Introduction

- The Governing Equations
- The Class
- The solver
- Porous media as a flow control
- Porous media in cylindrical coordinates
- Available at the course home page:

  `http://www.tfd.chalmers.se/~hani/kurser/OS_CFD_2008/`

The Governing Equations

Porous media is modeled by adding a source term S_i to the momentum equations.
A porosity value $0 < \gamma < 1$ is added to the time derivative:

$$\frac{\partial}{\partial t}(\gamma \rho u_i) + u_j \frac{\partial}{\partial x_j}(\rho u_i) = -\frac{\partial p}{\partial x_i} + \mu \frac{\partial \tau_{ij}}{\partial x_j} + S_i \tag{1}$$

The source term is represented in the `Darcy-Forchheimer` law as a viscous loss term and an inertial loss term:

$$S_i = -\left(\mu\, D_{ij} + \frac{1}{2}\rho\, |u_{kk}| F_{ij}\right) u_i \tag{2}$$

The source term can also be modeled as a `power-law` of the velocity magnitude:

$$S_i = -\rho\, C_0\, |u_i|^{(C_1-1)/2} \tag{3}$$

The Class

The porous media source files in `OpenFOAM-1.5` are located in the following directory:
`$FOAM_SRC/finiteVolume/cfdTools/general/porousMedia/`
The `porousMedia` folder contains the following files:

- `porousZones.H`
- `porousZones.C`
- `porousZone.H`
- `porousZone.C`
- `porousZonesTemplate.C`
- `porousZoneTemplate.C`

The porosity equations are implemented in the file `porousZoneTemplate.C`

The Solver

The solver name is `rhoPorousSimpleFoam`. It is a steady-state solver for turbulent flow of compressible fluids with implicit or explicit porosity treatment. It is located in the following directory:
`\$FOAM\_SOLVERS/compressible/rhoPorousSimpleFoam/`
The folder `rhoPorousSimpleFoam` contains a `Make` folder and the following files:

- `rhoPorousSimpleFoam.C`
- `initConvergenceCheck.H`
- `convergenceCheck.H`
- `createFields.H`
- `UEqn.H`
- `pEqn.H`
- `hEqn.H`

The `rhoPorousSimpleFoam` solver is based on the `rhoSimpleFoam` solver.

The Case

Two tutorials using `rhoPorousSimpleFoam` are included in `OpenFoam-1.5`. They are located at:
`$FOAM_TUTORIALS/rhoPorousSimpleFoam`

Here another prepared case having the same directory structure will be used. The geometry is the `ercoftacConicalDiffuser - Case1`

Copy the .tar file from the course homepage, extract it and remove the .tar file.

Copy the case `ergoCart` to your `run` directory:
`cp -r ergoCart $FOAM_RUN/`

Open `$FOAM_RUN/constant/porousZones`.

```
<case>
├─ system
│   ├─ controlDict
│   ├─ fvSchemes
│   └─ fvSolution
├─ constant
│   ├─ RASProperties
│   ├─ thermophysicalProperties
│   ├─ porousZones
│   └─ polyMesh
│       └─ blockMeshDict
└─ 0
    ├─ T
    ├─ U
    ├─ epsilon
    ├─ k
    └─ p
```

constant/porousZones

A porous media is added to a `cellZone` named `CD` created in the `blockMeshDict`. A local coordinate system belonging to the porous media is created from the vectors `e1` and `e2`. The pressure loss in the local coordinate system is then defined with the vectors `d` and `f`.

```
1
(
    CD
    {
        coordinateSystem
        {
            e1  (0 0.70710678 0.70710678);
            e2  (1 0 0);
        }
        Darcy
        {
            d   d [0 -2 0 0 0 0 0] (5e7 1000 1000);;
           // f   f [0 -1 0 0 0 0 0] (0 0 0);
        }
    }
)
```

Running the case

Create the `blockMeshDict` by entering the `polyMesh` folder and type:

```
m4 -P blockMeshDict.m4 > blockMeshDict
```

