fixedValue;
    value        uniform (10 0 0);

  }
```

CRheo@IPC

Initial Conditions

Velocity

outlet
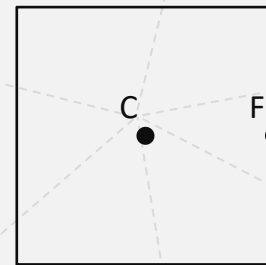
    {

       type   zeroGradient;

    }

walls

    {

       type   fixedValue;

       value   uniform (0 0 0);

    }

C   F

$U_F - U_C = 0$

$U_C = U_F$

frontAndBack

    {

       type   empty;

    }

Initial Conditions

Pressure

```
dimensions      [0 0 0 0 0 0 0];        SI [kg m s K mol A cd]

dimensions      [1 -1 -2 0 0 0 0];

dimensions      [0 2 -2 0 0 0 0];

internalField   uniform 0;

inlet

    {

        type        zeroGradient;

    }
```

CRheo@IPC

Initial Conditions

Pressure

```
outlet
    {
        type        fixedValue;
        value       uniform 0;
    }
walls
    {
        type        zeroGradient;
    }
```

```
frontAndBack
    {
        type        empty;
    }
```

CRheo@IPC

Simulation Controls

application	icoFoam;	runTimeModifiable yes;	
startFrom	startTime; latestTime;	adjustTimeStep	yes;
startTime	0;	maxCo	1;
stopAt	endTime;	maxDeltaT	0.1;
endTime	0.1;	writeFormat	ascii;
deltaT	1e-4;	writePrecision	6;
writeControl	adjustableRunTime;	writeCompression off;	
writeInterval	0.025;	timeFormat	general;
purgeWrite	0;	timePrecision	6;

CRheo@IPC

Other Files

Constant ⟶ tansportProperties ⟶ nu   1;

System ⟶ decomposeParDict

   numberOfSubdomains 4;

   method scotch;

   > lscpu   Cores per socket x Sockets

System ⟶ fvSchemes

System ⟶ fvSolutions

CRheo@IPC

Run and Analysis

Move to cavity folder

> foamIberia > cd C1/cavity

Generate mesh

> blockMesh

View in Paraview

> touch open.foam

Duplicate folder

> cp -r ../cavity ../cavity1

Run simulation

> icoFoam

Delete times

> rm -r 0.*

Change viscosity to 1e-3 and endTime to 0.5

> icoFoam > log.txt

Move to cavity1 folder

> cd ../cavity1

Double mesh refinement

> decomposePar

> mpirun -np 4 icoFoam -parallel > log.txt

> reconstructPar

CRheo@IPC

END

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

CRheo@IPC