

OpenFoam
Porous medium

Ehsan Mahravan

July 25, 2018



Please send me your feedback, comments, and suggestions

mahravan.ehsan@gmail.com



Copy the following tutorial to your working folder:

`$FOAM_TUTORIALS/incompressible/porousSimpleFoam/angledDuctImplicit`

Simple Questions

- 1 What are steps to run this tutorial?
- 2 where is porous media in geometry?
- 3 where porous inputs are given?



```
1 |#!/bin/sh
2 |cd ${0%/*} || exit 1    # Run from this directory
3 |
4 |m4 system/blockMeshDict.m4 > system/blockMeshDict
5 |
6 |# Source tutorial run functions
7 |. $WM_PROJECT_DIR/bin/tools/RunFunctions
8 |
9 |runApplication blockMesh
10|runApplication 'getApplication'
```

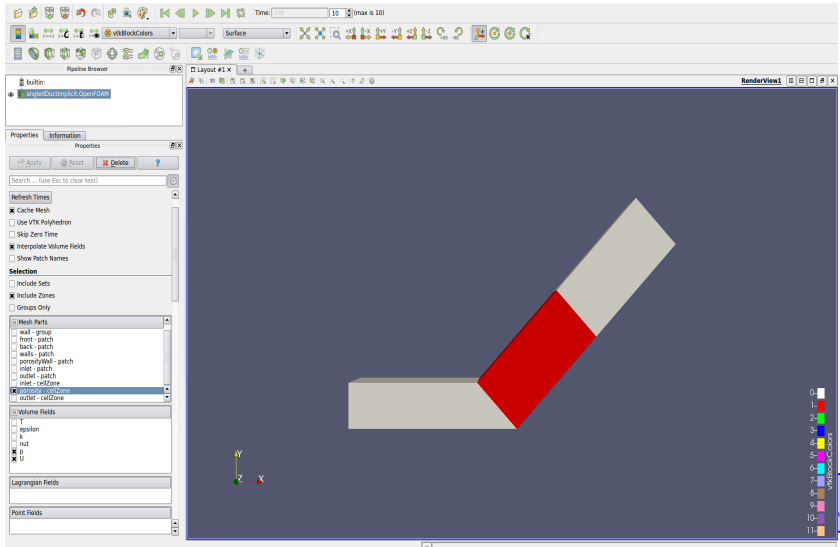
- 1 line 4: Creating `blockMeshDict` file using `m4`,
- 2 line 9: Creating mesh using `blockMesh`,
- 3 line 10: Solving the case using `porousSimpleFoam`.



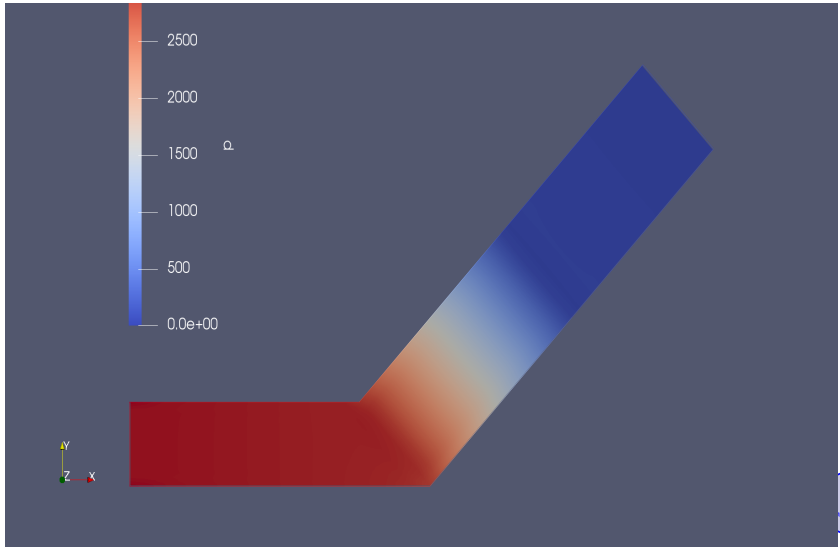
- 1 Execute Allrun script by: `./Allrun`
- 2 Open paraview using `paraFoam`
- 3 In properties panel toggle `include zone`,
- 4 In mesh parts select `porosity - cellZone`,
- 5 In display panel select `vtkBlockColors`,



- The red block in the picture is porous region.



- Porous medium causes a considerable pressure drop.