Run the case by typing:

```
blockMeshDict
rhoPorousSimpleFoam > log &
```

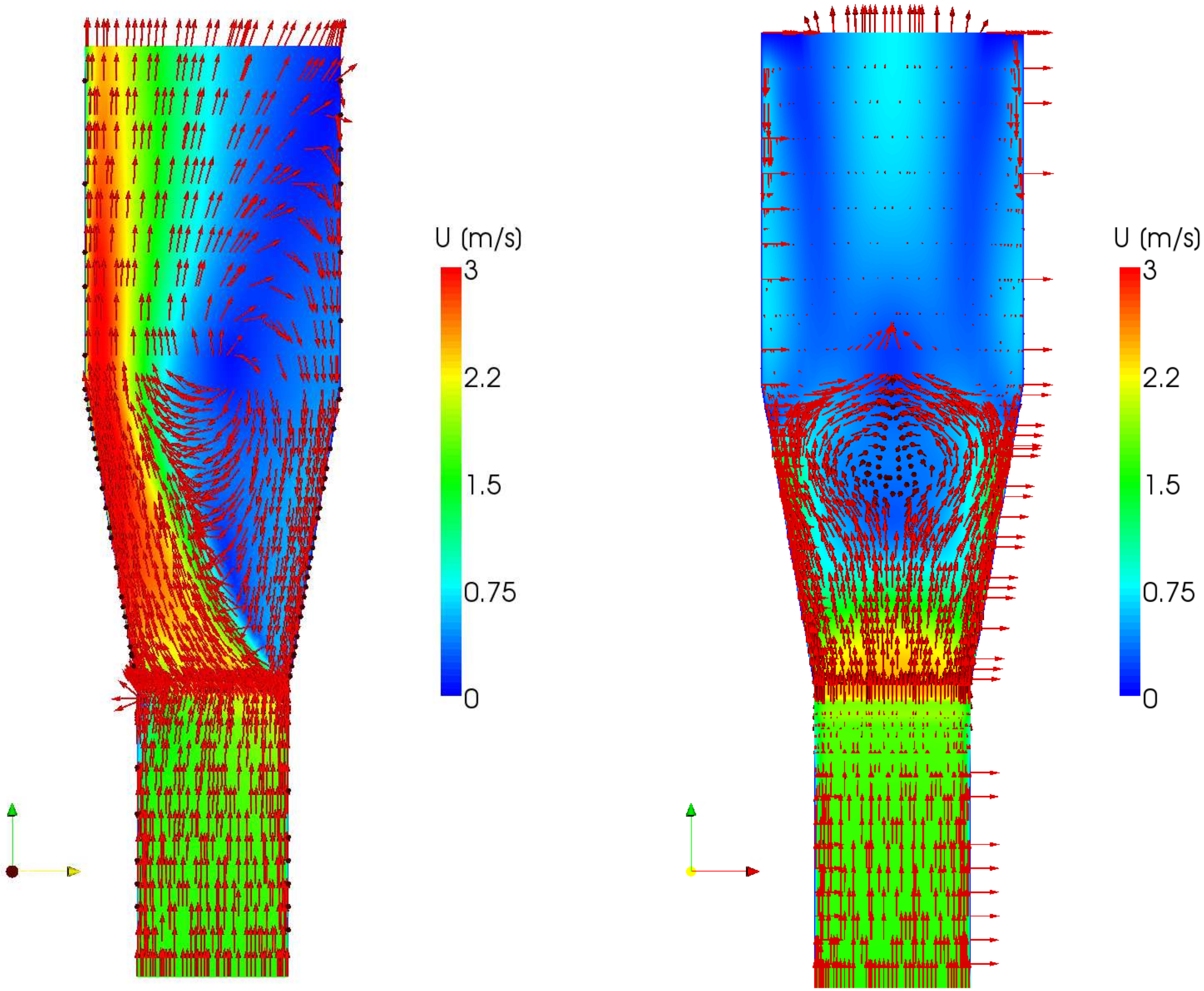
This will take some time...

When finished view the results in `paraView` by typing:

```
paraFoam &
```

Create `x-cut` and `y-cut` to view the velocity field.

