

1. Notes on the program
2. Assigned exercises

Submission deadline: Tuesday 5th March (6 pm)

1. NOTES ON THE PROGRAM

1.1 Accessing and Running the Program

The following files must be downloaded from Blackboard:

<code>bluff.exe</code>	(graphical user interface)
<code>gridbluff.exe</code>	(grid generator)
<code>streambluff.exe</code>	(CFD solver)

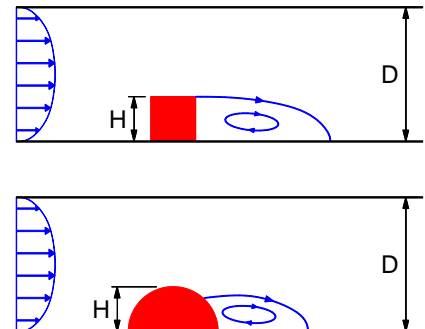
Start by double-clicking the graphical user interface `bluff.exe`. All data files and plot files produced will be saved in the folder from which you run the program.

1.2 The Flow Considered

The program simulates 2-d laminar or turbulent flow over a rectangular block or a semicircular cylinder.

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.



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 (block)]	set default grid parameters – rectangular block
[Set up (cylinder)]	set default grid parameters – semicircular cylinder
[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 ($Re = 10$)
[Set up (turbulent)]	set parameters for a default turbulent calculation ($Re = 10^5$), 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 parameters for a default plot focused on the object
[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
[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)
[General scalar]	plot a general scalar, as determined by the plot parameters
[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 the 1-d CFD calculation. Vectors interpolated to a regular grid are only available after completion of the 2-d CFD calculation.

2. ASSIGNED EXERCISES

2.1 Laminar Flow

(L1) Inflow Profile

Set up and generate the default grid for a rectangular block. 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 a channel is

$$\frac{u}{U_B} = 6 \frac{y}{D} \left(1 - \frac{y}{D}\right)$$

The inflow profile is recorded in file `inprof.dat` under the first two columns, headed ‘Y’ and ‘U’. (Remember that these contain the non-dimensional values y/H and u/U_B respectively.) The default grid has $D = 5H$. Choose 3 representative y/H 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 the 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 (the last interpolated onto a regular grid) and include these plots in your report. Choose a sensible plot region that encompasses the important features of the flow.

(L3) Flow Topology

Include in your report the coordinates of all flow-separation or reattachment points on the channel wall (as reported by the program) or block (as determined, approximately, by flow visualisation), indicating which are separation and which are reattachment.

Between which values of x/H is there backflow adjacent to the channel wall? Does the flow separate at the *upstream* top corner of the block?

(L4) Under-Relaxation

Edit the solver parameters. From the [Numerical method] options change the 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 (patience is a virtue!). 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 `NXupstream` etc.). 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?

2.2 Turbulent Flow

(T1) Inflow

Set up and generate the default grid for a rectangular block. 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 the non-dimensional friction velocity u_τ/U_B . Record this in your Report. The wall shear stress τ_w is related to the friction velocity by $\tau_w = \rho u_\tau^2$. By balancing forces for the whole domain, compute 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) Advection scheme

Change the output interval to 1. Run the 2-d simulation to convergence with the default UMIST advection scheme. Record the downstream reattachment point and the number of iterations required for convergence of the 2-d calculation.

Change the advection scheme for both momentum and turbulence scalars to UPWIND. (You do *not* need to re-run the 1-d calculation, which doesn't depend on advection). Run the 2-d calculation from scratch and record the downstream reattachment point and the number of iterations required for convergence.

Now create a finer grid by doubling the number of grid cells in each direction as in L5 and run a 1-d calculation (with either advection scheme). For each of the UMIST and UPWIND schemes record the downstream reattachment point and the number of iterations required for convergence of the 2-d calculation. Which scheme requires less iterations for convergence and why might you expect this? Which scheme is less sensitive to the mesh size and why?

(T3) Flow Pattern

Return to (which probably means re-run!) the converged (UMIST) solution from the default grid about the rectangular block, plot the streamlines and velocity profiles and include the plots (with a suitably refined plot window) in your report.

Does the flow separate at the *upstream* top corner of the block? If so, does it reattach on the top of the block?

(T4) Turbulent Kinetic Energy

Using the converged (UMIST) solution from the default grid about the rectangular block, produce one sensible plot of the 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?

(T5) Pressure Distribution

Using the converged (UMIST) solution from the default grid about the rectangular block, use the [Set near] graphics button to focus attention on the block. Plot the shaded pressure distribution in the vicinity of the block and include the plot in your report. What do you notice (qualitatively) about the pressures on the upstream and downstream faces of the block?

(T6) Effect of Blockage

For the rectangular block, the program reports (pressure, viscous, overall) drag coefficients

$$c_D = \frac{f_x}{\frac{1}{2}\rho U_B^2 H}$$

where f_x is streamwise force per unit span. By changing non-dimensional channel depth D/H in the grid-parameter file to values of 2, 3, 4, 6, 8, and running grid/1-d/2-d solver calculations for each, record the overall drag coefficient c_D and plot a graph of c_D against blockage ratio H/D . Comment on how drag changes with blockage and give a physical reason why.

(T7) Semi-Circular Cylinder – Post-Processing Exercise

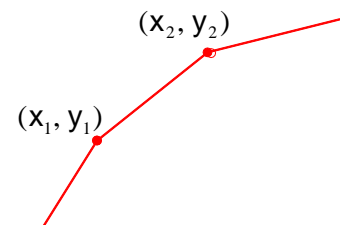
Using the default grid for a semi-circular cylinder, and the default turbulent-flow parameters, compute the flow. Plot the shaded pressure distribution and include the plot in your report.

The solver creates a file `surface.dat`, containing columns

P, U, V, X1, Y1, X2, Y2,

corresponding to non-dimensional pressure $p/(\rho U_B^2)$, non-dimensional near-wall velocity components u/U_B and v/U_B at the centre of a wall segment, and non-dimensional coordinates x/H and y/H at the ends of each wall segment. Use this output data to calculate:

- (i) the pressure drag coefficient;
- (ii) the coordinates of any separation or reattachment points *on the semi-circular cylinder*, indicating whether each is a separation or a reattachment point.



You must show and explain your working.