INFORMATICS INSTITUTE

OPENFOAM®'A GİRİŞ ÇALIŞTAYI

Örnek Uygulamalar

Emre Cenk Ersan

ITU Computational Biomechanics Research Group

https://valve.be.itu.edu.tr/

Workshop Supervisor: Prof. Dr. M. Serdar Çelebi

December 1, 2023

Overview

- We will look at the following test cases in this section:

    1. **Cavity (transient – 2D)**
    2. **Elbow (transient – 2D)**
    3. **Fully developed pipe flow (steady-state – 3D)**
        i. **Newtonian fluid**
        ii. **NonNewtonian fluid (shear-thinning Bird-Carreau model for blood)**

Test case: Cavity

- **Physics**

  - Let us assume a two-dimensional flow in a cavity.
  - The fluid flow as following parameters:

    Top wall velocity: $U_x = 1$ m/s

    Cavity edge length: $d = 0.1$ m

    Newtonian

    Laminar flow, Re = 10

    Incompressible

    Isothermal flow

    Transient (unsteady) flow

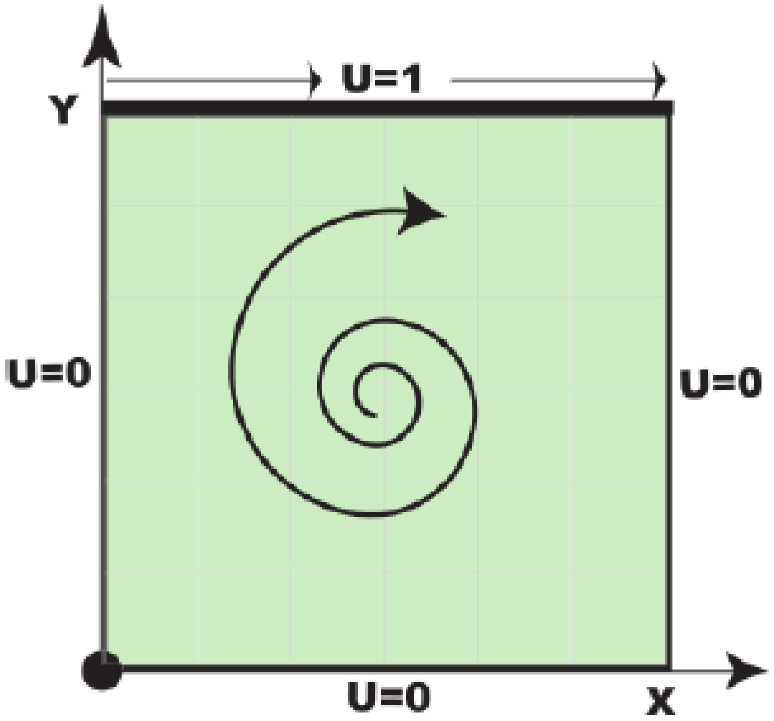

Figure 1: Cavity case geometry

Test case: Cavity

➢ **Working with the case**

- We (might) need to edit the files.
- Test case directory contains the mesh and all configuration files with boundary conditions, initial conditions, model constants, solver parameters, etc.
- Typical minimal configuration of a test case directory:

caseName	test case directory
∟ ***system***	solver parameters
∟ ***controlDict***	computation options
∟ ***fvSchemes***	numerical schemes options
∟ ***fvSolution***	linear solver options
∟ ***constant***	constant physics data
∟ ***polyMesh***	mesh directory
∟ ***transportProperties***	fluid properties
∟ ***turbulenceProperties***	turbulence properties
∟ ***0***	initial conditions

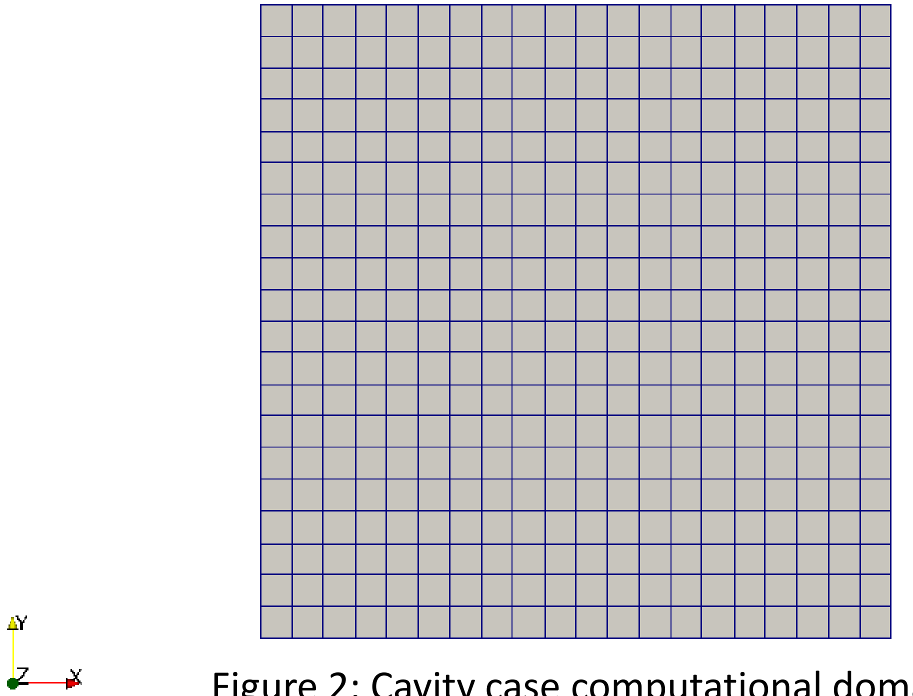

Figure 2: Cavity case computational domain

Test case: Cavity

- **Generating/importing and checking the mesh**
    - As mentioned in the first part of the workshop, we can generate our mesh with a mesh utility like **blockMesh** or **snappyHexMesh**; we can also import our “**.msh**” file with **fluentMeshToFoam**.
    - In this example, we use the **blockMesh** utility with its own dictionary file in ***caseName/system*** folder.
    - We run **blockMesh** in the test case directory and display the information about the generated mesh with the **checkMesh** application.
    - The mesh is located then in ***caseName/constant/polyMesh*** folder.

