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

POLITECNICO
MILANO 1863

Course of "Fluids Labs" A.Y. 2024-2025

Test Case 2

Turbulent water transport in a hydraulically smooth pipeline

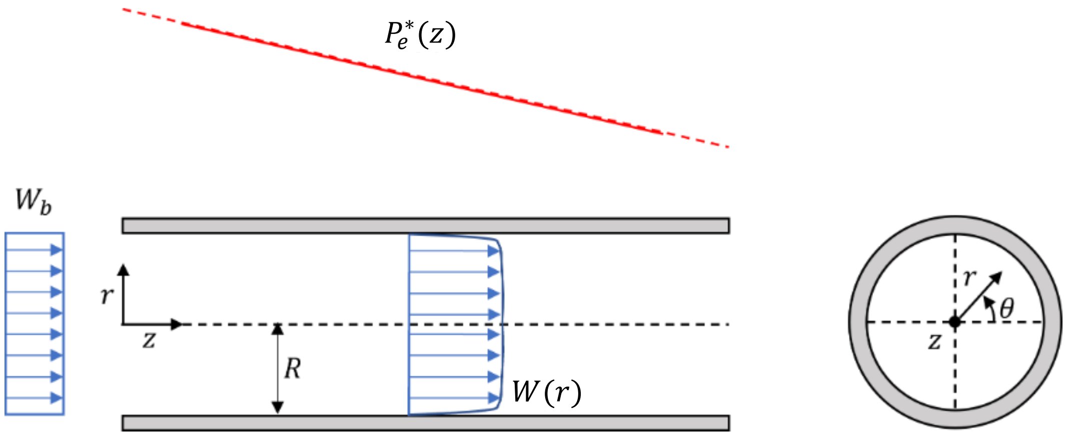

Figure 1. Sketch of the test case.

CASE DESCRIPTION

The second test case is the turbulent flow of water in a hydraulically smooth 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 a uniform velocity profile is imposed at the inlet boundary, the fully developed state is reached at a certain distance downstream of it, and the interest will be only about the fully developed region. Furthermore, due to the inherently unsteadiness of turbulence, considerations are only made in a time-averaged sense. Quantities of interest are the frictional losses along the pipeline, as well as radial profiles of axial velocity, shear stress (viscous and turbulent), and turbulent parameters. Unlike in the previous test case, in the present one no analytical solution is available; therefore, no benchmarking is possible. Instead, experimental data and empirical correlations are used for validation of the numerical set-up.

NUMERICAL SIMULATION

In this laboratory, PHOENICS is used to simulate the flow by solving the Reynolds-Averaged Navier-Stokes (RANS) equations coupled with the standard k-ε turbulence model. The wall function approach is adopted to model the effect of near-wall turbulence, and the equilibrium wall function of Launder and Spalding (1974) is employed. The computational burden of the simulations can be reduced by exploiting the axisymmetry of the mean flow field, which will be therefore simulated as 2D through a domain having the shape of a circular sector, as shown in Figure 2.

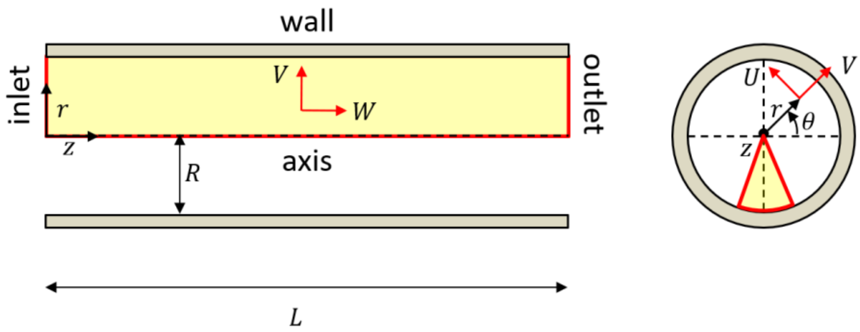

Figure 2 Computational domain and boundary conditions.

The boundary conditions employed to simulate the second test case are inlet, outlet, solid walls, and axis.

