

# Applied Computational Fluid Dynamics (Applied CFD) MEng/M.Sc. Module

## General Information.

Dr Hervé MORVAN, Module Convenor  
School of Civil Engineering, The University of Nottingham, UK  
[Herve.Morvan@Nottingham.ac.uk](mailto:Herve.Morvan@Nottingham.ac.uk)

### Table of Content:

<b>EXAMINATION AND COURSEWORK</b>	<b>2</b>
<b>PROJECTS</b>	<b>2</b>
Backward facing step	2
Flow in an Axisymmetric Sudden Expansion	3
Airfoil Aerodynamics	3
Venturi	4
Flow through an Orifice Plate	5
Turbulent Flows in Pipe Bends	6
Turbulent Flows in Pipe Bends II	6
T-Junction I	6
T-Junction II	6
Zig-Zag Classifier	7
<b>REFERENCES</b>	<b>7</b>

## General Information.

### 1. Examination and Coursework

A written exam will be given at the end of the module. It will count for 50% of the final mark. **The aim of the examination will be to test the students of their background knowledge.** It will be based on the material presented and discussed during the lectures and practicals.

The **coursework**, on the other hand, should test the students' practical, engineering sense. It should therefore be based on the implementation of a code, some numerical analysis and the writing of an engineering report addressing the problem given to the student. It should represent about **30-40 hours of work**, outside class and **will be done individually**, although several students may undertake the same project. The submission of a calculation or result note including **3,500 words (plus graphs)** by the term end to their CFD tutor is expected together with quantitative and qualitative charts or pictures.

The necessary files are at <http://www.nottingham.ac.uk/cfd/Teaching/H24CFD/>.

**The coursework should be submitted by the last Friday of term, 15:00, to The School of Civil Engineering General Office.** Normal penalties will apply after that deadline. The coursework will count towards 50% of the mark for the module.

### 2. Projects

Here is the list of projects proposed to you this year.

#### 2.1. Backward facing step

This is a typical CFD problem used to test turbulence models. It is also a configuration found in many industrial and natural conditions (pipe network, firths, estuaries etc.).

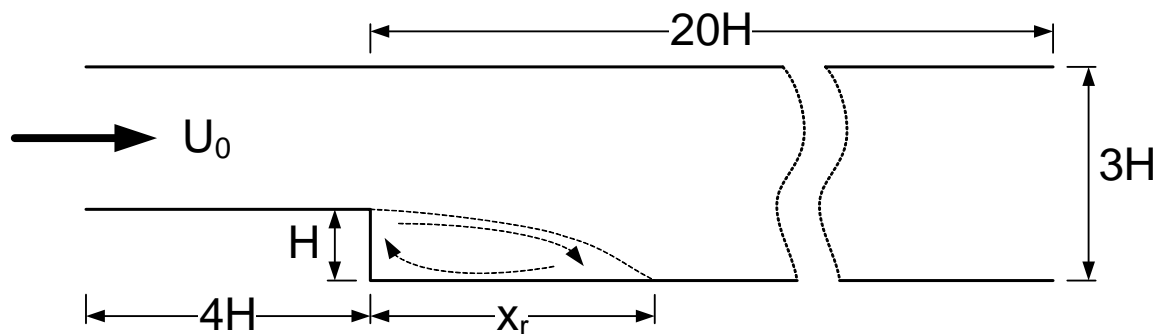


Figure 1: Schematic of the backward facing step domain.

The domain for this tutorial can be seen in Figure 1. A step of height  $H$  lies 4 step heights downstream of an inlet at which the velocity magnitude is  $U_0$  and a turbulence intensity of 5%. At the outlet, which lies  $20H$  downstream of the step, the relative pressure is 0 Pa. The walls of the step and the upper boundary are defined as smooth walls.

In the case used for this tutorial, the step height is 2 cm and the Reynolds number, based on the step height, is 44,000. This problem is 2D in design and nature, but CFX requires that a 3D grid be used. To keep the number of cells to a minimum, a grid just one cell deep is provided, Figure 2.

