1. Notes on the program
2. Assigned exercises (submission via Blackboard; deadline: Monday 3rd March, 6 pm)

---

1. NOTES ON THE PROGRAM

1.1 Accessing and Running the Program

The following files must be downloaded from Blackboard or the CFD web page:

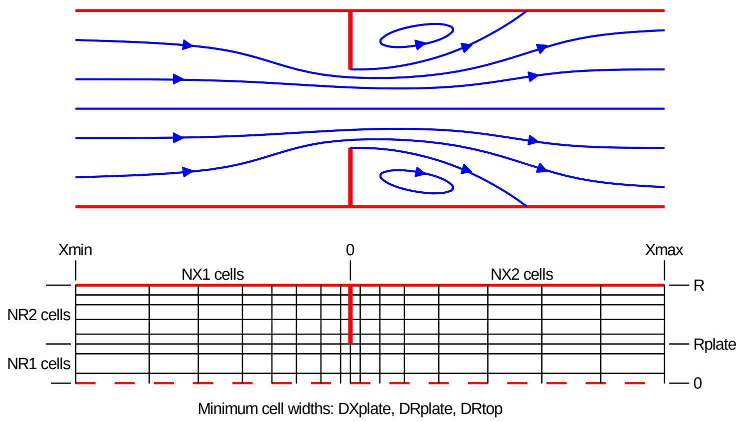
`plate.exe`	(graphical user interface)
`gridplate.exe`	(grid generator)
`streamplate.exe`	(CFD solver)

Start by double-clicking the graphical user interface `plate.exe`. All data files and plot files produced will be saved in the folder from which you run the program.

1.2 The Flow Considered

The program simulates axisymmetric laminar or turbulent flow through an orifice plate in a pipe. By relating the pressure drop across the plate to the volume flow rate this can be used as a cheap, but lossy, flowmeter.

The code uses a structured mesh produced by the grid generator. As the geometry is axisymmetric, only one half section is computed. Fully-developed flow (from an initial 1-d pipe-flow calculation) is applied at inflow.



Variables are non-dimensionalised using bulk (i.e. average) velocity U_b, pipe diameter D and fluid density ρ. Thus, $X = x/D$ and $U = u/U_b$ etc. The Reynolds number is defined as $U_b D/\nu$.

1.3 Program Operation

The sequence of operations for a particular geometry (and, if used, turbulence model) is:

- generate a grid;
- solve 1-d (to calculate the inflow profile);
- solve 2-d;
- analyse/plot.

No logic is built in to ensure that this sequence is followed. It is your responsibility to ensure that if you change or reset the grid or flow parameters then you re-run the 1-d calculation to establish a new inflow profile. As all transfer between components is done by reading and writing files, you can exit the program at any stage and restart from the same point.

1.4 Main Buttons

Grid

[Set up]	set default grid parameters
[Edit]	edit grid parameters after set-up
[Run]	generate the grid

Case

[Set up]	set test-case parameters (here, just an interpolation grid)
[Edit]	edit test-case parameters

Solver

[Set up (laminar)]	set parameters for a default laminar calculation ($\mathrm{Re} = 50$)
[Set up (turbulent)]	set parameters for a default turbulent calculation ($\mathrm{Re} = 10^5$), with the standard $k - \varepsilon$ eddy-viscosity model and wall functions
[Edit]	edit solver parameters
[Run (1d)]	run a 1-d calculation to get the fully-developed inflow profile
[Run (2d)]	run a 2-d calculation with orifice plate

Plot

[Set up (near)]	set parameters for a default plot focused on the plate
[Set up (far)]	set parameters for a default plot showing the whole domain
[Edit]	edit individual plot parameters after set-up
[Grid]	plot the grid
[Profiles]	plot mean-velocity profiles along the channel
[Vectors (all)]	plot velocity vectors at all grid nodes
[Vectors (reg)]	plot velocity vectors interpolated onto a regular grid
[Streamlines]	plot streamlines
[Pressure]	plot pressure field
[Turbulent KE]	plot turbulent-kinetic-energy field (turbulent case only)
[Inflow]	plot the inflow mean-velocity profile
[cp]	plot a graph of pressure coefficient
[cf]	plot a graph of skin-friction coefficient

Hard Copy

[Save plot]	save the current view as a plot file (`png` format)
[Quit]	does exactly what it says!

The [Edit] buttons launch menus for grid, case, solver and plot parameters. The program carries out a few checks to exclude unreasonable values, but it is not foolproof!

The latest field plots become available whenever the CFD solver backs up data. The inflow velocity profile is only available after completion of the 1-d CFD calculation. Velocity vectors interpolated to a regular grid are only available after completion of the 2-d calculation. All of these are controlled by parameters in `Solver > [Edit] > [Numerical method]`.

2. ASSIGNED EXERCISES

2.1 Laminar Flow

<u>(L1) Inflow Profile</u>
Generate the default grid and set up the default laminar flow (Re = 50). Run the 1-d simulation.

(i) Record the skin-friction coefficient c_f (calculated by the program and displayed in its command window at the end). c_f is defined by

$$c_f = \frac{\tau_w}{\frac{1}{2}\rho U_b^2}$$

where τ_w is the wall shear stress. Compare it with the theoretical value

$$\frac{16}{\mathrm{Re}}$$

where Re is the Reynolds number based on pipe diameter and average velocity.

(ii) Plot the inflow profile and include the plot in your report. The theoretical velocity profile for laminar flow in a round pipe is

$$\frac{u}{U_b} = 2\left[1 - \left(\frac{r}{R}\right)^2\right]$$

The inflow profile is recorded in file `inprof.dat` under the first two columns, headed "`r/D`" and "`u/Ub`". Obviously, $R = D/2$. Choose 3 representative r/D values and compare the numerical values of u/U_b with the exact solution. Include these three comparison values in your report. Give a reason for any small disparity between computed and analytical values.

