Ing. Luca Nicola Quaroni
Ing. Jorge Soto
Ing. Federico Lanteri
Prof. Gianandrea Vittorio Messa

POLITECNICO

MILANO 1863

Course of "Fluid Labs" A.A. 2025-2026

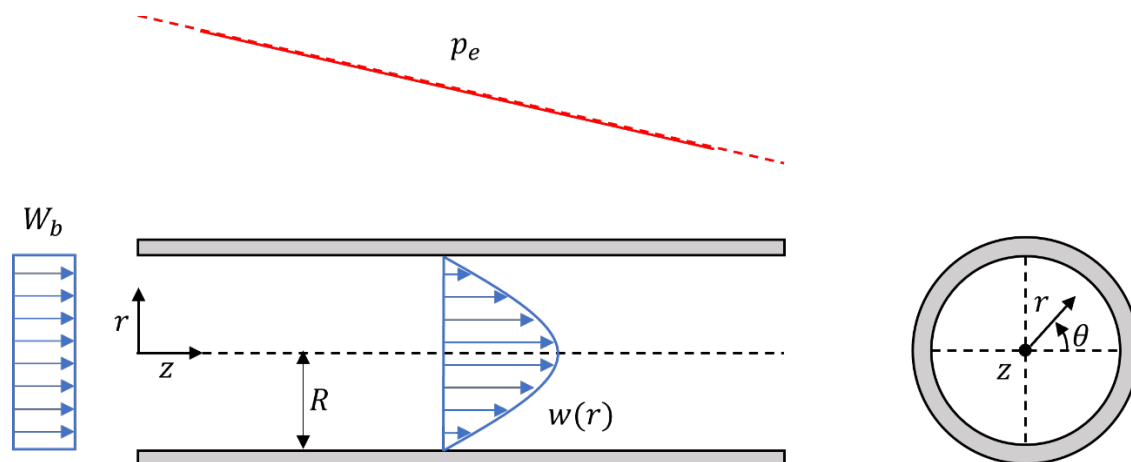# CFD Test Case 1
Laminar oil transport in a pipeline

Figure 1. Sketch of the case

Ing. Luca Nicola Quaroni
Ing. Jorge Soto
Ing. Federico Lanteri
Prof. Gianandrea Vittorio Messa

Fluids Labs A.A. 2025-26

Preliminary note: *in order to be able to import the PHOENICS results into MATLAB, it is necessary to set the correct data formatting option, selecting the PHI file format instead of the default option PHIDA. To this aim, in VR-Editor go to Options → File Format and select "Formatted" for both files. After being set the first time, the selected file format is stored and will be applied to all future simulations automatically.*

Case description

The first test case is the laminar flow of oil in a pipeline (Figure 1). If the pipeline is sufficiently long, one expects the flow to be fully developed, and this state must be reproduced in the CFD solution. However, if the inlet boundary condition is a uniform velocity profile, the flow will only reach a fully developed state asymptotically. A practical criterion must therefore be introduced to identify the pipe region where the flow can be treated as fully developed. The fully developed, laminar flow in a pipe admits an analytical solution, which is the cylindrical Poiseuille flow:

$$w\,(r,\theta,z) = w(z) = -\frac{1}{4\mu}\frac{dp_e}{dz}(R^2 - r^2) \quad u(r,\theta,z) = 0 \quad v(r,\theta,z) = 0$$

$$\frac{dp_e}{dz} = \text{const} < 0$$

$$\tau_{rz}(r,\theta,z) = -2\mu\,\frac{1}{2}\Big(\frac{\partial v}{\partial z} + \frac{\partial w}{\partial r}\Big) = -\mu\frac{dw}{dr} = -\frac{dp_e}{dz}\frac{r}{2}$$

where μ is the dynamic viscosity of the fluid, R is the inner radius of the pipe, u, v, w are the velocity components along directions θ, r, z (Figure 1 and Figure 2), p_e is the excess pressure (that is, the part of pressure which is not balance by gravity), and τ_{rz} is the only nonzero shear stress term.

