FLUENT CFD Project

CFD prediction of wind flow through tree canopy

1 Presentation of the project

The pdf version of this document and all necessary files for completing the project are available online at the course webpage:

UE 203 - Computational Fluid Dynamics M1 MEPP

at the ARCHE website (`http://arche.univ-lorraine.fr/enrol/index.php?id=9396`). Access to all users is permitted with password: CFD.

1.1 Wind flow through tree canopy

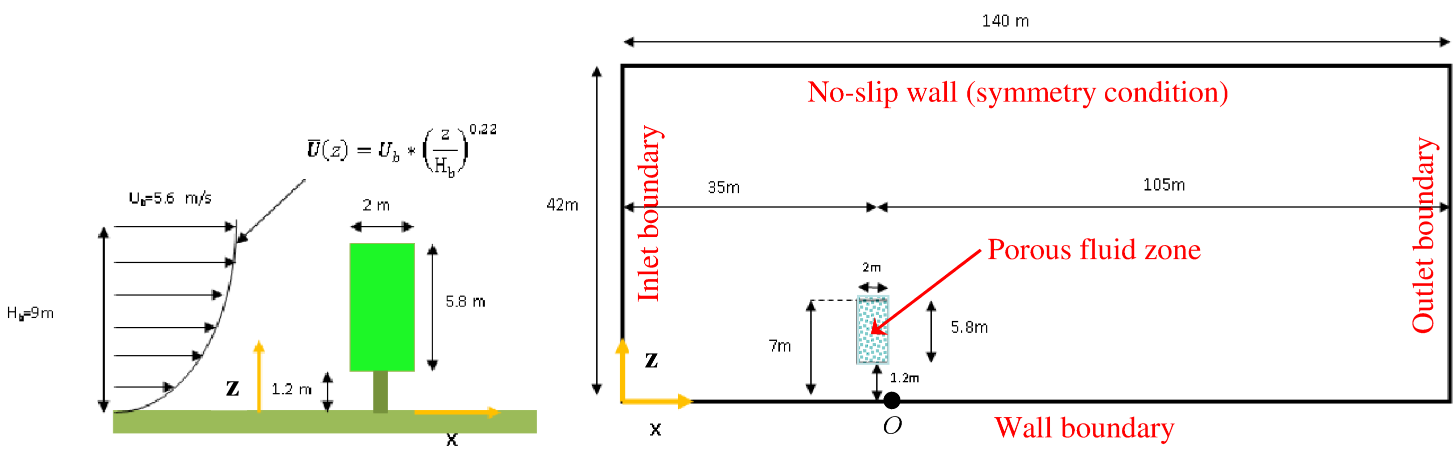

Figure 1: Left: Schematic view of the test case. Right: Domain definition and boundary conditions (Note that the tree trunk is not modelled).

This CFD project concerns the modelization of the aerodynamic effects of tree canopy in the framework of the numerical prediction of wind environment at pedestrian level [6]. This test case considers the air flow through a bush of pine trees as described in Fig. 1 (left), and is based on the experimental investigation of Kurotani *et al.* [3] and the numerical simulations of Mochida *et al.* [5] and Gamel & Muller [1]. The Reynolds number of the flow is :

$$\mathrm{Re} = \frac{\rho U_{\mathrm{b}} H}{\mu} \simeq 3.7 \times 10^6, \tag{1}$$

where reference quantities are $U_{\mathrm{b}} = 5.6\,m$, the wind speed at height $z_{\mathrm{b}} = 9\,m$, and $H = 5.8\,m$ is the height of the tree canopy. As a result, the flow can be considered as a 2D steady turbulent flow. The dimensions and boundary conditions of the computational domain are given in Fig. 1 (right). The tree canopy will be modelled as a porous medium (see Appendix 3.2, where source terms have to be defined in the flow equations in order to model the aerodynamic effects of the canopy.

