Interaction of a confined jet with an obstacle creating vortex separation

Andrzej F Nowakowski and Franck C G A Nicolleau
Sheffield Fluid Mechanics Group
Department of Mechanical Engineering
University of Sheffield
Sheffield S1 3JD, UK
E-mail: {a.f.nowakowski, f.nicolleau}@sheffield.ac.uk

Abstract. One of many engineering applications of confined jets is to use them as fluidic flowmeters for flow measurements. The purpose of placing an obstacle (target) behind a jet discharging into a chamber could be interpreted as an additional measure to improve the oscillation sensitivity of a flow signal at the lower Reynolds limit. In this work the flow behaviour has been investigated over a wide range of parameters and Reynolds numbers. To this end the Delayed Detached Eddy Simulation (DDES) approach was applied. This numerical technique addresses the problem of premature activation of Large Eddy Simulation (LES) boundary layer by redefining the length scale used in Detached Eddy Simulation (DES). The simulation were first performed for several Reynolds numbers below 1,500. For this range of Reynolds numbers, the incoming jet did not start oscillating and experimental results were available for validation purposes. More simulations were performed for an extended range of Reynolds numbers up to 10,000. The results show the suitability of DES for the application of separating internal flows with relatively low Reynolds numbers in addition to its successful applications to external flows.

1. Introduction

The dynamical behaviour of the Karman vortex street in wakes behind various geometries has been a subject of numerous studies. The excellent review on this topic was published by Williamson (1996). The flow past a simple geometry produces complicated structures which are related to the detachment of the flow and the time dependent vortex shedding. This simple geometrical configuration is used in various practical applications including the flow in heat exchangers and vortex flowmeters. In the present contribution we are interested in the enclosed chamber setup with an obstacle creating vortex separation in unsteady flow conditions. The dynamics of coherent structures is observed around the obstacle impinging jet flow in a chamber. When a jet and the cylinder coexist the interaction of free shear flow phenomena happens. The flow in the interacting region is sometimes called the self-sustained oscillating flow due to the impingement effect. It was found by Rockwell & Naudascher (1979) that the flow impingement of the coherent structures moves to a upstream flow and further amplifies the evolving instabilities within the shear layer. These interactions have important practical applications such as fuel-air mixing combustors, jet ejectors, flowmeters or high lift devices.
2. Experiment

The original experimental work conducted in the context of designing a new flowmeter by Mazharoglu (1988) provides an excellent foundation for the present computational investigation. The principle of operation of oscillating jet devices are explained by means of flow visualisations and pressure distribution analyses. Their performance characteristics are reported in detail. The schematic view of the device consisting of a plain target and a pair of symmetrically placed knife-edges with indicated major geometrical parameters is presented in Fig. 1. In this experiment, an oil rig was built for Reynolds numbers lower than 800 and an air rig was used for Reynolds numbers greater than 800 up to 9500.

![Schematic view of the device](image)

**Figure 1.** Major geometrical performance variables.

The geometrical parameters $Y_k$, $X_k$, $X_t$, $Y_w$, and $W_t$ are normalized with respect to the nozzle width $d_n$. Vortices are generated on each side of the obstacle, which has sharp edges stabilizing the separation point of the boundary layer. The moment for flow oscillations to occur is related to the minimum Reynolds number. The inlet nozzle has a short contraction length. As a result, the viscous boundary layer development is small and higher discharge coefficients at low Reynolds number can be attained. Additionally, the knife-edges act as flow diverters influencing boundary layer separation and causing faster occurrence of instability and oscillations. The Strouhal number is a dependent variable of the Reynolds number. However, the experimental investigations show that the Strouhal number is normally constant across a wide range of the Reynolds numbers ($10^2 \sim 10^7$). This yields the formula for the velocity as a function of the oscillations frequency $f$

$$u_n = \frac{fd_n}{Sn}.$$  

In other words, if the density and viscosity of the fluid are known, the frequency measured at the obstacle can be used to represent the flow velocity. Consequently, the relationship for the volumetric flow rate $Q$ is

$$Q = \frac{fd_n A}{Sn}.$$  

The prototypes of flowmeters built using similar configuration have shown accurate results, especially at low Reynolds numbers (see Boucher & Mazharoglu, 1988; Priestman & Boucher, 2005, 2006).
3. Numerical Method