Numerical simulation

The steady-state formulation of the Navier-Stokes equations are solved. The applied boundary conditions are inlet, outlet, wall and axis[1]. The computational burden of the simulations can be reduced by exploiting the axi-symmetry of the problem, which will be therefore simulated as 2D through a domain having the shape of a circular sector, as shown in Figure 2. At the inlet, a uniform z-velocity profile equal to W_b (bulk velocity) is imposed, whereas the r- and $-\theta$ velocity components are set to zero. At the outlet, the excess pressure is specified to a given value, which might be zero. The walls are assumed fixed and a no-slip condition is set at the boundaries, the fluid velocity is zero at the walls. Being in the laminar case, the wall shear stress is simply evaluated as the axial velocity in the near-wall cells divided by its distance from the wall and multiplied by the fluid viscosity. This set of boundary conditions results in a developing flow field which, starting from the uniform distribution imposed at the inlet section, reaches a fully developed configuration at a certain distance downstream of it.

Configuration of the problem

Pipe diameter = 150 mm,
Bulk velocity W_b = 0.45 m/s,
Fluid = heavy crude oil with density ρ = 910 kg/m³ and kinematic viscosity υ = 3.5·10⁻⁴ m²/s

---

[1] Note that other combinations of boundary conditions are possible for this problem. These include, for instance: (i) outlet (used as inflow), outlet, wall, and axis and (i) periodic boundary conditions with imposed mass flow rate, wall, and axis.
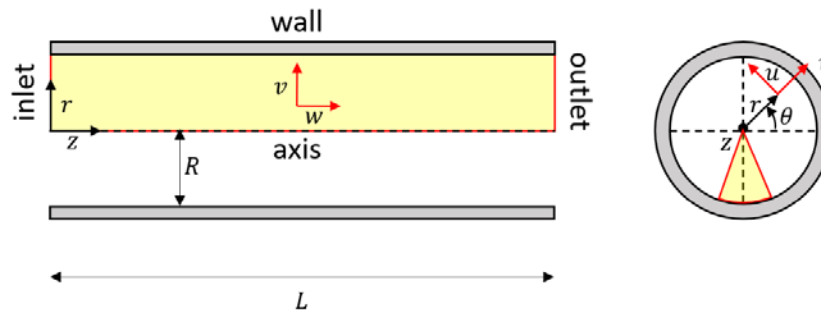
Ing. Luca Nicola Quaroni
Ing. Jorge Soto
Ing. Federico Lanteri
Prof. Gianandrea Vittorio Messa

Fluids Labs A.A. 2025-26

Figure 2. Domain and boundary conditions

Questions

Simulate the flow in PHOENICS, addressing the following issues.

1. Verify that the testing conditions indicated correspond to **laminar flow**. This is achieved by calculating the bulk Reynolds number and Re_b comparing it against the threshold value of about 2000.

2. Set the domain length, L, in such a way that **fully developed flow** conditions can be attained from the uniform velocity profile imposed at the inlet. Note that the goal is not to determine accurately the length required for flow development (often called "entry length"), which is, in principle, infinite. Instead, the aim it to identify a domain region that, in practice, could be regarded as representative of the fully developed state and could be therefore considered for extracting the relevant solution output (axial velocity profile, shear stress profile, hydraulic gradient).

   Hint: do not belittle the qualitative impact! Sometimes a qualitative criterion is more effective than a too abstract quantitative one.

   Hint: try to look at variables which have a clear physical meaning and are more directly understandable instead of metrics that, although meaningful, could look obscure. Remember that the intuitive understanding of the flow, corroborated by knowledge of fluid mechanics, is of utmost importance in applied CFD.

   Hint: since the fully developed state is reached asymptotically, whatever the chosen pipe region (or pipe section) is, strictly speaking, the CFD solution will not correspond to fully developed flow in any case. Hence, the CFD solution will be representative of a fully developed flow pending a given variability (or, if you prefer, uncertainty). The goal is to have this variability within acceptable limits for the purpose of the study – the key challenge is to understand what "acceptable limits" mean!