1.2 Numerical experiments to perform

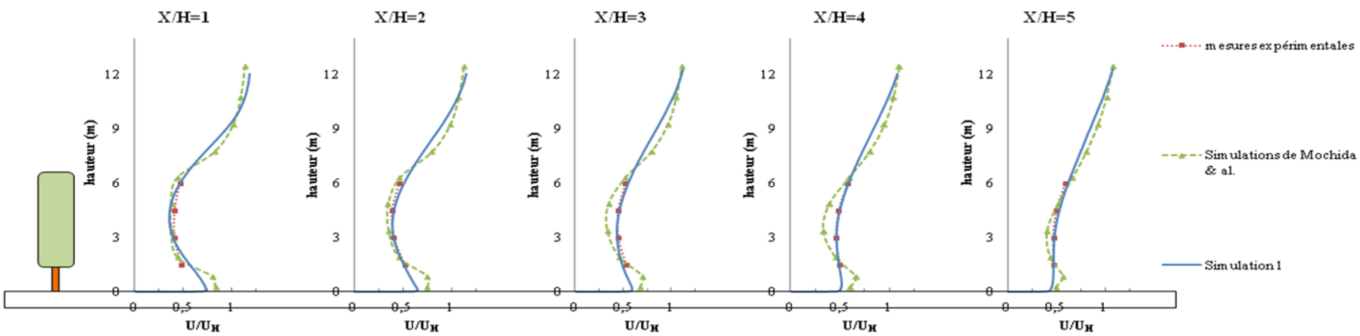

Figure 2: Numerical results obtained by Gamel & Muller [1] for the mean velocity.

The purpose of this project is to computer with ANSYS Fluent the flow at Re $\simeq 3.7 \times 10^6$ and compare your results with the experimental data of Kurotani *et al.* [3] and the numerical results of Mochida *et al.* [5], *i.e.* the profile of the mean velocity $\overline{u}$ and kinetic energy k on various cross sections in the wake of the pine tree (see Fig. 2). The data are given in the excel file `CFD_Project_Flow_thru_tree_ref_data.xls` available on the course webpage.

This document will help you build the mesh and set up the Fluent modelization. It is advised that your final report should include the following elements :

- Description of the mesh(es) you have built.
- Description of the turbulence model, discretization methods, etc . . . that you have used.
- Plot of residuals and convergence check: report velocity profile in the wake and mass flow balance for threshold $\epsilon = 10^{-3}$, $\epsilon = 10^{-4}$ and $\epsilon = 10^{-5}$ (see Sect. 2.2.1 for more information).
- Flow visualization.
- Comparison of results in the wake of the pine tree with reference data.
- Convergence study by comparing results obtained on refined meshes, with various turbulence models and numerical discretization.

2 Tips for practical implementation in ANSYS FLUENT

The version of the software used for writing this document is ANSYS 15.

2.1 Building of the domain and meshing

This section will give you guidelines for building a mesh with tri/quad elements such as the one displayed in Fig. 3, which should be highly refined in the vicinity of the canopy, the wake past the canopy, and in the turbulent boundary layer along the ground. For generating the mesh with Ansys Meshing , the computational domain should be decomposed into 4 disjoint faces as shown in Fig. 4. Then, before importing into Fluent the mesh should be separated into 2 disjoint cell fluid zones:

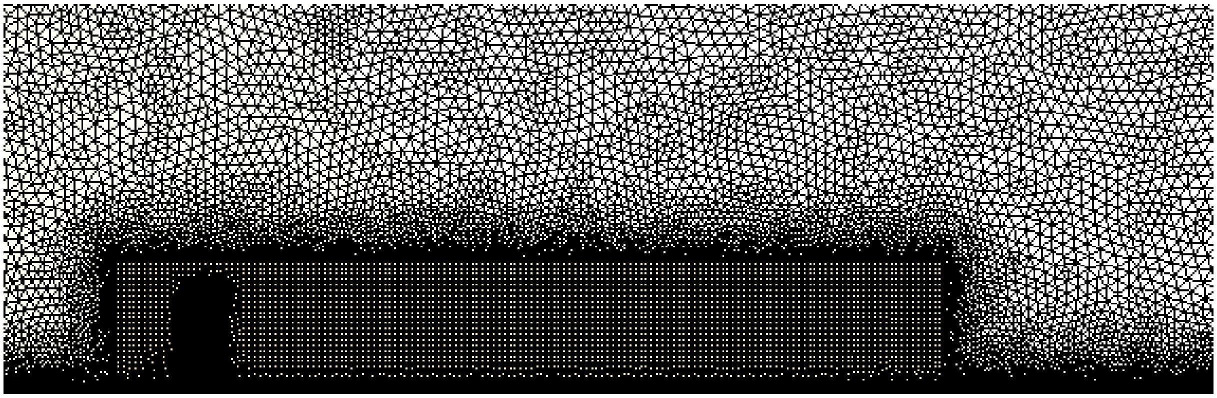

Figure 3: Close-up of the mesh of the computational domain. This mesh has about 120,000 mixed cells (tri and quad).