(iii) Show how the non-dimensional radius and velocity data in file `inprof.dat` can be used to calculate c_f, and confirm that it gives the same result as stated by the computer program.

<u>(L2) Flow Pattern</u>
For the default laminar flow, run the 2-d simulation. Plot streamlines, velocity profiles and velocity vectors (interpolated onto a regular grid) and include these plots in your report. Use a sensible plot region that illustrates the important features of the flow.

<u>(L3) Flow Topology</u>
Record all separation or reattachment points on the pipe wall or orifice plate. The code reports those on the wall; you can estimate those on the orifice plate from your plots. State which correspond to separation and which to reattachment points. Between which x/D values is there net backflow adjacent to the pipe wall?

<u>(L4) Under-Relaxation</u>
Using the `Solver > [Edit] > [Numerical method]` parameters change the output interval to 1 to see the exact iteration count. Run the default 2-d case with successive values of momentum under-relaxation factor, increasing it from 0.6 in steps of 0.05 until convergence

cannot be obtained. For each under-relaxation factor, record whether convergence to the predefined tolerance is obtained and, if so, the number of iterations required.

What is under-relaxation and why is it used? What is the effect of under-relaxation on: (i) the rate of convergence of the solution; (ii) the final solution itself?

(L5) Grid Dependence

Edit the grid parameters file to increase grid density in each direction by a factor of 2; i.e. double all the cell-number parameters (`NX1`, `NX2`, `NR1`, `NR2`). Re-run the grid generator. Reset the default laminar-flow parameters. Run 1-d and 2-d calculations.

What is the reported downstream reattachment point on the finer grid? Are there any significant differences between computed flows on coarser and finer grids? As well as increasing the total number of cells, what other grid parameters could one adjust here?

(L6) Change of Reynolds Number

Reset the default grid parameters and re-run the grid generator. Edit the solver parameters to use Reynolds numbers of 25, 50, 100, 200. For each, compute 1-d and 2-d solutions and record the position of furthest downstream reattachment. Plot a graph of the non-dimensional recirculation length (L_R/D) against Re. State and explain the variation of L_R/D with Re.

2.2 Turbulent Flow

(T1) Inflow – Comparison With Log-Law Velocity Profile

Set up and generate the default grid. Set up the default turbulent flow. Run the 1-d simulation. Plot the inflow velocity profile and include it in your report. Compare with the laminar inflow profile and give reasons for the differences.

As in the laminar case, record the reported value of the wall skin-friction coefficient c_f and compare it with the value obtained from the Colebrook-White equation for smooth pipes. (Refer to your Hydraulics 2 notes if necessary; note that friction factor $\lambda = 4c_f$.)

Friction velocity u_τ is defined by $\tau_w = \rho u_\tau^2$. Determine the non-dimensional value u_τ/U_b from the skin-friction coefficient c_f reported by the code. From the computed inflow velocity profile in `inprof.dat`, exclude the last data point and plot a graph of u/u_τ against $\ln(u_\tau y/\nu)$, where y is the distance from the pipe wall. Include this graph in your report, together with u_τ/U_b. Use the graph to estimate constants κ and E in one form of the universal log-law profile:

$$u = \frac{u_\tau}{\kappa} \ln\left(E \frac{u_\tau y}{\nu}\right)$$

(Use data from `inprof.dat`, but don't change the file itself or 2-d calculations won't run!)

(T2) Flow Pattern

Compute the 2-d flow. Using an appropriate plot window, plot the streamlines, velocity profiles and velocity vectors (on a regular mesh) and include these in your report.

(T3) Turbulent Kinetic Energy

Plot turbulent kinetic energy and include it in your report. What is the maximum non-dimensional turbulent kinetic energy k/U_b^2 in this flow domain? What features of the mean flow correlate with regions of high turbulence intensity?

(T4) Turbulent Diffusion

Compare the reattachment length for this turbulent flow at a Reynolds number of 10^5 with that for a laminar flow at $\mathrm{Re} = 200$ (part L6). Why is the non-dimensional reattachment length in the turbulent case shorter than for a laminar flow at a much smaller Reynolds number?

(T5) Pressure Distribution

Plot the pressure distribution near the plate and include the plot in your report. What do you notice (qualitatively) about the pressures on the upstream and downstream faces of the plate?

(T6) Surface Coefficients

Plot the line graphs of pressure coefficient c_p and friction coefficient c_f and include them in your report. What is observed about these coefficients (a) at a reattachment point; (b) well upstream? Note that these coefficients are non-dimensional pressure and non-dimensional stress:

$$c_p = \frac{p}{\frac{1}{2}\rho U_b^2}, \qquad c_f = \frac{\tau_w}{\frac{1}{2}\rho U_b^2}$$

(T7) Loss Coefficients

By balancing pressure and friction forces it may be shown that, in fully-developed pipe flow, the pressure gradient is proportional to the skin-friction coefficient. In non-dimensional terms:

$$\frac{\mathrm{d}c_p}{\mathrm{d}(x/D)} = -4c_f$$

(i) Using the value of c_f recorded by the code in part T1, calculate the drop in pressure coefficient that would occur along the domain if there was no orifice plate (i.e. in fully-developed flow).

(ii) In practical applications, pressure drops in pipe components such as valves, bends and local constrictions are quantified by loss coefficients K such that

$$\Delta p = -K\left(\frac{1}{2}\rho U_b^2\right)$$

From part T6 (or, better, the actual data in file `coefficients.dat`), use c_p to calculate K for this geometry.