   Hint: many formulas have been reported in the literature for estimating the entry length of laminar pipe flow. Try to find some and compare against your CFD results. Pay attention to check whether the comparison is consistent: for instance, is the meaning of the "entry length" in the literature formulas the same that you are attributing to this parameter in your calculations? Also, if you can find many formulas providing different estimates, do not limit to those providing the best match with your calculations; instead, ask yourself "why do these formulas provide different estimates for the same parameters? Why do some formulas agree with my CFD results and others do not?"

3. Verify whether the CFD solution is **physically sound** from a qualitative point of view, that is, check whether there is "anything strange" in the results when inspecting the colour plots or the vector maps.

Ing. Luca Nicola Quaroni
Ing. Jorge Soto
Ing. Federico Lanteri
Prof. Gianandrea Vittorio Messa

Fluids Labs A.A. 2025-26

4. Assess the **numerical convergence** of the CFD solution. This requires, on the one hand, verifying that the CFD simulation must be regarded as converged with respect to the solution algorithm and, on the other hand, that the mesh is sufficiently fine to keep the discretization error within acceptable limit. The convergence with respect to the algorithm is achieved through the monitoring of the normalized whole-field residuals and probe values, setting up an adequate maximum number of iterations and a suitable global convergence criterion. The grid sensitivity study (often called "grid independence study", but not that no solution is grid independent in a strict sense) must be carried out based on suitable target parameters. Note that the target parameters are decided by the user, and all quantities subject of investigation must be proven grid-independent.

   Hint: as for the attainment of the fully developed state, in principle the convergence with respect to the algorithm and the grid independence are obtained asymptotically, requiring an infinite number of iterations (global convergence criterion equal to zero) and cells with infinitesimal size. Hence, the goal is to identify the global convergence criterion and the cells' sizes so that the residual variability (or, if you prefer, uncertainty), is within acceptable limits. But, once again, the key challenge is to understand what "acceptable limits" mean!

   Hint: how do you design the mesh pattern for this case? Why? Once you have defined the first mesh, how do you define the (at least) two others to carry out the grid independence study?

5. Validate the CFD solution in the fully developed region with respect to the analytical solution of Poiseuille flow. Since there exists an analytical solution for this case, this validation is called **benchmarking**, and its purpose is simply to self-verify the correct implementation of the case, as well as the numerical convergence. The validation of the CFD solution must be made in terms of the velocity $w(r)$ and the shear stress $\tau_{rz}(r)$ profiles, as well as in terms of the (excess) pressure gradient dp_e/dz. So, the analytical solution presented in class, and recalled in the "Case description" section, must be manipulated so that all quantities are expressed as a function of the bulk velocity, W_b, which is the known quantity in the CFD study (as this is the parameter imposed at the inlet section).

Supplementary question (optional)

6. Manipulate the analytical solution to obtain the trend of the friction factor f versus the bulk Reynolds number, Re_b, which is $f = 64/Re_b$. Compare the CFD results and the analytical solution in terms of f for the present case study. Extend the comparison to other values of Re_b (that is, for different velocities or different pipe diameters). Note that, for all cases, the flow must be laminar, thus having Re_b smaller than say 2000.

7. In the academic and professional practice, formal techniques for estimating the grid discretization errors are sometimes used, and one of the most well-known is the Grid Convergence Index (GCI) method by Roache. Such formal techniques are useful but they are not free from criticisms, including the difficulties in evaluating some input parameters and coefficients for complex cases. As an attachment, you are given a short paper by Ismail Celik from West Virginia University, in which a formal, systematic procedure for evaluating the grid discretization error is given, which generalizes the GCI method by Roache. If you are interested, employ this procedure for strengthening your grid independence assessment.

   Hint: do not limit yourself to apply the formal procedure and obtain the uncertainty estimates, but try to assess its strengths and, most important, its weaknesses.