checkMesh

```
Create time

Create mesh for time = 0

Time = 0

Mesh stats
    points:           882
    internal points:  0
    faces:            1640
    internal faces:   760
    cells:            400
    faces per cell:   6
    boundary patches: 3
    point zones:      0
    face zones:       0
    cell zones:       0

Overall number of cells of each type:
    hexahedra:     400
    prisms:        0
    wedges:        0
    pyramids:      0
    tet wedges:    0
    tetrahedra:    0
    polyhedra:     0

Checking topology...
    Boundary definition OK.
    Cell to face addressing OK.
    Point usage OK.
    Upper triangular ordering OK.
    Face vertices OK.
    Number of regions: 1 (OK).
```

```
Checking patch topology for multiply connected surfaces...
    Patch               Faces    Points   Surface topology
    movingWall          20       42       ok (non-closed singly connected)
    fixedWalls          60       122      ok (non-closed singly connected)
    frontAndBack        800      882      ok (non-closed singly connected)

Checking faceZone topology for multiply connected surfaces...
    No faceZones found.

Checking basic cellZone addressing...
    No cellZones found.

Checking geometry...
    Overall domain bounding box (0 0 0) (0.1 0.1 0.01)
    Mesh has 2 geometric (non-empty/wedge) directions (1 1 0)
    Mesh has 2 solution (non-empty) directions (1 1 0)
    All edges aligned with or perpendicular to non-empty directions.
    Boundary openness (8.47033e-18 -8.47033e-18 -4.51751e-17) OK.
    Max cell openness = 1.35525e-16 OK.
    Max aspect ratio = 1 OK.
    Minimum face area = 2.5e-05. Maximum face area = 5e-05.  Face area magnitudes OK.
    Min volume = 2.5e-07. Max volume = 2.5e-07.  Total volume = 0.0001.  Cell volumes OK.
    Mesh non-orthogonality Max: 0 average: 0
    Non-orthogonality check OK.
    Face pyramids OK.
    Max skewness = 1.66533e-14 OK.
    Coupled point location match (average 0) OK.

Mesh OK.

End
```

blockMeshDict

```
scale   0.1; //Scale coordinates

vertices
(
    (0 0 0)   //vertex 0
    (1 0 0)   //vertex 1
    (1 1 0)   //vertex 2
    (0 1 0)   //vertex 3
    (0 0 0.1) //vertex 4
    (1 0 0.1) //vertex 5
    (1 1 0.1) //vertex 6
    (0 1 0.1) //vertex 7
);

blocks
( //Definition of the hexahedral block. Pay attention to the numbering.
    hex (0 1 2 3 4 5 6 7) (20 20 1) simpleGrading (1 1 1) //Uniform gradient
);
    //20 cells in x-direction, 20 cells in y-direction,
    //and only 1 cell in z-direction bcs the simulation will be 2D.

edges //This is simple geometry, edges can be empty.
(
);
boundary //Definition of boundary of the domain to apply the BC.
(
    movingWall
    {
        type wall;
        faces
        (
            (3 7 6 2)
        );
    }
    fixedWalls
    {
        type wall;
        faces
        (
            (0 4 7 3)
            (2 6 5 1)
            (1 5 4 0)
        );
    }
    frontAndBack
    {
        type empty; // Faces normal to Oz are "empty" to specify that the simulation is 2D.
        faces
        (
            (0 3 2 1)
            (4 5 6 7)
        );
    }
);
```

Test case: Cavity

- **Boundary and initial conditions**
  - Initial and boundary conditions are located in directories named with numbers.
  - The number in directory means the physical time layer of the solution.
  - Usually, the initial and boundary conditions are stored in directory ***caseName/0***.
  - For the incompressible laminar Newtonian transient fluid flow, we will use **icoFoam** which requires initial pressure (***caseName/0/p***) and velocity (***caseName/0/U***) conditions.

```
FoamFile
{
    version     2.0;
    format      ascii;
    class       volScalarField;
    object      p;
}
// * * * * * * * * * * * * * * * * * * * *

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

internalField   uniform 0;

boundaryField
{
    movingWall
    {
        type            zeroGradient;
    }

    fixedWalls
    {
        type            zeroGradient;
    }

    frontAndBack
    {
        type            empty;
    }
}


// *************************************
```

Pressure conditions

```
FoamFile
{
    version     2.0;
    format      ascii;
    class       volVectorField;
    object      U;
}
// * * * * * * * * * * * * * * * * * * * * * *

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

internalField   uniform (0 0 0);

boundaryField
{
    movingWall
    {
        type            fixedValue;
        value           uniform (1 0 0);
    }

    fixedWalls
    {
        type            noSlip;
    }

    frontAndBack
    {
        type            empty;
    }
}
```

Velocity conditions

Test case: Cavity

➢ Physical properties

- The physical properties of the fluid flow are located in directory ***caseName/constant***.
- Inside this directory we find the mesh and the configuration file named **transportProperties** (may also include **turbulenceProperties**, depends on the problem and the solver)
- In this problem, the fluid is Newtonian, so we need only one parameter which is the **kinematic viscosity** ***υ (nu)*** in **m²/s**.
- We also need to check the Reynolds number, since we defined the case as laminar.

$$Re = \frac{U \cdot d}{\upsilon} = \frac{1.0 \cdot 0.1}{0.01} = 10$$

```
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    object      transportProperties;
}
// * * * * * * * * * * * * * * * * * * * //

nu              0.01;
```

transportProperties

➢ Computation setup

- Parameters of computational process are set in files inside ***caseName/system*** directory.
- The file **controlDict** controls the computation in terms of time interval, time step, writing data etc.

```
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    object      controlDict;
}
// * * * * * * * * * * * * * * * * //

application     icoFoam;

startFrom       startTime;

startTime       0;

stopAt          endTime;

endTime         0.5;

deltaT          0.005;

writeControl    timeStep;

writeInterval   2;

purgeWrite      0;

writeFormat     ascii;

writePrecision  6;

writeCompression off;

timeFormat      general;

timePrecision   6;

runTimeModifiable true;
```

controlDict

We use icoFoam.

The simulation starts from $t = 0\ s$ and ends at $t = 0.5\ s$.

The time step size is $\Delta t = 0.005\ s$.

The solution is saved after each two timesteps.

We are allowed to change **controlDict** during computation.

Test case: Cavity

➢ **Numerical solution setup**

- Parameters of numerical solution are stored in files in files inside ***caseName/system*** directory.
- In file **fvSolution**, there are set parameters of solution of system of linear equations for each quantity and also parameters of numerical methods.
- In file **fvSchemes**, there are setup the numerical schemes for individual quantities and other members.

```
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    object      fvSolution;
}
// * * * * * * * * * * * * * * * * * * * * * * * * //

solvers
{
    p
    {
        solver          PCG;
        preconditioner  DIC;
        tolerance       1e-06;
        relTol          0.05;
    }

    pFinal
    {
        $p;
        relTol          0;
    }

    U
    {
        solver          smoothSolver;
        smoother        symGaussSeidel;
        tolerance       1e-05;
        relTol          0;
    }
}

PISO
{
    nCorrectors     2;
    nNonOrthogonalCorrectors 0;
    pRefCell        0;
    pRefValue       0;
}
```

fvSolution

```
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    object      fvSchemes;
}
// * * * * * * * * * * * * * * * * * * * * * * * * //

ddtSchemes
{
    default         Euler;
}

gradSchemes
{
    default         Gauss linear;
    grad(p)         Gauss linear;
}

divSchemes
{
    default         none;
    div(phi,U)      Gauss linear;
}

laplacianSchemes
{
    default         Gauss linear orthogonal;
}

interpolationSchemes
{
    default         linear;
}

snGradSchemes
{
    default         orthogonal;
}
```

fvSchemes

Test case: Cavity

- **Running the simulation sequentially**

  - We run the simulation sequentially on background and save the log file with the command