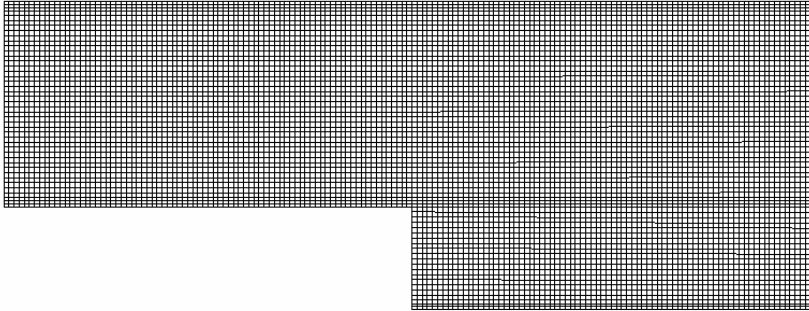


Figure 2: Close-ups of the three grids for the backward facing step: (a) coarse unstructured; (b) refined unstructured; (c) refined structured with boundary layers.

Use Kim et al. (1980) for comparison purposes. Consider in particular the velocity profiles at several locations downstream of the step and check the velocity reversal; use streamlines and vector plots to visualise the recirculation. Estimate what is the length of the recirculation as a function of the step height.

## 2.2. Flow in an Axisymmetric Sudden Expansion

Sudden expansions are akin to a double-backward facing step in several ways. You can use the description in Section 2.1 to understand the problem presented below.

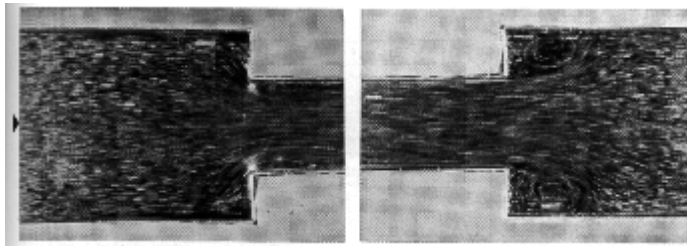


Figure 3: Flow past a Sudden Expansion (Source: [www.princeton.edu](http://www.princeton.edu))

In the case presented here by Aloui et al. (1999), as well as in Aloui's PhD thesis (1994), experimental measurements for velocity profiles and bed shear values are reported. You are asked to read Aloui's paper carefully and set up a single phase problem initially based on the geometry provided; then you will be expected to plot the velocity profiles at several locations downstream of the expansion, as well as bed shear and turbulence quantities. Additional figures from the PhD dissertation (unfortunately in French) will be handed to assist you and offer comparison material.

## 2.3. Airfoil Aerodynamics

Consider a classical airfoil problem as described below:

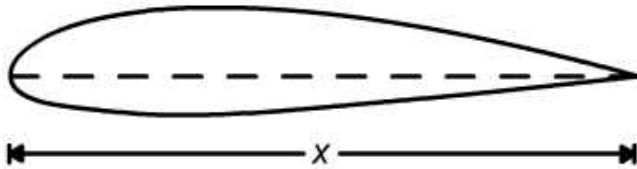


Figure 4: Airfoil profile

The main function of an airfoil is to create lift whilst minimising drag. When airfoils are studied several design parameters are considered:

- The lift is investigated as a function of the angle of attack
- The lift is also investigated as a function of drag (polar curve) to determine the operational characteristic of the airfoil.