The flow turns $45°$ from z-axis because of the porous media



Porous media in cylindrical coordinates

Create the library:

```
mkdir -p $WM_PROJECT_USER_DIR/src/finiteVolume/cfdTools/general/
cp -r cylPorousMedia \
$WM_PROJECT_USER_DIR/src/finiteVolume/cfdTools/general/
cd $WM_PROJECT_USER_DIR/src/finiteVolume/cfdTools/general/cylPorousMedia
wmake libso
```

Compile the solver:

```
mkdir -p $WM_PROJECT_USER_DIR/applications/solvers/compressible/
cp -r rhoCylPorousSimpleFoam \
$WM_PROJECT_USER_DIR/applications/solvers/compressible/
cd $WM_PROJECT_USER_DIR/applications/solvers/ \
compressible/rhoCylPorousSimpleFoam
wmake
```

Copy the case to your `run` **directory:**

```
cp -r ergoCyl $FOAM_RUN/
```

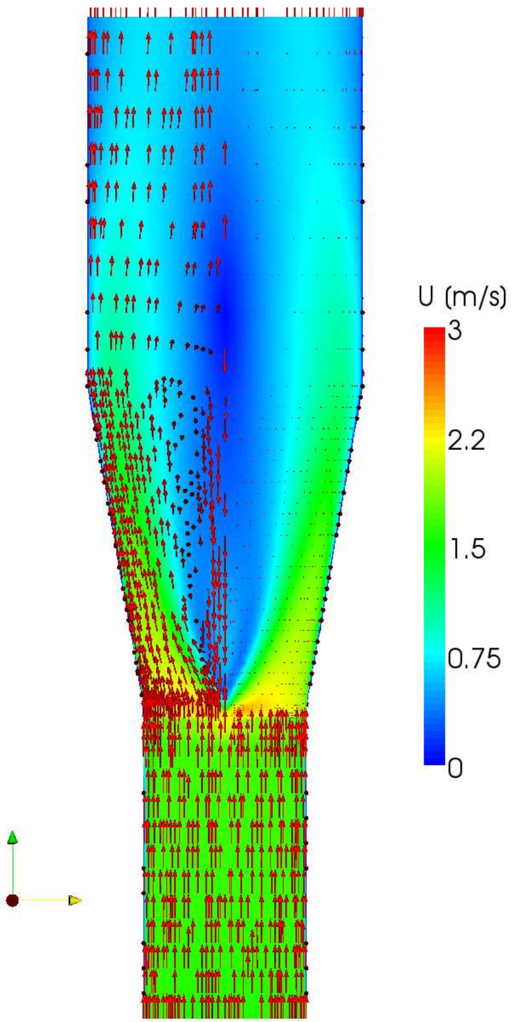
Open `$FOAM_RUN/ergoCyl/constant/cylPorousZones`