    ```
    # icoFoam > log.icofoam &
    ```

  - We can watch the log file with

    ```
    # tail –f log.icofoam
    ```

```
/*---------------------------------------------------------------------------*\
| =========                 |                                                 |
| \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
|  \\    /   O peration     | Version:  2206                                  |
|   \\  /    A nd           | Website:  www.openfoam.com                      |
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
Build  : _76d719d1-20220624 OPENFOAM=2206 version=2206
Arch   : "LSB;label=32;scalar=64"
Exec   : icoFoam
Date   : Nov 30 2023
Time   : 15:03:34
Host   : jcjnb
PID    : 46353
I/O    : uncollated
Case   : /home/jcj/CALISTAY/1_cavity
nProcs : 1
trapFpe: Floating point exception trapping enabled (FOAM_SIGFPE).
fileModificationChecking : Monitoring run-time modified files using timeStampMaster (fileModificationSkew 5, maxFileModificationPolls 20)
allowSystemOperations : Allowing user-supplied system call operations

// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //
Create time

Create mesh for time = 0


PISO: Operating solver in PISO mode

Reading transportProperties

Reading field p

Reading field U

Reading/calculating face flux field phi
```

Log file, initialization

```
Starting time loop

Time = 0.005

Courant Number mean: 0 max: 0
smoothSolver:  Solving for Ux, Initial residual = 1, Final residual = 8.90511e-06, No Iterations 19
smoothSolver:  Solving for Uy, Initial residual = 0, Final residual = 0, No Iterations 0
DICPCG:  Solving for p, Initial residual = 1, Final residual = 0.0492854, No Iterations 12
time step continuity errors : sum local = 0.000466513, global = -1.79995e-19, cumulative = -1.79995e-19
DICPCG:  Solving for p, Initial residual = 0.590864, Final residual = 2.65225e-07, No Iterations 35
time step continuity errors : sum local = 2.74685e-09, global = -2.6445e-19, cumulative = -4.44444e-19
ExecutionTime = 0.01 s  ClockTime = 0 s

Time = 0.01

Courant Number mean: 0.0976825 max: 0.585607
smoothSolver:  Solving for Ux, Initial residual = 0.160686, Final residual = 6.83031e-06, No Iterations 19
smoothSolver:  Solving for Uy, Initial residual = 0.260828, Final residual = 9.65939e-06, No Iterations 18
DICPCG:  Solving for p, Initial residual = 0.428925, Final residual = 0.0103739, No Iterations 22
time step continuity errors : sum local = 0.000110788, global = 3.77194e-19, cumulative = -6.72498e-20
DICPCG:  Solving for p, Initial residual = 0.30209, Final residual = 5.26569e-07, No Iterations 33
time step continuity errors : sum local = 6.61987e-09, global = -2.74872e-19, cumulative = -3.42122e-19
ExecutionTime = 0.01 s  ClockTime = 0 s
```

Log file, first two timesteps

```
Time = 0.495

Courant Number mean: 0.222158 max: 0.852134
smoothSolver:  Solving for Ux, Initial residual = 2.43518e-07, Final residual = 2.43518e-07, No Iterations 0
smoothSolver:  Solving for Uy, Initial residual = 5.33314e-07, Final residual = 5.33314e-07, No Iterations 0
DICPCG:  Solving for p, Initial residual = 5.25763e-07, Final residual = 5.25763e-07, No Iterations 0
time step continuity errors : sum local = 6.11228e-09, global = 1.51821e-18, cumulative = 1.09644e-17
DICPCG:  Solving for p, Initial residual = 7.01015e-07, Final residual = 7.01015e-07, No Iterations 0
time step continuity errors : sum local = 7.5484e-09, global = -4.1723e-19, cumulative = 1.05471e-17
ExecutionTime = 0.16 s  ClockTime = 0 s

Time = 0.5

Courant Number mean: 0.222158 max: 0.852134
smoothSolver:  Solving for Ux, Initial residual = 2.3091e-07, Final residual = 2.3091e-07, No Iterations 0
smoothSolver:  Solving for Uy, Initial residual = 5.0684e-07, Final residual = 5.0684e-07, No Iterations 0
DICPCG:  Solving for p, Initial residual = 8.63844e-07, Final residual = 8.63844e-07, No Iterations 0
time step continuity errors : sum local = 8.8828e-09, global = 4.94571e-19, cumulative = 1.10417e-17
DICPCG:  Solving for p, Initial residual = 9.59103e-07, Final residual = 9.59103e-07, No Iterations 0
time step continuity errors : sum local = 9.66354e-09, global = 1.13175e-18, cumulative = 1.21735e-17
ExecutionTime = 0.16 s  ClockTime = 0 s

End
```

Log file, last two timesteps

Test case: Cavity

- **Iteration output**

```
Time = 0.495

Courant Number mean: 0.222158 max: 0.852134
smoothSolver:  Solving for Ux, Initial residual = 2.43518e-07, Final residual = 2.43518e-07, No Iterations 0
smoothSolver:  Solving for Uy, Initial residual = 5.33314e-07, Final residual = 5.33314e-07, No Iterations 0
DICPCG:  Solving for p, Initial residual = 5.25763e-07, Final residual = 5.25763e-07, No Iterations 0
time step continuity errors : sum local = 6.11228e-09, global = 1.51821e-18, cumulative = 1.09644e-17
DICPCG:  Solving for p, Initial residual = 7.01015e-07, Final residual = 7.01015e-07, No Iterations 0
time step continuity errors : sum local = 7.5484e-09, global = -4.1723e-19, cumulative = 1.05471e-17
ExecutionTime = 0.16 s  ClockTime = 0 s
```

  - Take a closer look at time 0.495 s.
  - **Courant Number** mean/max are the mean/maximal **CFL number** for **convergence condition**, related to the well-known formula $CFL = \frac{|U| \cdot \Delta t}{\Delta x}$.
  - **smoothSolver** and **DICPCG** are the linear system solvers for velocity components and pressure, respectively, which are set up in **fvSolution**.
  - **Initial residual** is a measure of stability of solution with respect to previous iteration, while **Final residual** defines the residual of the system at the end of that timestep.
  - **No Iterations** is the number of iterations of the linear system solver.
  - **Time step continuity errors** is continuity equation error, basically it is a sum of fluxes over all mesh faces and ideally should be zero.
  - **ExecutionTime** is the time elapsed since start of the computation.
  - **ClockTime** is the time spent on processor(s).

Test case: Elbow

- **Physics**

  - Two-dimensional flow in an elbow geometry with two inlets and one outlet.