For this particular project you should investigate the following cases:

- For an airflow flow of 50 m/s compute the lift and drag, with and without viscosity, for an angle of attack  $\alpha = 0^\circ, 5^\circ$  and  $10^\circ$ . What change do you see in the  $C_p$  distribution on the upper and lower surfaces as you increase the angle of attack? Which part of the airfoil surface contributes most to the increase in lift with increasing  $\alpha$ ? Compare the values between viscous and inviscid cases.
- Make a table of  $C_l$  and  $C_d$  values obtained for  $\alpha = 0^\circ, 5^\circ$ , and  $10^\circ$ . Plot  $C_l$  vs.  $\alpha$  for the three values of  $\alpha$ . Make a linear least squares fit of this data and obtain the slope. Compare your result to that obtained from inviscid, thin airfoil theory:

$$\frac{dC_l}{d\alpha} = \frac{2\pi^2}{180}$$

You will find general information about this topic in textbooks on the theory of flight and on the internet. This subject has no paper; comparisons will be made against the theory given in the subject.

#### 2.4. Venturi

You should have become familiar with the Venturi<sup>1</sup> phenomenon over the duration of your previous studies. Essentially the Venturi set up is a contracted pipe, i.e. a pipe which cross-sectional area is reduced, Fig. 5.

Considering Bernoulli:

$$\frac{V_1^2}{2g} + \frac{p_1}{\rho g} = \frac{V_2^2}{2g} + \frac{p_2}{\rho g}$$

With:

$$V_1 \times \frac{\pi D_1^2}{4} = V_2 \times \frac{\pi D_2^2}{4} \text{ (continuity)}$$

Therefore, since the pressure drop between 1 and 2 is equal to the hydrostatic pressure difference in the U-shaped barometer (which we can therefore measure), it is possible to compute the velocities and mass flow in the pipe. This forms the principle of the apparatus called the Venturimeter.

<sup>1</sup> [http://en.wikipedia.org/wiki/Venturi\\_effect](http://en.wikipedia.org/wiki/Venturi_effect)

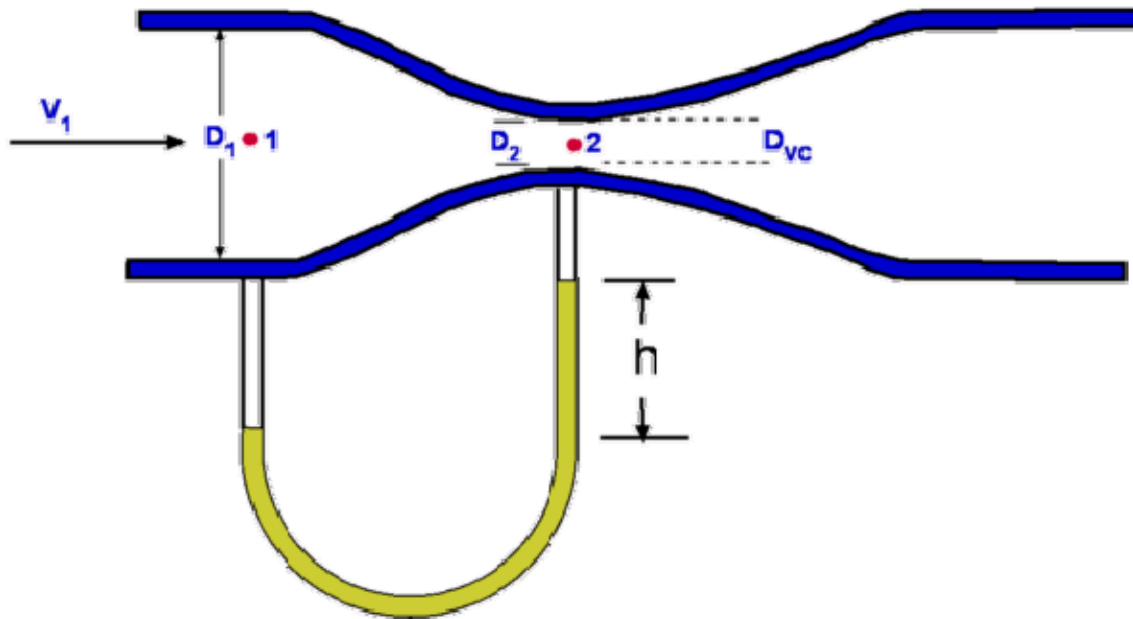


Figure 5: Venturi (Source: [www.aeromech.usyd.edu.au](http://www.aeromech.usyd.edu.au))

In the paper by Bates () a Venturimeter is used. Read the paper carefully. You are asked to investigate the flow in Bates' set-up and endeavour to replicate the results he shows on Figs. 4 and 5 as well as Table 1 in particular. Discuss the impact of turbulence and the Reynolds number for example.

