

Tutoriel 2: Ecoulement de Poiseuille

Asmaa HADANE

ENS Paris-Saclay

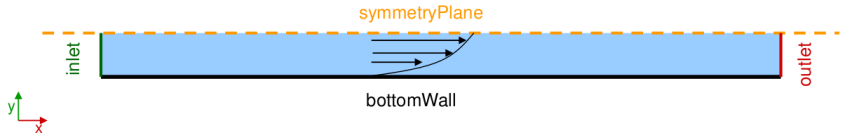
February 22, 2025

Objectifs :

- 🔗 Simuler un écoulement de Poiseuille dans un tube (2D) , avec condition de symétrie
- 🔗 Résolution de Navier-Stokes incompressible en régime laminaire (solver *icoFoam*)

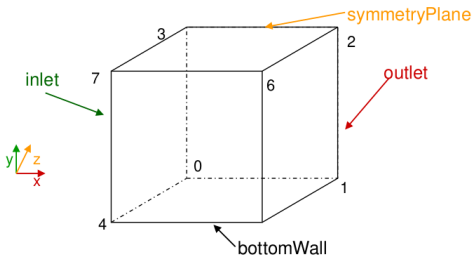
$$\nabla \cdot \mathbf{U} = 0$$

$$\frac{\partial \mathbf{U}}{\partial t} + \nabla \cdot (\mathbf{U}\mathbf{U}) = \nabla \cdot (\nu \nabla \mathbf{U}) - \nabla p$$



```
$ run ¶  
$ cp -r $FOAM_TUTORIALS/incompressible/icoFoam/cavity Exo3 ¶  
$ cd Exo3 ¶
```

\$ gedit constant/polyMesh/blockMeshDict ¶



convertToMeters 0.1;

vertices

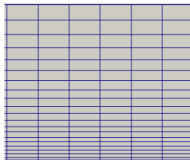
```
(
  (0 0 0)
  (20 0 0)
  (20 1 0)
  (0 1 0)
  (0 0 0.1)
  (20 0 0.1)
  (20 1 0.1)
  (0 1 0.1)
);
```

blocks

```
(
  hex (0 1 2 3 4 5 6 7) (100 20 1) simpleGrading (1 5 1)
);
```

edges

```
(
```



```
boundary
(
  symmetryPlane
  {
    type symmetryPlane;
    faces
    (
      (3 7 6 2)
    );
  }
  inlet
  {
    type patch;
    faces
    (
      (0 4 7 3)
    );
  }
  outlet
  {
    type patch;
    faces
    (
      (2 6 5 1)
    );
  }
  bottomWall
  {
    type wall;
    faces
    (
      (1 5 4 0)
    );
  }
  frontAndBack
  {
    type empty;
    faces
    (
      (0 3 2 1)
      (4 5 6 7)
    );
  }
);
```

\$ blockMesh ¶

\$ gedit 0/U¶



```
blockMeshDict p U
=====
Field Operation And Manipulation
OpenFOAM: The Open Source CFD Toolbox
Version: 2.0.1
Web: www.OpenFOAM.com
=====
FoamFile
{
    version      2.0;
    format       ascii;
    class        volVectorField;
    object       U;
}
// ..... //

dimensions      [0 1 -1 0 0 0];

internalField   uniform (0 0 0);

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

    outlet
    {
        type      zeroGradient;
    }

    bottomWall
    {
        type      fixedValue;
        value      uniform (0 0 0);
    }

    symmetryPlane
    {
        type      symmetryPlane;
    }

    frontAndBack
    {
        type      empty;
    }
}
```

\$ gedit 0/p ¶

```
blockMeshDict  p  x
-----*-- C++ -*-----
V      F field      OpenFOAM: The Open Source CFD Toolbox
      O peration    Version: 2.0.1
      A nd          Web: www.OpenFOAM.com
      M anipulation

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

dimensions      [0 2 -2 0 0 0];

internalField   uniform 0;

boundaryField
{
  inlet
  {
    type        zeroGradient;
  }

  outlet
  {
    type        fixedValue;
    value       uniform 0;
  }

  bottomWall
  {
    type        zeroGradient;
  }

  symmetryPlane
  {
    type        symmetryPlane;
  }

  frontAndBack
  {
    type        empty;
  }
}
```

🔧 Lancement du calcul : `$ icoFoam ¶`

🔧 Visualisation du résultat : `$ paraFoam ¶`