  - The fluid flow as following parameters:

    Inlet 1 velocity: U_x = 1 m/s, Inlet 2 velocity: U_y = 3 m/s

    Inlet 1 and outlet diameter: D_1 = 16 m

    Inlet 2 diameter: D_1 = 4 m, length in x-axis: L = 64 m

    Newtonian

    Laminar flow, Re_{D_1} = 1600

    Incompressible

    Isothermal flow

    Transient (unsteady) flow

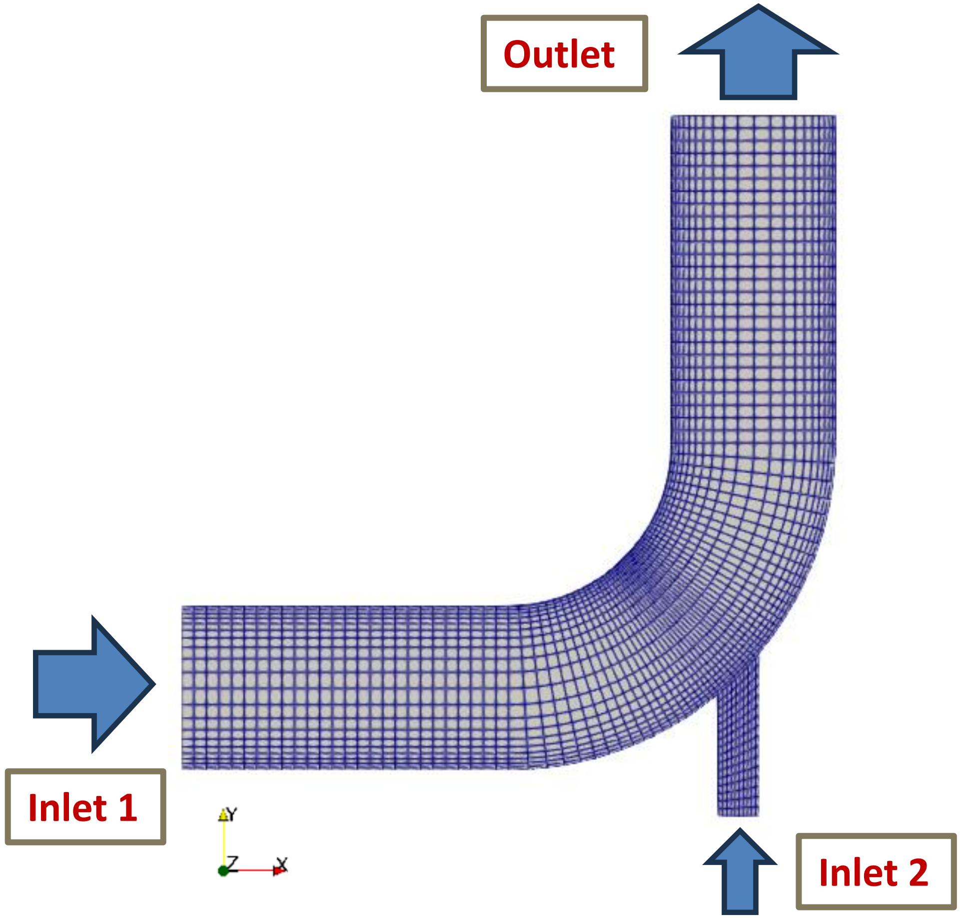

Figure 3: Elbow case geometry and quad mesh

Test case: Elbow

checkMesh

```
Create mesh for time = 0

Time = 0

Mesh stats
    points:           4678
    internal points:  0
    faces:            8938
    internal faces:   4262
    cells:            2200
    faces per cell:   6
    boundary patches: 5
    point zones:      0
    face zones:       0
    cell zones:       0

Overall number of cells of each type:
    hexahedra:     2200
    prisms:        0
    wedges:        0
    pyramids:      0
    tet wedges:    0
    tetrahedra:    0
    polyhedra:     0

Checking topology...
    Boundary definition OK.
    Cell to face addressing OK.
    Point usage OK.
    Upper triangular ordering OK.
    Face vertices OK.
    Number of regions: 1 (OK).

Checking patch topology for multiply connected surfaces...
    Patch               Faces    Points   Surface topology
    wall-4              226      458      ok (non-closed singly connected)
    pressure-outlet-7   20       42       ok (non-closed singly connected)
    velocity-inlet-6    10       22       ok (non-closed singly connected)
    velocity-inlet-5    20       42       ok (non-closed singly connected)
    frontAndBackPlanes  4400     4678     ok (non-closed singly connected)

Checking faceZone topology for multiply connected surfaces...
    No faceZones found.

Checking basic cellZone addressing...
    No cellZones found.

Checking geometry...
    Overall domain bounding box (-32 -36.5385 -0.937738) (32 32 0.937738)
    Mesh has 2 geometric (non-empty/wedge) directions (1 1 0)
    Mesh has 2 solution (non-empty) directions (1 1 0)
    All edges aligned with or perpendicular to non-empty directions.
    Boundary openness (7.19807e-19 -9.78938e-19 1.15169e-18) OK.
    Max cell openness = 2.06735e-16 OK.
    Max aspect ratio = 5.34145 OK.
    Minimum face area = 0.0623165. Maximum face area = 3.68914.  Face area magnitudes OK.
    Min volume = 0.116873. Max volume = 4.57578.  Total volume = 3155.59.  Cell volumes OK.
    Mesh non-orthogonality Max: 48.197 average: 7.57964
    Non-orthogonality check OK.
    Face pyramids OK.
    Max skewness = 0.477427 OK.
    Coupled point location match (average 0) OK.

Mesh OK.

End
```

```
FoamFile
{
    version     2.0;
    format      ascii;
    class       volScalarField;
    object      p;
}
// * * * * * * * * * * * * * * * * * * * *

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

internalField   uniform 0;

boundaryField
{
    wall-4
    {
        type            zeroGradient;
    }

    velocity-inlet-5
    {
        type            zeroGradient;
    }

    velocity-inlet-6
    {
        type            zeroGradient;
    }

    pressure-outlet-7
    {
        type            fixedValue;
        value           uniform 0;
    }

    wall-8
    {
        type            zeroGradient;
    }

    frontAndBackPlanes
    {
        type            empty;
    }
}
```

Pressure conditions

```
FoamFile
{
    version     2.0;
    format      ascii;
    class       volVectorField;
    object      U;
}
// * * * * * * * * * * * * * * * * * * * *

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

internalField   uniform (0 0 0);

boundaryField
{
    wall-4
    {
        type            noSlip;
    }

    velocity-inlet-5
    {
        type            fixedValue;
        value           uniform (1 0 0);
    }

    velocity-inlet-6
    {
        type            fixedValue;
        value           uniform (0 3 0);
    }

    pressure-outlet-7
    {
        type            zeroGradient;
    }

    wall-8
    {
        type            noSlip;
    }

    frontAndBackPlanes
    {
        type            empty;
    }
}
```

Velocity conditions

```
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    object      transportProperties;
}
// * * * * * * * * * * * * * * * * *

nu              0.01;
```

transportProperties

```
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    object      controlDict;
}
// * * * * * * * * * * * * * *

application     icoFoam;

startFrom       latestTime;

startTime       0;

stopAt          endTime;

endTime         500;

deltaT          0.05;

writeControl    timeStep;

writeInterval   100;

purgeWrite      0;

writeFormat     ascii;

writePrecision  6;

writeCompression off;

timeFormat      general;

timePrecision   6;

runTimeModifiable true;
```

controlDict

Test case: Elbow

```
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    object      fvSolution;
}
// * * * * * * * * * * * * * * * * * * * * * //

solvers
{
    p
    {
        solver          PCG;
        preconditioner  DIC;
        tolerance       1e-06;
        relTol          0;
    }

    pFinal
    {
        $p;
        relTol          0;
    }

    U
    {
        solver          smoothSolver;
        smoother        symGaussSeidel;
        tolerance       1e-06;
        relTol          0;
    }
}

PISO
{
    nCorrectors     2;
    nNonOrthogonalCorrectors 2;
}
```

fvSolution

```
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    object      fvSchemes;
}
// * * * * * * * * * * * * * * * * * * * * * * //

ddtSchemes
{
    default         Euler;
}

gradSchemes
{
    default         Gauss linear;
}

divSchemes
{
    default         Gauss linear;

}

laplacianSchemes
{
    default         Gauss linear corrected;
}

interpolationSchemes
{
    default         linear;
}

snGradSchemes
{
    default         corrected;
}
```

fvSchemes

- **Running the simulation in parallel**
  - To run the simulation in parallel, first, we need decompose the computational domain.

```
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    object      decomposeParDict;
}
// * * * * * * * * * * * * * * * * //

numberOfSubdomains  4;

method              scotch;
```