### 2.5. Flow through an Orifice Plate

An orifice plate<sup>2</sup> helps measure flow through the differences in pressure from the upstream side to the downstream side of a partially obstructed pipe. The plate offers a precisely measured obstruction that narrows the pipe and forces the flowing substance to constrict. The pressure difference between the two sides is then measured. The greater the flow, the greater the difference in pressure as the substance maintains its constricted state for a longer distance, passing the downstream element. The principle is therefore very similar to that of the Venturimeter and can be explained using Bernoulli's equation once again.

Different kinds of orifice plates include concentric, eccentric, and segmental, each of which has different shapes and placements for measuring different processes. Orifice plates are in common use in many installations.

In the paper on orifice plates Bates (1981) studies a standard BS 1042 orifice plate and displays his measurements for various Reynolds number in terms of axial velocity and turbulence intensities. You are asked to familiarise yourself with his work and replicated numerically the results Bates displays in his paper, then assess and discuss the quality of your predictions in your report.

<sup>2</sup> [http://en.wikipedia.org/wiki/Orifice\\_plate](http://en.wikipedia.org/wiki/Orifice_plate)

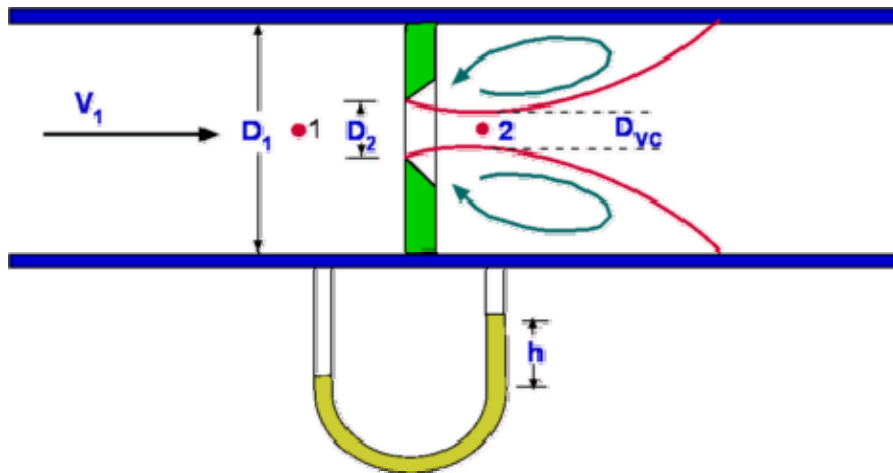


Figure 6: Orifice Plate (Source: [www.aeromech.usyd.edu.au](http://www.aeromech.usyd.edu.au))

### 2.6. Turbulent Flows in Pipe Bends I

Pipe bends affect the nature of the flow the pipe carries. The bulk of the flow tends to be carried toward the outside of the bend and secondary currents can develop. By secondary current it is meant that transversal flow develops in the pipe cross section, in rotation. These currents can affect mixing, heat transfer or erosion/deposition at the walls; they therefore need to be understood properly when designing an industrial device.

In their paper, Al-Rafai et al (1990) report on their numerical findings re the distribution of mass flow and the formation of secondary currents in two bends of different radius. You are asked to read their paper carefully, reproduce the cases presented in the paper and discuss how good your predictions are and why, based on your understanding of the problem and the knowledge acquired in class.

### 2.7. Turbulent Flows in Pipe Bends II

This subject is very similar to the above; please refer to 2.6. However it relies on the paper by Enayet et al.(????) and its set-up.