Separated flows have been often simulated using Unsteady Reynolds Averaged Navier-Stokes (URANS) approach. The disadvantage of these methods is the necessity of using several constant input data in model equations, which were earlier calibrated on the basis of the mean parameters for turbulent flows with relatively basic physics. The occurrence of eddies is not always the case when geometrically-specific structures are generated to produce separated flows in enclosed domains. The Detached Eddy Simulation (DES) method is a robust numerical technique for the simulation of unsteady flow of boundary layer separation, which has been used predominantly for external flows (see Spalart, 2009; Xia, 2005; Zheng et al., 2010). The accuracy of DES has been shown to be superior to steady or unsteady RANS methods, yet also allows a lower computational cost than Large Eddy Simulation (LES). DES decomposes a computational domain. It employs the unsteady RANS equations in near-wall regions and the filtered versions of the Navier–Stokes equations in the regions away from walls. In this approach LES is used in the regions where the eddy structures are detached and RANS in regions where the grid is not fine enough to support the LES requirements. In the present implementation of the RANS approach the Spalart-Allmaras (Spalart & Allmaras, 1994) model is used.

The standard Spalart-Allmaras model uses the distance to the closest wall to define the length scale, $d$. The DES model works by replacing this length scale with a new length scale $\tilde{d}$, which is defined as

$$\tilde{d} = \min(d, C_{des}\Delta),$$

where the grid spacing $\Delta$ is based on the largest spacing increment of the computational cell in either $x$, $y$ or $z$ direction. $C_{des}$ is an empirical constant and has the value of 0.65. This formulation can easily trigger the LES mode of DES at an inappropriate moment (i.e. within an attached boundary layer). This happens if a grid is too refined in the wall parallel directions. The problem has already been discussed and is explained in detail by Spalart et al. (2006). As a result a new form of DES, Delayed Detached Eddy Simulation with the Spalart-Allmaras model (DDES-SA) is proposed.

$$\tilde{d} = d - f_d \max(0, d - C_{des}\Delta)$$

$$f_d = 1 - \tanh(|8r_d|^3)$$

$$r_d = \frac{\nu_t + \nu}{\sqrt{U_{i,j}U_{i,j}\kappa^2d^2}}$$

In DDES-SA equation 3 is replaced by equation 4, where $d$ is the distance to the wall, $U_{i,j}$ are the velocity gradients, $\nu$ is the molecular viscosity, $\nu_t$ is the kinematic eddy viscosity and $\kappa$ is the Karman constant. $\tilde{d}$ does not only depend on the grid but also depends on the eddy viscosity field. This makes the method less susceptible to mesh induced error.

4. Computational Results

All simulations were performed using available implementation of DES method in a commercial computational fluid dynamics package FLUENT. The Dirichlet boundary conditions for the velocity field vector were defined at the inlet and as a no slip boundary condition at all solid walls. At the outlet boundary, a condition of zero-flux in the streamwise direction was applied. It assumes no velocity gradient in the normal direction to the outlet boundary and is referred to as the outflow boundary condition. This is more appropriate than modelling part of the
exit pipe, as the length of the downstream chamber means the exit conditions should not affect the obstacle region. The initial conditions were defined with a zero value velocity field in a computational domain. The multi-blocked structured grid was employed with around 530,000 - 650,000 nodes depending on Re number. During every time step, iterations were done until the residuals dropped below $10^{-4}$. Some benchmark configurations were simulated with the reference results obtained earlier either numerically with previously used DES approach (Nowakowski et al., 2010) or experimentally (Mazharoglu, 1988). The simulation were first performed for several Reynolds numbers below 1,500. For this range of Reynolds numbers, the incoming jet did not start oscillating and experimental results were available for validation purposes Fig. 2. More simulations were performed for an extended range of Reynolds numbers up to 10,000. A Reynolds number of 75 was the lowest value at which the benchmark case, used for validation against the experimental data, ($W_t = 1.5$, $X_t = 9$, $Y_k = 5.5$, $X_k = 8$, $Y_w = 8.5$, $a_r = 5$) was found to experience the onset of jet oscillations. The studies with different shapes of the obstacle were shown to be unable to produce jet oscillations at a Reynolds number lower than 75.

Figure 2. Dependence of Strouhal number on Reynolds number. Comparison of experimental and numerical results ($W_t = 2.5$, $X_t = 13$).