- Faces 1,2 and 3 is the *air fluid zone*. Name the zone: `Exterior_Fluid_Zone`.
- Face 4 represents the *tree canopy*, and should be named `Tree_Canopy` in order to set porous media conditions in Fluent (see Sect. 2.2.3).

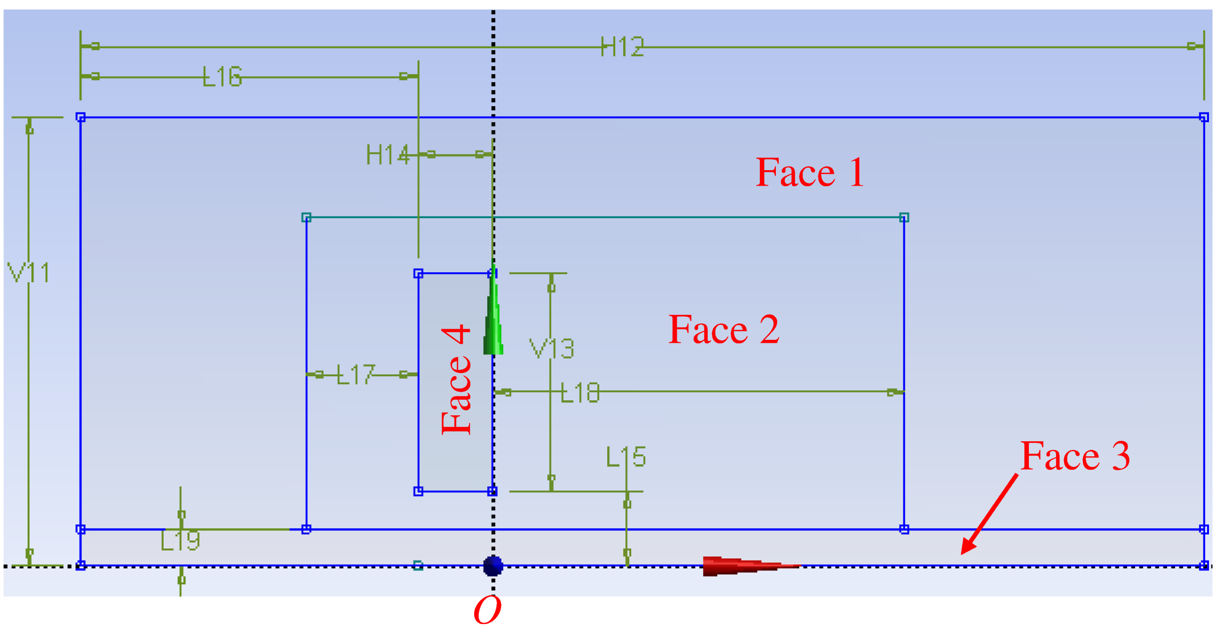

Figure 4: Decomposition of the domain into 4 faces in Design Modeler (note that figure is not scaled). The dimensions are H12 $= 150\,m$, V11 $= 35\,m$, L19 $= 0.6\,m$, L16 $= 35\,m$, L17 $= 5\,m$, H14 $= 2\,m$, L18 $= 50\,m$, L15 $= 1.2\,m$, V13 $= 5.8\,m$.

2.1.1 Creating the geometry with Design Modeler

Here are some tips for creating the geometry:

- Sketch the computational domain on the XYPlane with the dimensions given in Fig. 4 (Note that there is no unique way to build the sketch). It is advised to create each face on a different sketch. The origin O of the domain should be positioned as in the figure, in order to compare with the experimental and numerical reference data.