To decompose the domain, we use the command

`# decomposePar`

  - We run the simulation in parallel on background and save the log file with the command

`# mpirun -np 4 icoFoam -parallel > log.icofoam &`

Test case: Elbow

```
Build  : _76d719d1-20220624 OPENFOAM=2206 version=2206
Arch   : "LSB;label=32;scalar=64"
Exec   : icoFoam -parallel
Date   : Nov 30 2023
Time   : 04:56:21
Host   : jcjnb
PID    : 40965
I/O    : uncollated
Case   : /home/jcj/CALISTAY/3_2_elbow_quad
nProcs : 4
Hosts  :
(
    (jcjnb 4)
)
Pstream initialized with:
    floatTransfer      : 0
    nProcsSimpleSum    : 0
    commsType          : nonBlocking
    polling iterations : 0
trapFpe: Floating point exception trapping enabled (FOAM_SIGFPE).
fileModificationChecking : Monitoring run-time modified files using timeStampMaster (fileModificationSkew 5, maxFileModificationPolls 20)
allowSystemOperations : Allowing user-supplied system call operations

// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //
Create time

Create mesh for time = 0


PISO: Operating solver in PISO mode

Reading transportProperties

Reading field p

Reading field U

Reading/calculating face flux field phi
```

Log file, initialization

```
Starting time loop

Time = 0.05

Courant Number mean: 0.000416034 max: 0.0982579
smoothSolver:  Solving for Ux, Initial residual = 1, Final residual = 2.42637e-11, No Iterations 2
smoothSolver:  Solving for Uy, Initial residual = 1, Final residual = 1.34815e-10, No Iterations 2
DICPCG:  Solving for p, Initial residual = 1, Final residual = 9.85997e-07, No Iterations 104
DICPCG:  Solving for p, Initial residual = 0.00940477, Final residual = 9.01899e-07, No Iterations 93
DICPCG:  Solving for p, Initial residual = 0.00251053, Final residual = 9.36394e-07, No Iterations 88
time step continuity errors : sum local = 1.23326e-07, global = -1.11068e-09, cumulative = -1.11068e-09
DICPCG:  Solving for p, Initial residual = 0.00199114, Final residual = 9.62667e-07, No Iterations 87
DICPCG:  Solving for p, Initial residual = 0.000954229, Final residual = 7.50071e-07, No Iterations 86
DICPCG:  Solving for p, Initial residual = 0.000535386, Final residual = 9.93675e-07, No Iterations 82
time step continuity errors : sum local = 1.31772e-07, global = -3.13951e-09, cumulative = -4.2502e-09
ExecutionTime = 0.03 s  ClockTime = 0 s

Time = 0.1

Courant Number mean: 0.0776366 max: 0.692664
smoothSolver:  Solving for Ux, Initial residual = 0.410648, Final residual = 6.21884e-07, No Iterations 5
smoothSolver:  Solving for Uy, Initial residual = 0.409889, Final residual = 5.64684e-07, No Iterations 5
DICPCG:  Solving for p, Initial residual = 0.0120064, Final residual = 9.65585e-07, No Iterations 95
DICPCG:  Solving for p, Initial residual = 0.284778, Final residual = 7.11365e-07, No Iterations 102
DICPCG:  Solving for p, Initial residual = 0.116282, Final residual = 7.05207e-07, No Iterations 100
time step continuity errors : sum local = 1.94665e-09, global = -1.45069e-11, cumulative = -4.2647e-09
DICPCG:  Solving for p, Initial residual = 0.188029, Final residual = 7.83351e-07, No Iterations 100
DICPCG:  Solving for p, Initial residual = 0.062367, Final residual = 9.77574e-07, No Iterations 95
DICPCG:  Solving for p, Initial residual = 0.0389483, Final residual = 9.97815e-07, No Iterations 96
time step continuity errors : sum local = 1.80072e-09, global = -7.70661e-12, cumulative = -4.27241e-09
ExecutionTime = 0.04 s  ClockTime = 0 s
```

Log file, first two timesteps

```
Time = 499.95

Courant Number mean: 0.0818688 max: 0.600555
smoothSolver:  Solving for Ux, Initial residual = 0.000272859, Final residual = 8.53207e-07, No Iterations 1
smoothSolver:  Solving for Uy, Initial residual = 0.000183287, Final residual = 4.95606e-07, No Iterations 1
DICPCG:  Solving for p, Initial residual = 0.00223091, Final residual = 8.36427e-07, No Iterations 80
DICPCG:  Solving for p, Initial residual = 6.54528e-06, Final residual = 9.90617e-07, No Iterations 2
DICPCG:  Solving for p, Initial residual = 1.31079e-06, Final residual = 7.50271e-07, No Iterations 1
time step continuity errors : sum local = 2.27207e-10, global = 3.90036e-13, cumulative = 5.67177e-09
DICPCG:  Solving for p, Initial residual = 5.80851e-05, Final residual = 9.63296e-07, No Iterations 8
DICPCG:  Solving for p, Initial residual = 1.51691e-06, Final residual = 7.53125e-07, No Iterations 1
DICPCG:  Solving for p, Initial residual = 8.6013e-07, Final residual = 8.6013e-07, No Iterations 0
time step continuity errors : sum local = 2.60485e-10, global = -2.65115e-11, cumulative = 5.64526e-09
ExecutionTime = 45.41 s  ClockTime = 45 s

Time = 500

Courant Number mean: 0.0818695 max: 0.600558
smoothSolver:  Solving for Ux, Initial residual = 0.000272837, Final residual = 8.54738e-07, No Iterations 1
smoothSolver:  Solving for Uy, Initial residual = 0.000183435, Final residual = 4.94544e-07, No Iterations 1
DICPCG:  Solving for p, Initial residual = 0.00220913, Final residual = 7.44935e-07, No Iterations 85
DICPCG:  Solving for p, Initial residual = 5.98304e-06, Final residual = 5.86548e-07, No Iterations 3
DICPCG:  Solving for p, Initial residual = 9.87917e-07, Final residual = 9.87917e-07, No Iterations 0
time step continuity errors : sum local = 2.98907e-10, global = 6.55367e-13, cumulative = 5.64591e-09
DICPCG:  Solving for p, Initial residual = 5.98826e-05, Final residual = 9.13068e-07, No Iterations 8
DICPCG:  Solving for p, Initial residual = 1.51504e-06, Final residual = 7.53873e-07, No Iterations 1
DICPCG:  Solving for p, Initial residual = 8.70743e-07, Final residual = 8.70743e-07, No Iterations 0
time step continuity errors : sum local = 2.63466e-10, global = -2.68716e-11, cumulative = 5.61904e-09
ExecutionTime = 45.42 s  ClockTime = 45 s

End

Finalising parallel run
```

Log file, last two timesteps

- To combine the results of a decomposed case run in parallel, we use the command

```
# reconstructPar
```

which will reconstruct all time or solution steps and save the corresponding directories in the case directory.