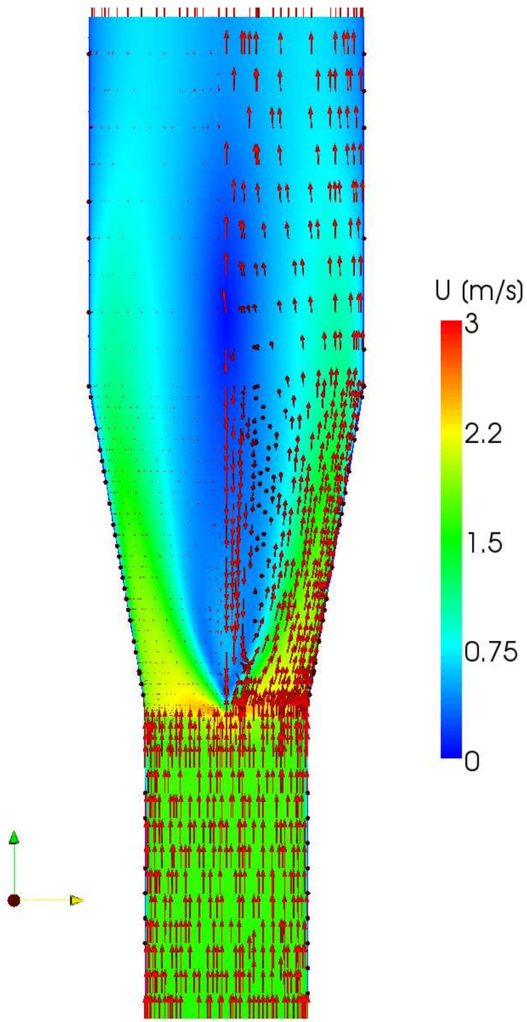
constant/cylPorousZones

The variables radialAngle and rotationAngle control the turning of the local coordinate-system. It is recommended that if the radialAngle is to be set to a nonzero value then the rotationAngle should be set to zero and vice versa.

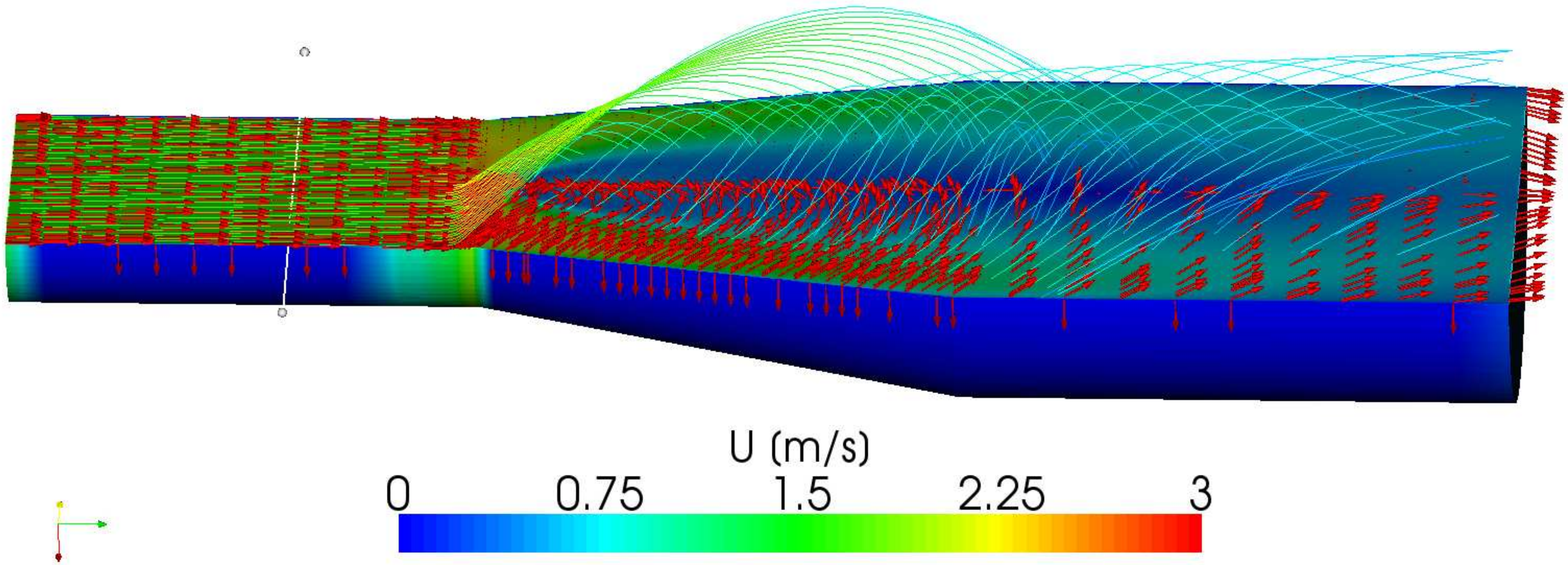
```
1
(
   CD
    {
         radialAngle 0;
         rotationAngle 45;
         Darcy
         {
             d   d [0 -2 0 0 0 0 0] (1000 5e7 1000);
            // f   f [0 -1 0 0 0 0 0] (0 0 0);
         }
    }
)
```