- At the inlet, a uniform z-velocity profile of value W_b is imposed, whereas the r- and θ- velocity components are specified as zero. No turbulence is assumed at the inlet; this is achieved by setting the turbulence intensity parameter TI to 0.
- At the outlet, a zero <u>external</u> P^* is imposed.
- At the solid walls, owing to the no-slip condition, the fluid velocity is zero. In the Finite Volume framework with staggered-grid arrangement, only the r-velocity is imposed at the wall. The no-slip condition is imposed for the z-velocity indirectly. On the one hand, the advection flux of variable W through the near-wall cell faces is set to zero. On the other hand, the diffusion flux of variable W through the near-wall cell faces (that is, the wall shear force) is specified by obtaining the wall shear stress from the equilibrium wall function of Launder and Spalding (1974). Note that, since the wall function approach is adopted, the dimensionless wall distance y^+ of the first grid nodes must be greater than 30 (and generally lower than about 130). Such dimensionless wall distance y^+ can only be calculated *a posteriori*.

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 D=0.10 m
Bulk-mean velocity W_b=0.75 m/s
Fluid = water at 20°C, treated as incompressible (ρ=998.23 kg/m^3, ν=10^{-6} m^2/s)
Pipe roughness ≈ 0

USEFUL PHOENICS VARIABLES
- YPLS is the non-dimensional wall distance of the first grid nodes in wall units.
- STRS is τ_w/ρ, that is, the squared friction velocity w_τ. Note that STRS uses the modulus of the wall shear stress. This must be considered when addressing flows with boundary layer separation and recirculation (not this case).
- ENUT is the kinematic eddy viscosity μ_t/ρ.
- KE is the turbulent kinetic energy k.
- EP is the turbulent dissipation rate ε.

Note: *Storage of variables YPLS and STRS in the .phi file must be specifically requested by the user through the panel Output → Derived variables.*
Note: *In order to improve the numerical estimate of the shear stresses, use arithmetic averaging for the diffusion coefficients of the variable W1. This is achieved by going to Models → Solution Control / Extra variables and then change from "Y" to "N" the Harmonic average button for variable W1.*

QUESTIONS
Simulate the flow in PHOENICS, addressing the following issues.

Preliminary calculations

1. Verify that the test conditions indicated correspond to a **turbulent flow**. This is achieved by calculating the bulk Reynolds number $Re_b = DW_b/\nu$ and 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"), but just to observe fully developed flow in the CFD solution. Note also that, in principle, fully developed flow is reached in an asymptotic manner, that is, at an infinite distance downstream of the inlet. So, you must develop yourself a practical criterion to assess that fully developed flow has been established; note that any criterion has strengths and weaknesses, so try to highlight them. **From this point onward, the interest is only about the fully developed flow region.**

   Hint: *The same considerations and suggestions presented for the previous test cases apply also to the present one. That is, do not strive to assess the attainment of fully developed flow through quantitative criteria that, in the end, might be arbitrary. Instead, do not belittle the qualitative impact, possibly complementing it with a quantitative evaluation only in a second stage. Also, compare your CFD results against available formulas, bearing in mind their nature and applicability. Finally, try to understand whether the nature of the flow (laminar and turbulent) affects the length required for flow development starting from the uniform inlet condition; to this aim, it might be useful to express the entry length in dimensionless form, normalizing it with respect to the pipe diameter.*

Inspection of the CFD solution and assessment of its physical consistency

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 them. Particularly, it is recommended to extract and inspect:

- the distribution of pressure P^* along the axis of the pipe.
- the profile of the Reynolds average velocity along the pipe radius, which can be expressed in the following two dimensionless forms
    - W/W_b versus r/R, being R the radius of the pipe
    - w^+ versus y^+, where $w^+ = W/\, w_\tau$ and $y^+ = yw_\tau/\nu$, being $w_\tau = \sqrt{\tau_w/\rho}$ the friction velocity and y the distance from the pipe wall ($y = R - r$). Present these results in the form of a log-log plot.
- the profile of the total shear stresses $\tau(r)$ and of its components $\tau_{visc}(r)$ and $\tau_{Re}(r)$ along the pipe radius, referring to the dimensionless variables τ/τ_w (τ_{visc}/τ_w and τ_{Re}/τ_w) versus r/R.

Note: *The values of τ_{visc} and τ_{Re} are obtained numerically from the constitutive model for the stresses tensor and the Boussinesq assumption for the Reynolds stresses tensor, that is,*

$$\tau_{visc} = \mu \frac{dw}{dr} \qquad \tau_{Re} = \mu_t \frac{dw}{dr}$$

Conversely, the wall shear stress must not be evaluated from the W1-velocity profile (why?), but referring to the PHOENICS output variable STRS.