### 2.8. T-Junction I

T-Junctions are commonly found in various industrial applications, in water mains and pipelines, in engines, in many types of machinery used in the process industry and in the energy sector. In the paper by Yuki et al. (2001) the application is in the nuclear sector where the mixing that is observed at T-junctions is known to have led to early fatigue of the connection and leakages.

In this project you will be asked to investigate fluid mixing in the geometry provided to you and aim to replicate the main results displayed in Yuki's paper, in particular those pertaining to mixing, and then discuss these in light of your newly acquired knowledge.

### 2.9. T-Junction II

As highlighted here above, T-junctions are used to split the flow and are known to generate mixing and recirculation. Popp and Sallet (1983) report on the mass flow distribution at a T-junction and provide detailed velocity profiles which you

are asked to reproduce. The recirculation length is also investigated as a function of the mass flow distribution in the pipes, which you should also touch on. You will then be expected to discuss your predictions in light of these data and the material presented in class.

---

### Zig-Zag Classifier

2.10.

Zig-Zag classifiers are used to separate bulk solid material (transported by the fluid) from the carrier gas, Fig. 7. How this is achieved is in the content of the paper by Gillandt et al. () which you are asked to consider carefully.



Figure 7: Zig-Zag Classifier (Source: [www.hosokawamicron.com](http://www.hosokawamicron.com))

You are asked to model the bulk gas flow in the zig-zag geometry provided to you in a first instance and then to study the transport of solid particles. You will consider and discuss your results in light of Gillandt's paper.

---

### 3. References

F. Aloui, L. Doubriez, J. Legrand and M. Souhar, (1999), "Bubbly Flow in an Axisymmetric Sudden Expansion: Pressure Drop, Void Fraction, Wall Shear Stress, Bubble Velocities and Sizes", *Exp. Thermal and Fluid Science*, 19:118-130.

F. Aloui, (1994), "Etude des Ecoulements Monophasiques et Diphasiques dans Les Elargissements Brusques Axisymetrique et Bidimensionnel", PhD Thesis, Ecole nationale Supérieure d'Electricité et de Mécanique (in French).

W.N. Al-Rafai, Y.D. Tridimas and N.H. Woolley, (1990), "A Study of Turbulent Flows in Pipe Bends", *Proc Instn Mech Engrs*, 204:399-408.

C.J. Bates, (????), "Flowrate Measurements using a LDA/Computer Combination", FLOMEKO – Conference on Flow Measurement.

C.J. Bates, (1981), "LDA Measurements of the Flow through an Orifice Plate", *Flow: Its Measurement and Control in Science and Industry*, 2<sup>nd</sup> Int. Conf., St Louis, USA, March 1981.

M.M. Enayet, M.M. Gibson, A.M.K.P. Taylor and M. Yianneskis, (1982), "Laser-Doppler Measurements of Laminar and Turbulent Flow in a Pipe Bend", *Int. J. Heat and Fluid Flow*, 3(4):213-219.

I. Gillandt, C. Riehle and U. Fritsching, (????), "Gas-Solid Flow in a Zig-Zag Classifier – Comparison of Measurements and Simulations".

J. Kim, S.J. Kline, and J.P. Johnston, (1980), "Investigation of a reattaching turbulent shear layer: flow over a backward-facing step", *ASME J. Fluids Engng.*, 102:302–308.

M. Popp and D.W. Sallet, (1983), "Experimental Investigation of One- and Two-Phase Flow through a T-Junction", *Int. Conf. Physical Modelling of Multiphase Flow*, Coventry, UK, April 1983.

K. Yuki, K. Okuyama, S. Toda and T. Muramatsu, (2001), "Fundamental Study on Fluid Mixing Mechanism I a T-Junction Area", *Exp. Heat Transfer, Fluid Mechanics and Thermodynamics* by G.P. Celata, P. Di Marco, A. Goulas and A. Mariani (Ed.): 1573-1578.

v.2.1  
HPM/DMH/BJA,  
Nottingham  
Oct. 2006