rotationAngle = 45, 135, radialAngle = 0

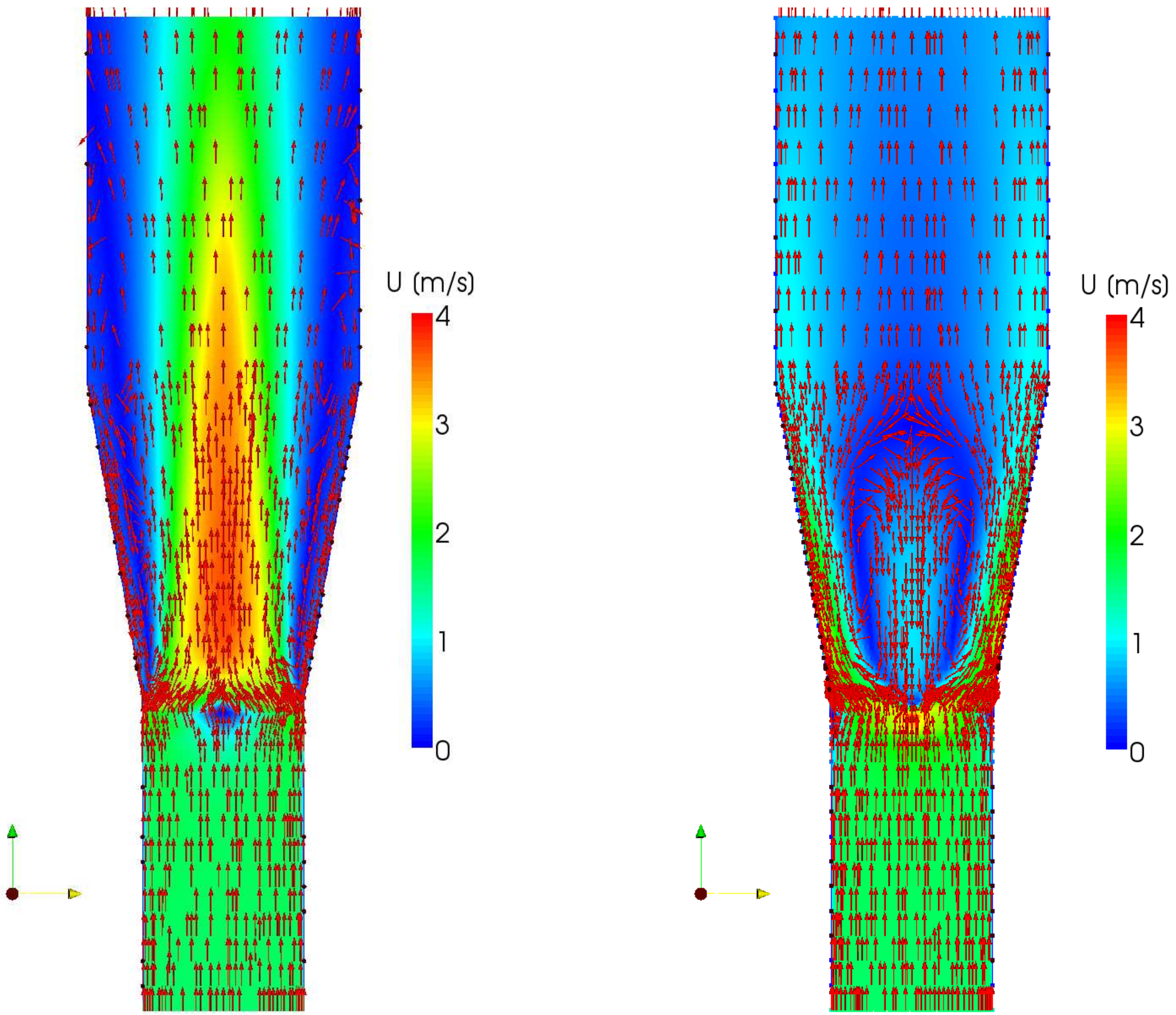




Streamlines visualizing the rotation of the flow.



`radialAngle = 45, 135, rotationAngle = 0`



Thank you