- the profile of the turbulent kinetic energy, k, along the pipe radius, presented in dimensionless form as k/W_b^2 versus r/R
- the profile of the turbulent dissipation rate, ε, along the pipe radius, presented in dimensionless form as $\varepsilon R/W_b^3$ versus r/R

Assessment of the numerical convergence of the CFD solution

Assess the **numerical convergence** of the CFD solution. This requires, on the one hand, to verify that the CFD simulation must be converged with respect to the solution algorithm and, on the other, that the solution is grid independent. The convergence with respect to the solution algorithm is achieved through the monitoring of the normalized whole-field residuals, stability of spot values and the max/min corrections applied. The grid-independence study must be carried out based on suitable target parameters. Note that the target parameters are decided by the user, and <u>all quantities</u> subject of investigation must be proven to be grid-independent. Note that, since the wall function approach is adopted, the value of y^+ of the first grid nodes must be greater than 30 (and generally lower than about 130).

Validation of the CFD solution

Since no analytical solution is available for this case, the CFD results are compared against experimental data and some empirical formulas available in the literature. Specifically, the experimental data originate from the PhD thesis by Lars Even Torbergsen (1998), who investigated the fully developed pipe flow of air at similar Reynolds number to those of the present test case. The scope of the comparison against reference data is two-fold. On the one hand, it can be used to check that the proper selection of the modelling settings (here, the turbulence model) has been made; in this case, the term "validation" is used. On the other hand, it can be used to decide the most appropriate modelling settings, e.g., the best turbulence model within a given set of options; in this case, the term "calibration" is used. At this stage, you are requested to perform a validation of the CFD setup.

Frictional losses

Firstly, the validation is carried out by referring to the frictional losses, which are quantified through the friction coefficient f. Note that the friction coefficient f, the skin friction coefficient C_f, the hydraulic gradient J, the pressure gradient $-dP^*/dz$ and the wall shear stress τ_w are interchangeable parameters related to the frictional losses. Thus, a validation with respect to say f is practically equivalent to a validation with respect to say, e.g., τ_w or J.

The friction factor obtained from the CFD simulation, f^{CFD} (note: how will you calculate it?), must be compared against the following.

- the value obtained experimentally by Torbergsen (1998), equal to 0.0190
- the Moody chart (graphic estimate)
- the formula by Prandtl for hydraulically smooth pipes

$$\frac{1}{\sqrt{f}} = -2\log_{10}\left(\frac{2.51}{Re_b\sqrt{f}}\right)$$

- the formula by Haaland for hydraulically smooth pipes

$$\frac{1}{\sqrt{f}} = -1.8\log_{10}\left(\frac{6.9}{Re_b}\right)$$

Note: *The variability of these estimates might be regarded as some sort of inherent uncertainty in the evaluation of the friction factor. This aspect must be considered when interpreting the results of the validation process.*

Profile of the Reynolds-averaged axial velocity along the pipe radius

Secondly, the validation is carried out by referring to the Reynolds averaged axial velocity profile. For these variables, the terms of comparison are as follows.

- the data obtained experimentally by Torbergsen (1998), and presented in dimensionless form W/W_b versus r/R. Specifically, two data series are provided, which correspond to two different post-processing operations on the experimental data.
- the power law profile of Nikuradse:

$$\frac{W}{W_0} = \left(1 - \frac{r}{R}\right)^{\frac{1}{n}}$$

  where W_0 is the axial velocity in the pipe centerline and the power n is equal to the friction factor to power -0.5, that is $n = f^{-0.5}$. In order to evaluate n, the estimates of f from the formulas previously listed (Moody chart, Prandtl's formula, Haaland's formula) shall be used. Thus, the comparison will be made against three different literature curves.

  Note: *In this formulation, the Nikuradse profile is expressed in dimensionless form as W/W_0 versus r/R. For coherence with the previous term of comparison, it might be convenient to express the Nikuradse profile as W/W_b versus r/R; such relation could be obtained analytically by manipulating the formula here above, and it results equal to*

$$\frac{W}{W_b} = \frac{(2n+1)(n+1)}{2n^2}\left(1-\frac{r}{R}\right)^{\frac{1}{n}}$$

- the wall function of Launder and Spalding (1974), which is expressed in dimensionless form as w^+ versus y^+ and it reads as follows

$$w^+ = \begin{cases} y^+ & y^+ < 11.6 \\ \frac{1}{\kappa}\ln\left(Ey^+\right) & 11.6 < y^+ \leq 130 \end{cases}$$

  where $\kappa = 0.41$ is the Von Karman's constant, and E is a roughness coefficient set equal to 8.6 for hydraulically smooth walls. Compare the $w^+(y^+)$ profile obtained from the CFD simulation against the estimate from the equilibrium wall function in its range of validity; it is recommended to present the results in a log-log plot. Which considerations can you make?