- Create a surface sketch from these 4 sketches.
- Create surface bodies and label them as in Fig. 4.
- Use boolean subtraction to make each surface body disjoints. Do not suppress any tool body.
- Create one part from the 4 surfaces bodies. Don't forget to declare each surface body as Fluid.
- Export the geometry into Ansys Meshing .

2.1.2 Creating the mesh with Ansys Meshing

Now, each of the 4 faces should use different mesh settings which are described below. To avoid conflicts when the 4 face meshes are connected together, only the boundary edges of the computational domain and the tree canopy should be meshed, but not the (interior) edges of face 2.

- The maximal face size of the elements of the mesh is $0.4m$
- Meshing Method:
  - Use Triangles Only Method for Face 1.
  - Use Automatic Method for other faces.
- Edge Sizing:
  - Face 1 has the coarsest elements with boundary edge size of $0.4\,m$.
  - Face 4 has the finest mesh elements, with boundary edge size of $0.06\,m$.
  - The edge representing the ground should have elements of size $0.1\,m$.
- Face Sizing:
  - Face 2 and 3 have elements of size $0.2\,m$.
  - To build a high quality structured mesh for the ground boundary layer (Face 3), use Inflation method with first cell size of $0.01\,m$ and about 20 layers.
- Named Selection:
  - Name the boundary edges and the 2 fluid faces with meaningful names (see beginning of the section) before importing the mesh in Fluent .

The mesh you obtain should have about $120,000$ cells. Note that this mesh is based on the coarsest mesh used by Gamel & Muller [1], so you may need to refine the mesh if the numerical results you obtain with Fluent are unsatisfactory.

2.2 FLUENT modelization

2.2.1 Solver settings

- The flow is 2D, steady, but highly turbulent so use a turbulence model such as the $k-\epsilon$ model with standard wall functions.
- To avoid stability issues during computations, use the best available discretization in your version of Fluent : third-order MUSCL for convection, and PRESTO! for pressure interpolation.
- For better convergence, start computations with no porous model in the canopy, then turn on the porous model.
- Conventional residual threshold $\epsilon = 10^{-3}$ is insufficient to get a converged solution for highly turbulent flow. This threshold has to be lowered. In addition, to check that the solution has sufficiently converged, verify during the SIMPLE iterations that :
  - Mass flow balance between inflow and outflow boundaries goes to zero :
    Reports → Fluxes → Set Up ... → Mass Flow Rate
  - Velocity profile in the wake has converged.