- `topoSet` is not used, so how the `porous zone` is specified?
- This zone must be specified in `blockMeshDict`. Where blocks are defined, one can also give them a name and after generating the mesh it will be a separate zone.

```
1 blocks
2 (
3     // inlet block
4     hex (0 4 5 1 2 6 7 3)
5     inlet ( 15 20 20 ) simpleGrading (1 1 1)
6
7     // porosity block
8     hex (4 8 9 5 6 10 11 7)
9     porosity ( 20 20 20 ) simpleGrading (1 1 1)
10
11    // outlet block
12    hex (8 12 13 9 10 14 15 11)
13    outlet ( 20 20 20 ) simpleGrading (1 1 1)
14 );
```



- Porosity properties are given in `constant / porosityProperties`
- This file can be provided only for solvers which has *poros* word in their name.
- Other solvers can also handle porosity. For these solvers one can provide porosity information in *fvOptions*.

```
1 | porous1
2 | {
3 |     type            DarcyForchheimer;
4 |
5 |     cellZone        porosity;
6 |
7 |     d   (5e7 -1000 -1000);
8 |     f   (0 0 0);
9 |
10 |    coordinateSystem
11 |    {
12 |        type        cartesian;
13 |        origin      (0 0 0);
14 |        coordinateRotation
15 |        {
16 |            type        axesRotation;
17 |            e1          (0.70710678 0.70710678 0);
18 |            e2          (0 0 1);
19 |        }
20 |    }
21 | }
```



- **type**: Name of the model to encounter porous effect.
- **cellZone**: is the name of the zone for which porosity model will be applied.
- **active**: One can choose whether porosity model be applied or not by changing *active* entry to yes or no.
- **modelNameCoeff**: Requirements of selected model.



For encountering porosity effect a source term (S) must be added to the momentum equation. This can be done using three well known porosity model as follows

- 1 **DarcyForchheimer**: Viscous and inertial resistances vectors (d and f) must be specified for diagonal of the source term added to Navier-Stocks equations. Coordinate of vectors correspond to local coordinate system.

$$S = -(\mu d + \frac{\rho|U|}{2}f)U$$

- 2 **fixedCoeff**: Two vector constants (**alpha** and **beta**) are needed and for compressible flow reference density is also required.

$$S = -\rho_{ref}(\alpha + \beta|U|)U$$

- 3 **powerLaw**: Two constants (**C_0** and **C_1**) are needed, the first in Coefficient and the second is power used in the source term.

$$S = -\rho C_0|U|^{(C_1-1)}U$$



A local coordinate system which represent direction of porosity must be specified. The only available type is *cartesian* which needs an origin and orientation.

Orientation here is along angled part of the duct. *coordinateRotation* type are as follows:

- 1 axesRotation: The 3 rotations are defined in the axis convention around X, around Y and around Z).
- 2 STARCDRotation: The 3 rotations are defined in the STAR-CD convention (around Z, around X' and around Y'').
- 3 EulerCoordinateRotation: The 3 rotations are defined in the STAR-CD convention (around Z, around X' and around Y'') for STARCDRotation
- 4 cylindrical



```
1 | coordinateRotation
2 | {
3 |     type STARCDRotation; // EulerCoordinateRotation
4 |     degrees false; // true
5 |     rotation (0 0 3.141592654); // (0 0 180)
6 | }
7 | coordinateRotation
8 | {
9 |     type EulerRotation
10 |    degrees false;
11 |    rotation (0 0 3.141592654);
12 | }
13 | cylindrical
14 | {
15 |     type localAxes;
16 |     e3 (0 0 1);
17 | }
```



```
1 | porosity1
2 | {
3 |     type          fixedCoeff;
4 |     active        yes;
5 |     cellZone      porosity;
6 |
7 |     fixedCoeffCoeffs
8 |     {
9 |         alpha      (500 -1000 -1000);
10 |        beta       (0 0 0);
11 |
12 |        coordinateSystem
13 |        {
14 |            type      cartesian;
15 |            origin    (0 0 0);
16 |            coordinateRotation
17 |            {
18 |                type    axesRotation;
19 |                e1      (0.70710678 0.70710678 0);
20 |                e2      (0 0 1);
21 |            }
22 |        }
23 |    }
24 | }
```



\$FOAM_TUTORIALS/incompressible/pisoFoam/laminar/porousBlockage/

```
1 porosity1
2 {
3     type                explicitPorositySource;
4
5     explicitPorositySourceCoeffs
6     {
7         selectionMode    cellZone;
8         cellZone         porousBlockage;
9
10        type              DarcyForchheimer;
11
12        // D 100; // Very little blockage
13        // D 200; // Some blockage but steady flow
14        // D 500; // Slight waviness in the far wake
15        D 1000; // Fully shedding behavior
16
17        d ($D $D $D);
18        f (0 0 0);
19
20        coordinateSystem
21        {
22            type          cartesian;
23            origin        (0 0 0);
24            coordinateRotation
25            {
26                type      axesRotation;
27                e1        (1 0 0);
28                e2        (0 1 0);
29            }
30        }
31    }
32 }
```



- OpenFOAM header files.
- Description of porousSimpleFoam and adding the Brinkmann model to the porous models, Reza Gooya, A course at Chalmers University of Technology.