<u>Profile of the turbulent kinetic energy along the pipe radius</u>

Then, the validation is extended to the profile of the turbulent kinetic energy along the pipe radius. It is recommended to present the results in dimensionless form as k/W_b^2 versus r/R. The terms of comparison are as follows.

- the data obtained experimentally by Torbergsen (1998), already presented in dimensionless form k/W_b^2 versus r/R.
- the following empirical formula reported in the PhD thesis of Kam Hong Ng (1971), once again already presented in dimensionless form k/W_b^2 versus r/R.

$$\frac{k}{W_b^2} = \frac{f}{8}\left[1 + \frac{2}{3}\left(\frac{r}{R}\right) + \frac{10}{3}\left(\frac{r}{R}\right)^3\right]$$

  Also in this case, the estimates of f from the formulas previously listed (Moody chart, Prandtl's formula, Haaland's formula) shall be used.

<u>Profile of the turbulent dissipation rate along the pipe radius</u>

Finally, the validation is focused on the profile of the turbulent dissipation rate. It is recommended to present the results in dimensionless form as $\varepsilon R/W_b^3$ versus r/R. The terms of comparison are as follows.

- the data obtained experimentally by Torbergsen (1998), already presented in dimensionless form $\varepsilon R/W_b^3$ versus r/R. Specifically, three data series are provided, which correspond to different methods for estimating ε form the same experimental data point. The difference between the three series underlines the challenges in providing experimental estimates of ε.
- the following relation at the basis of the standard *k-ε* turbulence model:

$$\varepsilon(r) = C_d\frac{k^{\frac{3}{2}}(r)}{l_m(r)} = 0.1643\frac{k^{\frac{3}{2}}(r)}{l_m(r)}$$

  where l_m is the mixing length, that is, a characteristic length scale of turbulence. In order to apply the formula here above, one must know the radial profiles of k and l_m. These might be obtained from the cubic correlation by Kam Hong Ng (1972), presented previously, and from the following empirical distribution of Nikuradse:

$$l_m = R\left[0.14 - 0.08\left(\frac{r}{R}\right)^2 - 0.06\left(\frac{r}{R}\right)^4\right]$$

Further questions

1. The wall function approach employed in this test case requires the near wall cells to be characterized by y^+ above 30 and below say 130. What does it happen if this condition is not satisfied? Run the numerical simulations disregarding this constraint, that is, calculate the CFD solution on fine, near-wall meshes in the range $y^+ = 1 \div 130$. Analyze what happens when y^+ is outside the admissible range, that is, for y^+ lower than 30, by referring to the friction factor, the Reynolds-averaged axial velocity profile, and the radial profiles of the turbulent variables k and ε.
2. As an alternative to the wall function approach, the low-Reynolds one relies on the solution of the near-wall flow field up to the viscous sub-layer. This requires, on the one hand, defining very fine meshes close to the wall, so that the near-wall cells are characterized by $y^+ \approx 1$ and, on the other hand, modifying the turbulence model equations close to the wall by means of dumping functions. Solve the same test case using a low-Reynolds turbulence model, specifically, the Two-Layer k-ε model by Rodi (1991); to activate it in PHOENICS, *Models → Turbulence models → Low Re KE variants → Two layer KE*. Pay attention to the definition of the mesh and make your own considerations on strengths and weaknesses of the low-Reynolds approach against the wall function one. Repeat the analysis and the validation also for this new turbulence model and, particularly, analyze the information that you can gather about the shear stress profile and its components (viscous and turbulent) using the "low Reynolds" approach.

REFERENCES

Launder, B.E., Spalding, D.B. The numerical computation of turbulent flows. Comput. Meth. App. Mech. Eng. 3 (1974) 269-289.

Moody, L.F. Friction factors for pipe flow. Transaction of the ASME 66 (1944) 671-684.

Ng, K.H. Predictions of turbulent boundary-layer developments using a two-equation model of turbulence. PhD Thesis, Imperial College London, 1971.

Rodi, W. Experience with two-layer models combining the k-ε model with a one equation model near the wall. AIAA-91-0216, 29th Aerospace Sciences Meeting, January 7-10, Reno, Nevada, USA, 1991.

Torbergsen, L.E. Experiments in turbulent pipe flow. PhD Thesis, Norwegian University of Science and Technology, 1998.