Test case: Fully-developed pipe flow

- **Physics**

  - Three-dimensional flow through a cylindrical pipe geometry with an inlet and outlet.

  - The fluid flow as following parameters:

    Inlet velocity: $U_z = 0.05$ m/s

    Pipe diameter: D = 0.01 m, pipe length: L = 0.1 m

    Blood flow – two cases (**Newtonian** & **Bird-Carreau shear-thinning model)**

    Laminar flow, Re ≈ 150

    Incompressible

    Isothermal flow

    Steady-state

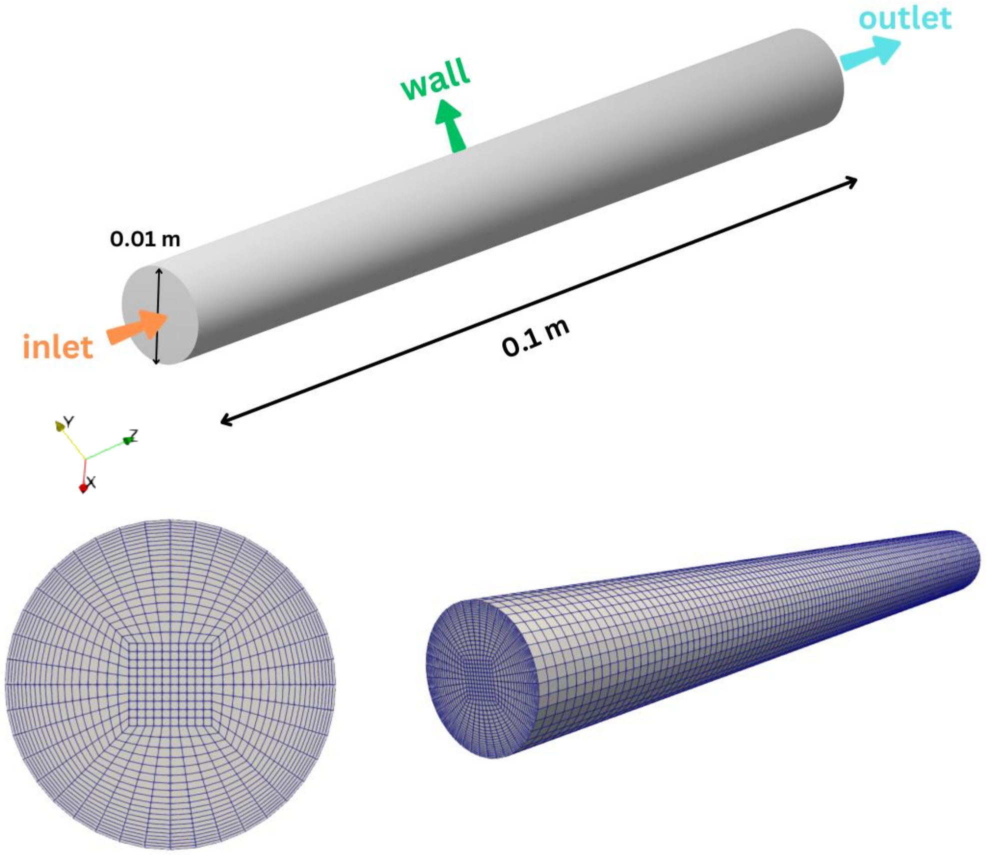

Figure 4: Pipe geometry and hexahedral mesh with o-grid structure.

Test case: Fully-developed pipe flow

```
Create time

Create mesh for time = 0

Time = 0

Mesh stats
    points:           111441
    faces:            327300
    internal faces:   320700
    cells:            108000
    faces per cell:   6
    boundary patches: 3
    point zones:      0
    face zones:       0
    cell zones:       0

Overall number of cells of each type:
    hexahedra:     108000
    prisms:        0
    wedges:        0
    pyramids:      0
    tet wedges:    0
    tetrahedra:    0
    polyhedra:     0

Checking topology...
    Boundary definition OK.
    Cell to face addressing OK.
    Point usage OK.
    Upper triangular ordering OK.
    Face vertices OK.
    Number of regions: 1 (OK).

Checking patch topology for multiply connected surfaces...
    Patch               Faces    Points   Surface topology
    inlet               900      921      ok (non-closed singly connected)
    outlet              900      921      ok (non-closed singly connected)
    wall                4800     4840     ok (non-closed singly connected)

Checking faceZone topology for multiply connected surfaces...
    No faceZones found.

Checking basic cellZone addressing...
    No cellZones found.

Checking geometry...
    Overall domain bounding box (-0.005 -0.005 0) (0.005 0.005 0.1)
    Mesh has 3 geometric (non-empty/wedge) directions (1 1 1)
    Mesh has 3 solution (non-empty) directions (1 1 1)
    Boundary openness (6.88787e-16 6.73362e-16 1.51325e-16) OK.
    Max cell openness = 2.07351e-16 OK.
    Max aspect ratio = 9.83512 OK.
    Minimum face area = 5.88846e-08. Maximum face area = 6.53826e-07.  Face area magnitudes OK.
    Min volume = 4.90705e-11. Max volume = 9.48379e-11.  Total volume = 7.82172e-06.  Cell volumes OK.
    Mesh non-orthogonality Max: 30.3305 average: 5.36658
    Non-orthogonality check OK.
    Face pyramids OK.
    Max skewness = 0.620784 OK.
    Coupled point location match (average 0) OK.

Mesh OK.

End
```

checkMesh

```
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    object      controlDict;
}
// * * * * * * * * * * * * * * * * * //

application     simpleFoam;

startFrom       startTime;

startTime       0;

stopAt          endTime;

endTime         1000;

deltaT          1;

writeControl    timeStep;

writeInterval   100;

purgeWrite      0;

writeFormat     ascii;

writePrecision  6;

writeCompression off;

timeFormat      general;

timePrecision   6;

runTimeModifiable true;
```

controlDict

```
FoamFile
{
    version     2.0;
    format      ascii;
    class       volScalarField;
    object      p;
}
// * * * * * * * * * * * * * * * * * * * * //

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

internalField   uniform 0;

boundaryField
{
    wall
    {
        type            zeroGradient;
    }

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

Pressure conditions

```
FoamFile
{
    version     2.0;
    format      ascii;
    class       volVectorField;
    object      U;
}
// * * * * * * * * * * * * * * * * * * * * //