2.2.2 User Defined Fonction (UDF) for inflow condition

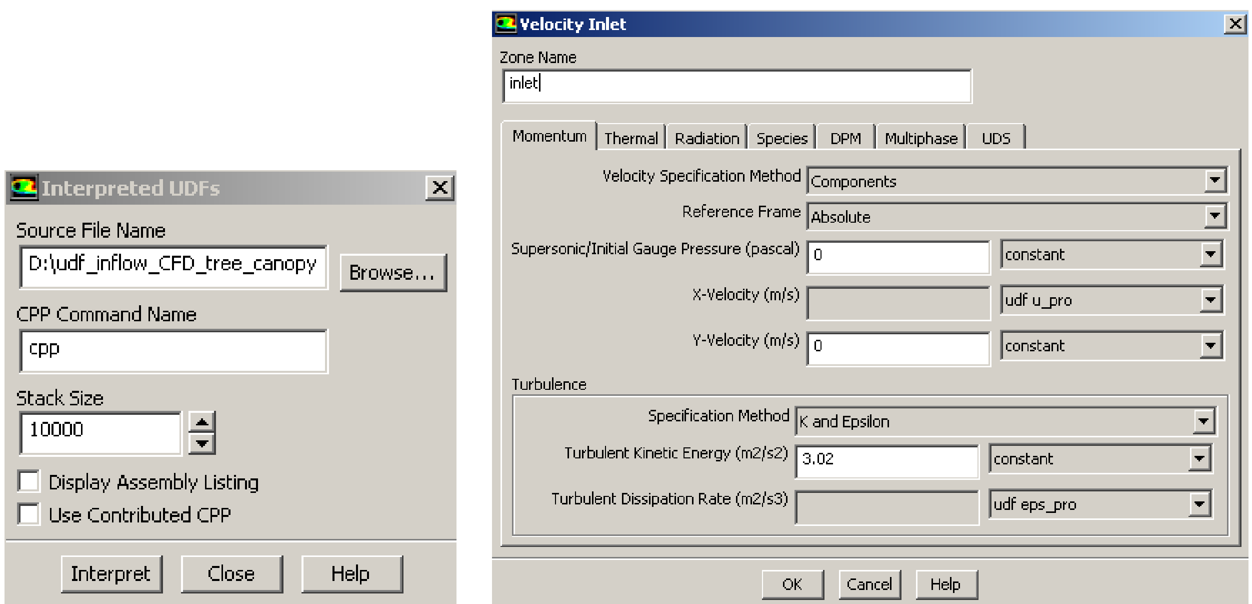

Figure 5: Using *UDFs* for implementing non-uniform boundary conditions.

For flow computations with a turbulent model such as $k-\epsilon$, inflow conditions are required for the velocity and turbulent quantities such as the kinetic energy k and the turbulent dissipation rate ϵ. These inflow conditions should be determined from the available experimental measurements. According to [6], the velocity at inlet is modelled as the power-law profile :

$$\overline{u}(z) = U_{\mathrm{b}} \left(\frac{z}{z_{\mathrm{b}}} \right)^{\alpha}, \quad \alpha = 0.22, \tag{2}$$

the kinetic energy is uniform such as $k = 3.02 \mathrm{m}^2 \cdot \mathrm{s}^{-2}$, and the turbulent dissipation is modelled by making the classical assumption that turbulent production is equivalent to its dissipation [4], which yields :

$$\epsilon(z) = \sqrt{C_\mu} k \frac{\partial \overline{u}}{\partial z}, \tag{3}$$

with $C_\mu = 0.09$. In the Fluent software, non-uniform inflow conditions are provided by the means of the *User Defined Function* (UDF) `DEFINE_PROFILE` (see Appendix 3.1). The C code of the UDFs for $\overline{u}(z)$ and $\epsilon(z)$ are written in the file `udf_inflow_CFD_tree_canopy.c` (available at the course webpage). The only modification you have to do is the definition of the mean velocity U_b (in $\mathrm{m} \cdot \mathrm{s}^{-1}$). For setting these inflow conditions in Fluent , open the Interpreted UDFs panel (Fig. 5, left) as

Define → User-Defined → Functions → Interpreted. . . ,

load the UDF file and click on Interpret. If Fluent does not give error messages then the UDFs `u_pro` and `k_pro` will be ready to be implemented in the Velocity Inlet panel as shown in Fig. 5 (right). Note that each time you edit the UDF program, you need to do interpret the UDFs to make the changes effective in Fluent .

2.2.3 Porous media modelization for air flow through plant canopy

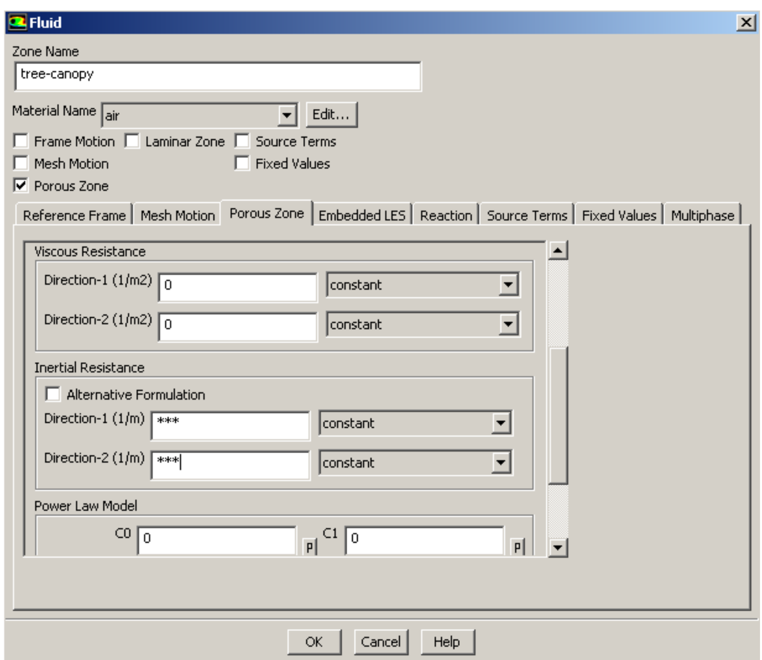

Figure 6: Cell Zone Conditions panel for porous medium.