Figure 3 shows the contours of velocity magnitude on the $x - y$ plane midway through the computational domain at different time steps. This illustrates the mechanism of the jet oscillation from onset through to one complete oscillation. It also allows confirmation that the problem of jet reattachment downstream of the knife edges is not occurring within the model and consequently it is not necessary to include the stepping of the downstream chamber to increase simulation accuracy.

5. Conclusions

The interaction of a confined jet with an obstacle creating vortex separation was investigated. The DES simulations were conducted for this type of a flow configuration using existing experimental data for the sake of comparison. The different performance parameters were identified and their numerical investigation allowed a new, optimised configuration to be suggested. The validation of the DES model against experimental results showed that the DES approach could be successfully applied to internal separating flow problems with a good degree of accuracy.

The need to design the mesh for a specific case i.e. geometry and Reynolds number, and not just for each geometry tested, coupled with the long computational times associated with
Figure 3. Contours of velocity magnitude at flow times of 2.16s, 2.64s, 3.12s, 3.60s, 4.08s and 4.58s. The images show a simulation at a Reynolds number of 193. The contours show the velocity values from 0 m/s (blue) to 0.6 m/s (red).

DES may lead to the conclusion that the method is not suitable for a comprehensive parametric study. One would need a substantial time to be allocated to obtain a high degree of accuracy in both the creation of meshes, and the running CPU time of simulations. However, this work was focused on a small selection of relatively low Reynolds numbers and allowed the successful study of the effects of several parameters. The DDES-SA version of DES approach was shown to be highly successful in modelling this type of internal flows with Strouhal numbers in the computational simulations falling within about 10% of those measured experimentally.

Acknowledgments
The authors are very grateful to our former research students: Falk Hannemann, Arnaud Morillo and Mathieu Allen for assistance with the computational work. The supply of computation time on Iceberg, the Sheffield node of the White Rose Computing Grid, is greatly appreciated.

References
Boucher, R.F. & Mazharoglu, C. 1988 Low Reynolds number fluidic flowmeter. Journal of Physics E: Scientific Instruments 21, 977–989.
Mazharoglu, Cetin 1988 Low Reynolds Number Fluidic Flowmetering. PhD thesis, Mechanical Engineering Department, University of Sheffield.
Nowakowski, A.F., Nicolleau, F.C.G.A. & Salim, S.M.M. 2010 Computational study of a target fluidic flowmeter. In ASME 2010 10th Biennial Conference on Engineering Systems Design and Analysis, ESDA2010, , vol. 3, pp. 361–367.
Priestman, G.H. & Boucher, R.F. 2005 The biased laminar by-pass fluidic flowmeter. Journal of Fluids Engineering, Transactions of the ASME 127 (6), 1199–1204.
Priestman, G.H. & Boucher, R.F. 2006 Smart fluidic meters for simultaneous measurement of fluid flowrate, Reynolds number, density and viscosity. *Journal of Chemical Engineering of Japan* **39** (4), 383–393.

Rockwell, D & Naudascher, E 1979 Self-sustained oscillations of impinging free shear layers. *Annual Review of Fluid Mechanics* **11** (1), 67–94.

Spalart, P.R. & Allmaras, S.R. 1994 One-equation turbulence model for aerodynamic flows. *Recherche Aerospatiale* **1**, 5–21.

Spalart, P., Deck, S., Shur, M., Squires, K., Strelets, M. & Travin, A. 2006 A new version of detached-eddy simulation, resistant to ambiguous grid densities. *Theoretical and Computational Fluid Dynamics* **20**, 181–195.

Spalart, P. R. 2009 Detached-Eddy Simulation. *Annual Review of Fluid Mechanics* **41**, 181–202.

Williamson, C H K 1996 Vortex dynamics in the cylinder wake. *Annual Review of Fluid Mechanics* **28** (1), 477–539.

Xia, H. 2005 Dynamic Grid Detached-Eddy Simulation for Synthetic Jet Flows. PhD Thesis, University of Sheffield, Sheffield, UK.

Zheng, H.W., Nicolleau, F.C.G.A. & Qin, N. 2010 Assessment of DES on the flow after a snow-flake orifice. *Notes on Numerical Fluid Mechanics* **111**, 157–165.