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

internalField   uniform (0 0 0.05);

boundaryField
{
    wall
    {
        type            fixedValue;
        value           uniform (0 0 0);
    }

    inlet
    {
        type            fixedValue;
        value           uniform (0 0 0.05);
    }
    outlet
    {
        type            zeroGradient;
    }
}
```

Velocity conditions

Test case: Fully-developed pipe flow

- **Blood properties**

  - Blood behaves as a non-Newtonian fluid, which shows all signs of non-Newtonian rheology including shear-thinning, viscoelasticity, and thixotropy.
  - In a non-Newtonian fluid, the shear stresses and the shear rates have a non-linear relation.
  - Two major groups of non-Newtonian fluids are shear thickening and shear thinning fluids.
  - Blood shows shear thinning behavior and decreasing shear rate results in a strong increase of viscosity.

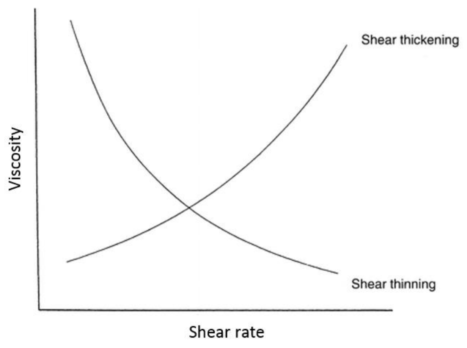

Figure 5: Comparison of shear thickening and shear thinning fluids.

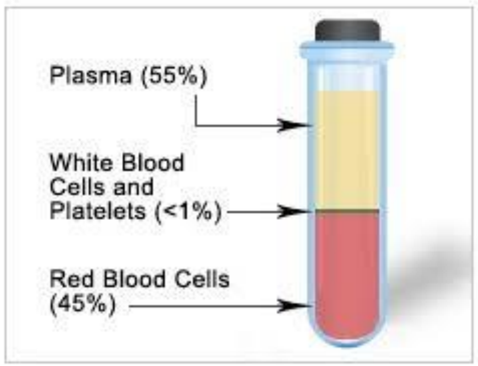

Figure 6: Main components of blood.

Test case: Fully-developed pipe flow

- **Viscosity models**

  - Several mathematical models explain the shear-thinning behaviour of blood.

  - The most common constitutive equations describing this behaviour are the Power-Law model, Casson model and Bird-Carreau model.

  - In this test case, we want to show you the difference between Newtonian and Bird-Carreau models.

  - In Bird-Carreau model, the viscosity ν is related to the shear rate $\dot{\gamma}$ by the equation

$$\nu = \nu_\infty + (\nu_0 - \nu_\infty)[1 + (k \cdot \dot{\gamma})^a]^{(n-1)/a}$$

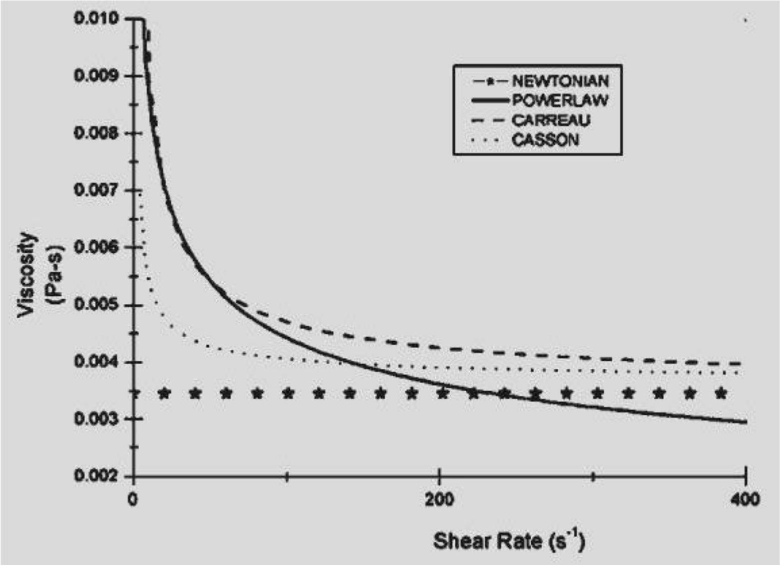

Figure 7: The change of blood viscosity with shear rate for four different viscosity models.

Test case: Fully-developed pipe flow

```
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    object      transportProperties;
}
// * * * * * * * * * * * * * * * * * * * * * * *

transportModel  Newtonian;

nu              3.302e-6;

BirdCarreauCoeffs
{
    nu0         5.283e-5;  //mu0=0.056
    nuInf       3.302e-6;  //muInf=0.0035
    k           3.313;
    n           0.3568;
}
```

transportProperties Newtonian

```
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    object      transportProperties;
}
// * * * * * * * * * * * * * * * * * * * * * * *

transportModel  BirdCarreau;

BirdCarreauCoeffs
{
    nu0         5.283e-5;  //mu0=0.056
    nuInf       3.302e-6;  //muInf=0.0035
    k           3.313;
    n           0.3568;
}
```

transportProperties Bird-Carreau

```
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    object      fvSolution;
}
// * * * * * * * * * * * * * * * * * * * *

solvers
{
    p
    {
        solver          GAMG;
        smoother        GaussSeidel;
        tolerance       1e-06;
        relTol          0;
    }

    U
    {
        solver          smoothSolver;
        smoother        symGaussSeidel;
        tolerance       1e-06;
        relTol          0;
    }

}

SIMPLE
{
    nNonOrthogonalCorrectors 0;
    residualControl
    {
        p               1e-6;
        U               1e-6;
    }
}

relaxationFactors
{
    fields
    {
        p               0.3;
    }
    equations
    {
        U               0.7;
    }
}
```

fvSolution

```
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    location    "system";
    object      fvSchemes;
}
// * * * * * * * * * * * * * * * * * * * * * * *

ddtSchemes
{
    default         steadyState;
}

gradSchemes
{
    default         Gauss linear;
}

divSchemes
{
    default         Gauss linear;
}

laplacianSchemes
{
    default         Gauss linear orthogonal;
}

interpolationSchemes
{
    default         linear;
}

snGradSchemes
{
    default         corrected;
}
```

fvSchemes

```
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    object      decomposeParDict;
}
// * * * * * * * * * * * * * * * *

numberOfSubdomains  4;

method              scotch;
```

decomposeParDict

Test case: Fully-developed pipe flow

```
Build  : _76d719d1-20220624 OPENFOAM=2206 version=2206
Arch   : "LSB;label=32;scalar=64"
Exec   : simpleFoam -parallel
Date   : Nov 30 2023
Time   : 22:33:54
Host   : jcjnb
PID    : 53276
I/O    : uncollated
Case   : /home/jcj/CALISTAY_CENK/5_pipe_newtonian_parallel
nProcs : 4
Hosts  :
(
    (jcjnb 4)
)
Pstream initialized with:
    floatTransfer      : 0
    nProcsSimpleSum    : 0
    commsType          : nonBlocking
    polling iterations : 0
trapFpe: Floating point exception trapping enabled (FOAM_SIGFPE).
fileModificationChecking : Monitoring run-time modified files using timeStampMaster (fileModificationSkew 5, maxFileModificationPolls 20)
allowSystemOperations : Allowing user-supplied system call operations

// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //
Create time

Create mesh for time = 0


SIMPLE: convergence criteria
    field p     tolerance 1e-06
    field U     tolerance 1e-06

Reading field p

Reading field U

Reading/calculating face flux field phi

Selecting incompressible transport model Newtonian
Selecting turbulence model type laminar
Selecting laminar stress model Stokes
No MRF models present