Air flow through tree canopy is modelled by the addition of a sink term (similar to Forchheimer's model for porous media, see Appendix 3.2) to the i^{th} momentum equation [5] :

$$S_i = \eta C_\mathrm{D} \, \overline{\mathrm{LAD}} \, \overline{u}_i \, |u|, \tag{4}$$

where:

- $\eta = 1$ is the fraction of the area covered with canopy,

- $C_{\rm D}$ is the drag coefficient (determined experimentally),
- $\overline{\rm LAD}$ is the mean leaf-area density of the canopy (in m^{-1}).

The LAD (and also LAI, *leaf-area index*) is a quantity that characterizes the density of plant canopies. LAD and LAI are determined experimentally for each type of plants. According to Mochida (private communication), the value for pine trees is $\overline{\rm LAD} = 1.17\,m^{-1}$ and the drag coefficient is $C_{\rm D} = 0.8$.

For modelling flow through porous media, Fluent adds a momentum source term which is composed of a viscous resistance term (Darcy model) and an inertial resistance term (Forchheimer model) as:

$$S_i = \sum_{j=1}^{3} \mu \mathcal{D}_{i,j} v_j + \sum_{j=1}^{3} \tfrac{1}{2}\rho\, \mathcal{C}_{i,j} |u| \overline{u}_j, \tag{5}$$

where $\mathcal{D}$ and $\mathcal{C}$ are prescribed matrices. In this project, you will set $\mathcal{D} = 0$ and calculate the diagonal components of the inertial resistance matrix $\mathcal{C}$ by identification with the tree canopy model (4). For setting the porous media model in Fluent , edit the Cell Zone Conditions panel (Fig. 6) for the tree-canopy fluid zone :

Define → Cell Zone Conditions...

and enter the value of the inertial resistance terms.

3 Appendix

3.1 Basics of UDFs in Fluent

A User Defined Function or UDF is a function (programmed by the user) written in C which can be dynamically liked with the Fluent solver. An UDF uses standard functions from the C programming language but also pre-defined Fluent macros that allow access to field variables, material property, cell geometry data, etc ... UDFs are useful because the Fluent software is not programmed to anticipate all needs, such as:

- Customization of boundary conditions, source terms in the governing equations, material properties, ...
- Customization of physical models,
- Solution initialization,
- Execute on Demand functions for customizing the postprocessing of the results.

UDFs can either be run *compiled* or *interpreted*. Compiled UDFs are translated in machine language through the use of a C++ compiler, which gives the best run-time efficiency. The drawback is the need to have a third-party compiler (Microsoft Visual Studio) linked with Fluent. Instead, for this CFD project you will use the own C++ interpreter of Fluent , which executes the C code on a "line-by-line" basis; the drawback is that interpreted UDFs are slower than if they were compiled.

