1. Notes on the program
2. Assigned exercises

Submit to Canvas as a single PDF file. Deadline: Monday March 9th (week 6), 2 pm.

---

1. NOTES ON THE PROGRAM

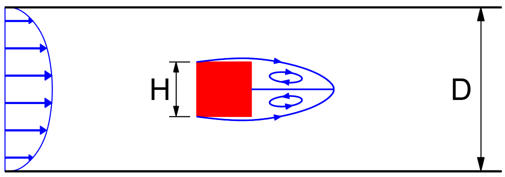
1.1 Accessing and Running the Program

The following files must be downloaded from Canvas into a temporary folder:

`block.exe`	(graphical user interface)
`gridblock.exe`	(grid generator)
`streamblock.exe`	(CFD solver)

Start by double-clicking the graphical user interface `block.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 2-d laminar or turbulent flow around a rectangular block in a channel.

The code uses multi-block structured meshes produced by a grid generator. Fully-developed flow (determined by an initial 1-d channel-flow calculation) is applied at inflow.

All variables are non-dimensionalised using the bulk (i.e. depth-averaged) velocity U_b, object height H and fluid density ρ. Thus, $X = x/H$ and $U = u/U_B$ etc. The Reynolds number is defined as $U_B H/\nu$ and the blockage ratio is H/D, where D is the depth of the channel.

Such a flow is prone to time-dependent vortex shedding. To avoid this and run as steady state we will keep the Reynolds number low in laminar flow and the blockage ratio quite high.

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 get 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 ($\text{Re} = 10$)
[Set up (turbulent)]	set parameters for a default turbulent calculation ($\text{Re} = 10^5$), 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 over block or cylinder

Plot

[Set up (near)]	set plot parameters for a default plot focused on the object
[Set up (far)]	set plot parameters for a default plot showing the whole domain
[Edit]	edit individual plot parameters after set-up
[Grid]	plot the grid
[Blocks]	plot the block structure
[Profiles]	plot mean-velocity profiles along the channel
[Vectors (all)]	plot velocity vectors at all grid nodes
[Vectors (reg)]	plot velocity vectors interpolated on a regular grid
[Streamlines]	plot streamlines
[Pressure]	plot pressure contours
[Turbulent KE]	plot turbulent-kinetic-energy contours (turbulent case only)
[Vorticity]	plot vorticity
[Inflow]	plot the inflow mean-velocity profile

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 a 1-d CFD calculation. Vectors interpolated to a regular grid are only available after completion of a 2-d CFD calculation.

The program reports the non-dimensional friction velocity u_τ / U_b and the drag and lift coefficients based on streamwise and cross-stream forces per unit depth, f_x and f_y.

2. ASSIGNED EXERCISES

2.1 Laminar Flow

(L1) Inflow Profile

Set up and generate the default grid. Set up the default laminar flow. Run the 1-d simulation. Plot the inflow profile and include the plot in your report.

The theoretical profile for laminar flow in such a channel (walls at $y = \pm\frac{1}{2}D$) is

$$\frac{u}{U_B} = \frac{3}{2}\left(1 - 4\frac{y^2}{D^2}\right)$$

The inflow profile is recorded in file `inprof.dat` under the first two columns, headed '`y/H`' and '`u/Ub`'. The default grid has $D = 4H$. Compare your computed values with this theoretical profile and give a reason for any small disparity between computed and analytical values.

(L2) 2-D Test Case

For the default laminar flow (Re = 10), run the 2-d simulation. Plot streamlines, velocity profiles, pressure distribution and velocity vectors (on a regular grid) and include these in your report. Choose a sensible plot region that encompasses the important features of the flow.

(L3) Under-Relaxation

Edit the solver parameters. From the [`Numerical method`] options change output interval to 1. 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 you can no longer obtain a converged solution. 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?

(L4) Effect of Reynolds Number

Run the 2-d case with successive Reynolds numbers Re = 10 (default), 20, 30, 40, 50. (Vortex shedding occurs for larger Reynolds numbers.) For each Re record drag coefficient and non-dimensional length of reversed flow. Explain, physically, the effect of Reynolds number.

(L5) Grid Dependence

Improve grid resolution by multiplying each of the grid parameters `dxmin`, `dxmax`, `dymin`, `dymax` by 0.5. Re-run the grid generator. Reset the default laminar-flow parameters.

Run 1-d and 2-d calculations.

By allusion to drag coefficient and non-dimensional recirculation length, comment on whether you think the flow has been satisfactorily grid-resolved.

(L6) Flow Asymmetry

By changing the centreline of the block from $y/H = 0$ (default) successively to 0.5, 1, 1.25 investigate the effect on drag and lift coefficients. (A negative lift coefficient simply means the relevant force component is downward). Comment on the differences in flow behaviour between a symmetrically-placed block ($y/H = 0$) and one close to a wall ($y/H = 1.25$).

2.2 Turbulent Flow

(T1) Inflow

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.

The code outputs non-dimensional friction velocity u_τ/U_B. Record this in your Report. Wall shear stress τ_w is related to friction velocity by $\tau_w = \rho u_\tau^2$. By balancing forces for the whole domain, compute the value of the non-dimensional pressure gradient. Show all calculations.

Using the data from file `inprof.dat`, and the value of the non-dimensional friction velocity u_τ/U_B reported by the program, exclude the first and last data points (which are on the channel walls) and plot a graph of u/u_τ against ln $(u_\tau y_n/\nu)$, where y_n is the distance from the *nearest* wall. Include this graph in your report, together with the reported value of u_τ/U_B, and use the graph to estimate constants κ and B in the universal log-law profile

$$\frac{u}{u_\tau} = \frac{1}{\kappa}\ln\frac{u_\tau y}{\nu} + B$$

(T2) Effect of Asymmetry

Repeat the calculations of L6 with successive block centreline positions of $y/H = 0, 0.5, 1.0, 1.25$. Comment physically on the difference between laminar and turbulent flow.

(T3) Effect of blockage

By changing grid parameters `ymin` and `ymax` compute turbulent flow around the block for symmetric blockage ratios of 1/4 (default), 1/3 and 1/2. Record and describe the effect on drag coefficient, recirculation length and other flow features and provide a physical explanation for the changes with blockage ratio.