No finite volume options present
```

Log file, initialization
Newtonian

```
Starting time loop

Time = 1

smoothSolver:  Solving for Ux, Initial residual = 0.125447, Final residual = 2.43747e-07, No Iterations 7
smoothSolver:  Solving for Uy, Initial residual = 0.125308, Final residual = 2.43421e-07, No Iterations 7
smoothSolver:  Solving for Uz, Initial residual = 1, Final residual = 2.12733e-07, No Iterations 9
GAMG:  Solving for p, Initial residual = 1, Final residual = 7.37293e-07, No Iterations 66
time step continuity errors : sum local = 1.10347e-08, global = 1.3752e-09, cumulative = 1.3752e-09
ExecutionTime = 0.68 s  ClockTime = 0 s

Time = 2

smoothSolver:  Solving for Ux, Initial residual = 0.556019, Final residual = 4.74506e-07, No Iterations 9
smoothSolver:  Solving for Uy, Initial residual = 0.556019, Final residual = 4.74271e-07, No Iterations 9
smoothSolver:  Solving for Uz, Initial residual = 0.19411, Final residual = 2.80613e-07, No Iterations 8
GAMG:  Solving for p, Initial residual = 0.0191035, Final residual = 6.74874e-07, No Iterations 44
time step continuity errors : sum local = 7.81154e-07, global = 5.92055e-08, cumulative = 6.05807e-08
ExecutionTime = 0.95 s  ClockTime = 1 s
```

Log file, first two iterations
Newtonian

```
Time = 377

smoothSolver:  Solving for Ux, Initial residual = 1.05278e-06, Final residual = 1.07346e-07, No Iterations 1
smoothSolver:  Solving for Uy, Initial residual = 1.05952e-06, Final residual = 1.07987e-07, No Iterations 1
smoothSolver:  Solving for Uz, Initial residual = 7.15186e-08, Final residual = 7.15186e-08, No Iterations 0
GAMG:  Solving for p, Initial residual = 9.45886e-07, Final residual = 9.45886e-07, No Iterations 0
time step continuity errors : sum local = 3.87966e-07, global = -1.02126e-07, cumulative = 2.16188e-06
ExecutionTime = 48.57 s  ClockTime = 48 s

Time = 378

smoothSolver:  Solving for Ux, Initial residual = 9.88133e-07, Final residual = 9.88133e-07, No Iterations 0
smoothSolver:  Solving for Uy, Initial residual = 9.94596e-07, Final residual = 9.94596e-07, No Iterations 0
smoothSolver:  Solving for Uz, Initial residual = 6.93526e-08, Final residual = 6.93526e-08, No Iterations 0
GAMG:  Solving for p, Initial residual = 8.95907e-07, Final residual = 8.95907e-07, No Iterations 0
time step continuity errors : sum local = 3.67467e-07, global = -1.04182e-07, cumulative = 2.0577e-06
ExecutionTime = 48.62 s  ClockTime = 48 s


SIMPLE solution converged in 378 iterations

End

Finalising parallel run
```

Log file, last two iterations
Newtonian

Test case: Fully-developed pipe flow

```
Build  : _76d719d1-20220624 OPENFOAM=2206 version=2206
Arch   : "LSB;label=32;scalar=64"
Exec   : simpleFoam -parallel
Date   : Nov 30 2023
Time   : 22:48:54
Host   : jcjnb
PID    : 53665
I/O    : uncollated
Case   : /home/jcj/CALISTAY_CENK/5_pipe_birdcarreau_parallel
nProcs : 4
Hosts  :
(
    (jcjnb 4)
)
Pstream initialized with:
    floatTransfer      : 0
    nProcsSimpleSum    : 0
    commsType          : nonBlocking
    polling iterations : 0
trapFpe: Floating point exception trapping enabled (FOAM_SIGFPE).
fileModificationChecking : Monitoring run-time modified files using timeStampMaster (fileModificationSkew 5, maxFileModificationPolls 20)
allowSystemOperations : Allowing user-supplied system call operations

// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //
Create time

Create mesh for time = 0


SIMPLE: convergence criteria
    field p     tolerance 1e-06
    field U     tolerance 1e-06

Reading field p

Reading field U

Reading/calculating face flux field phi

Selecting incompressible transport model BirdCarreau
Selecting turbulence model type laminar
Selecting laminar stress model Stokes
No MRF models present

No finite volume options present
```

Log file, initialization
Bird-Carreau

```
Starting time loop

Time = 1

smoothSolver:  Solving for Ux, Initial residual = 0.430471, Final residual = 6.86945e-07, No Iterations 9
smoothSolver:  Solving for Uy, Initial residual = 0.429431, Final residual = 6.81165e-07, No Iterations 9
smoothSolver:  Solving for Uz, Initial residual = 1, Final residual = 4.44393e-07, No Iterations 12
GAMG:  Solving for p, Initial residual = 1, Final residual = 6.70242e-07, No Iterations 64
time step continuity errors : sum local = 3.1172e-09, global = 3.45267e-10, cumulative = 3.45267e-10
ExecutionTime = 0.56 s  ClockTime = 0 s

Time = 2

smoothSolver:  Solving for Ux, Initial residual = 0.71762, Final residual = 6.48025e-07, No Iterations 11
smoothSolver:  Solving for Uy, Initial residual = 0.71762, Final residual = 6.4757e-07, No Iterations 11
smoothSolver:  Solving for Uz, Initial residual = 0.413215, Final residual = 5.48578e-07, No Iterations 10
GAMG:  Solving for p, Initial residual = 0.0438605, Final residual = 8.16686e-07, No Iterations 52
time step continuity errors : sum local = 3.85537e-07, global = 3.45647e-08, cumulative = 3.491e-08
ExecutionTime = 0.89 s  ClockTime = 1 s
```

Log file, first two iterations
Bird-Carreau

```
Time = 859

smoothSolver:  Solving for Ux, Initial residual = 1.41232e-06, Final residual = 2.81987e-07, No Iterations 1
smoothSolver:  Solving for Uy, Initial residual = 1.06885e-06, Final residual = 2.11971e-07, No Iterations 1
smoothSolver:  Solving for Uz, Initial residual = 3.00508e-07, Final residual = 3.00508e-07, No Iterations 0
GAMG:  Solving for p, Initial residual = 1.43679e-06, Final residual = 5.25676e-07, No Iterations 1
time step continuity errors : sum local = 1.81752e-07, global = 2.85669e-08, cumulative = 1.03825e-05
ExecutionTime = 105.98 s  ClockTime = 106 s

Time = 860

smoothSolver:  Solving for Ux, Initial residual = 8.73389e-07, Final residual = 8.73389e-07, No Iterations 0
smoothSolver:  Solving for Uy, Initial residual = 9.529e-07, Final residual = 9.529e-07, No Iterations 0
smoothSolver:  Solving for Uz, Initial residual = 2.98144e-07, Final residual = 2.98144e-07, No Iterations 0
GAMG:  Solving for p, Initial residual = 8.1104e-07, Final residual = 8.1104e-07, No Iterations 0
time step continuity errors : sum local = 2.80417e-07, global = 2.02678e-08, cumulative = 1.04028e-05
ExecutionTime = 106.04 s  ClockTime = 106 s


SIMPLE solution converged in 860 iterations

End

Finalising parallel run
```

Log file, last two iterations
Bird-Carreau

Thank you ☺

Emre Cenk ERSAN
PhD Candidate – Research Assistant
Istanbul Technical University
Computational Science and Engineering
ersane@itu.edu.tr