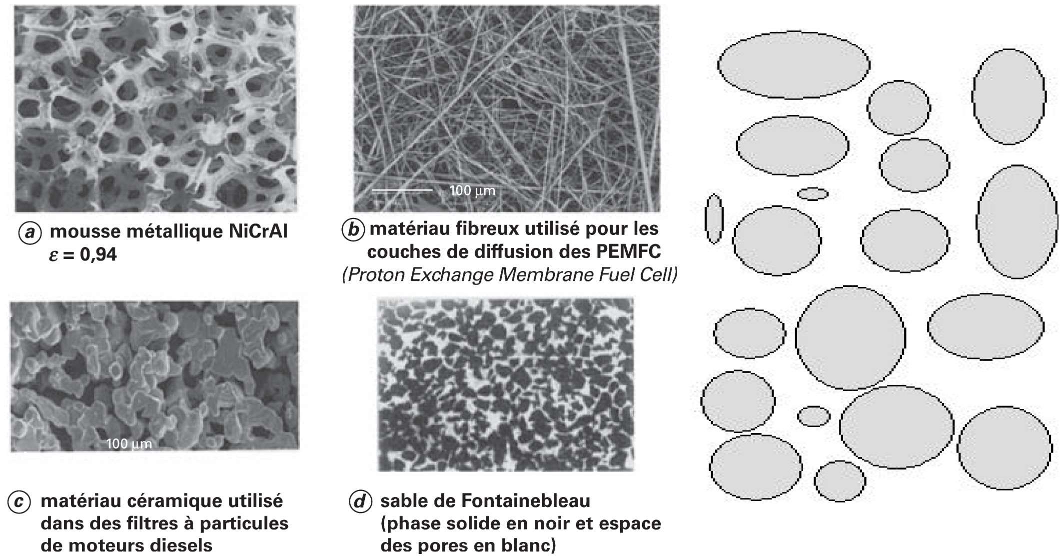

Figure 7: Examples of porous materials.

3.2 Notes on fluid flow through porous media

A porous medium is a material containing pores (voids), see Fig. 7 and Ref. [2]. The pores are typically filled with a fluid (liquid or gas). Many natural substances such as rocks and soil (e.g., aquifers, petroleum reservoirs), biological tissues (e.g. bones, wood, cork), and man made materials such as cements and ceramics can be considered as porous media. The concept of porous media is used in many areas of applied science and engineering: filtration, mechanics (acoustics, geomechanics, soil mechanics, rock mechanics), engineering (petroleum engineering, construction engineering), geosciences (hydrogeology, petroleum geology, geophysics), biology and biophysics, material science, etc.

A porous medium is most often characterized by its *porosity* ϕ :

$$\phi = \frac{\text{pore volume}}{\text{bulk volume}}, \tag{6}$$

and its *permeability* K (in m^2), which measures the ability of fluids to flow through the porous medium. Fluid flow through porous media is often determined from Darcy's law (1856), which states that the flow rate is proportional to the pressure drop as :

$$\boldsymbol{\nabla} p = -\frac{\mu}{K}\overline{u}, \tag{7}$$

where μ is the fluid viscosity. Darcy's law is used extensively in petroleum engineering to describe oil, water, and gas flows through petroleum reservoirs. Darcy's law is usually valid until Re $\simeq 10$, so when inertial effects become significant a correction term due to Forchheimer (1886) is added to Darcy's equation :

$$\boldsymbol{\nabla} p = -\frac{\mu}{K}\overline{u} - \frac{\rho}{K_{\mathrm{I}}}\overline{u}^2, \tag{8}$$

where K_{I} is the *inertial permeability.* Forchheimer's correction term is similar to the drag force $F = \frac{1}{2}\rho C_{\mathrm{D}} S\,\overline{u}^2$ exerted by fluid flow past solids.

More generally, the computation of fluid flow through porous medium uses the Navier-Stokes equations with Darcy and Forchheimer momentum sink terms (similar to the RHS of Eq. 8) for heuristically modelling the viscous and inertial resistance of the medium.

References

[1] H. Gamel and C. Muller. Etude des mouvements d'air place Thiers à Nancy : Influence de la végétation. Rapport de Projet de Master 1 MEPP, Université de Lorraine, 2010.

[2] E. Guyon, J.P. Hulin, and L. Petit. *Physical Hydrodynamics*. Oxford University Press, 2001.

[3] Y. Kurotani, N. Kiyota, and S. Kobayashi. Windbreak effect of Tsuijimatsu in Izumo Part 2, Summaries of technical papers of annual meeting. *AIJ Environmental Engineering*, I:745–746, 2002.

[4] R. Manceau. *Modélisation statistique de la turbulence*. cours école printemps, 2007.

[5] A. Mochida, Y. Tabata, T. Iwata, and H. Yoshino. Examining tree canopy models for CFD prediction of wind environment at pedestrian level. *Journal of Wind Engineering*, 96:1667–1677, 2008.

[6] Y. Tominaga, A. Mochida, R. Yoschie, H. Kataoka, T. Nozu, M. Yoshikawa, and T. Shirasawa. AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings. *Journal of Wind Engineering and Industrial Aerodynamics*, 96:1749–1761, 2008.