Numerical simulation of confined laminar flow over a backward-facing step using a novel viscous-splitting vortex algorithm

R.W. Barber¹, A. Fonty²
¹Centre for Microfluidics, Department of Computational Science and Engineering, Daresbury Laboratory, U.K.
²Lower Base Engine Design Service, Renault, France

Abstract

A novel two-dimensional discrete vortex model is described for predicting low Reynolds number laminar flow behind a backward-facing step. The numerical model employs an ‘operator splitting’ technique to solve the vorticity-transport equation governing the evolution of vorticity. A semi-Lagrangian scheme is used to update the positions of the vortices during the advective stage of the computation whilst an analytical diffusion algorithm employing Oseen vortices is implemented during the diffusive time step. Redistributing the vorticity analytically instead of using the more traditional random-walk method enables the model to simulate steady flows directly and avoids the need to filter the numerical results to remove the oscillations created by the random-walk algorithm. Predictions from the model are compared against experimental velocity profiles of confined laminar flow behind a backward-facing step at Reynolds numbers of 73 and 229. In addition, computed reattachment lengths are compared against alternative numerical predictions. The results demonstrate that the analytical vortex diffusion scheme provides an effective alternative to random-walk procedures.

1 Introduction

Discrete vortex models offer a powerful computational technique for the simulation of viscous two-dimensional flows particularly in cases involving flow
separation. Early discrete vortex models often neglected vorticity diffusion and consisted of pure Lagrangian numerical schemes (Rosenhead [1]; Laird [2]; Clements & Maull [3]). However, Chorin [4] proposed an operator splitting method whereby the advective and diffusive processes could be treated using separate numerical schemes. Chorin's technique of simulating vorticity diffusion using a random-walk approach has subsequently been adopted in many studies including Stansby & Dixon [5], Sarpkaya [6] and Lewis [7].

The present paper describes an alternative method of simulating vorticity diffusion using an exact analytical formulation which allows the discrete vortices to diffuse smoothly in space and time. Following Benson et al. [8], each vortex is redistributed onto new independently moving point vortices at the end of the diffusion time step thereby removing the numerical instabilities which are sometimes associated with the simulation of vortex sheets. In addition, the present scheme is ideally suited to low Reynolds number steady flows since the numerical solution is free from the artificial fluid oscillations created by the random-walk procedure.

2 Governing hydrodynamic equations

The vorticity-transport equation governing the flow of an incompressible Newtonian fluid can be derived from the Navier-Stokes equations in the form:

$$\frac{D\omega}{Dt} = \frac{\partial \omega}{\partial t} + \mathbf{u} \cdot \nabla \omega = \omega \cdot \nabla \mathbf{u} + \nu \nabla^2 \omega$$

(1)

where $D/Dt$ is the total derivative, $\mathbf{u} = (u, v)$ is the velocity vector (streamwise $x$ and transverse $y$ components respectively), $\omega$ is the vorticity, defined as the curl of the velocity vector ($\omega = \nabla \times \mathbf{u}$), $\nu$ is the kinematic viscosity and $t$ is the time. The $\omega \cdot \nabla \mathbf{u}$ term on the right-hand side of eqn. (1) accounts for vortex stretching and is only applicable to three-dimensional flows. Consequently, in a planar two-dimensional flow, the resulting vorticity-transport equation simplifies to

$$\frac{D\omega}{Dt} = \frac{\partial \omega}{\partial t} + \mathbf{u} \cdot \nabla \omega = \nu \nabla^2 \omega .$$

(2)

Furthermore the vorticity can be related to the stream function, $\psi$, through Poisson's equation:

$$\nabla^2 \psi = -\omega$$

(3)

whilst the stream function can be defined in terms of the Cartesian velocity components as follows:

$$u = \frac{\partial \psi}{\partial y} \quad \text{and} \quad v = -\frac{\partial \psi}{\partial x} .$$

(4)

In the present study, the governing hydrodynamic equations (2, 3 & 4) are solved using the Eulerian-Lagrangian (vortex-in-cell) method developed by Christiansen [9] and later used by Baker [10] and Stansby & Dixon [5].
3 Vorticity transport

The vorticity-transport equation is solved using the ‘operator splitting’ method proposed by Chorin [4]. Equation (2) is therefore divided into separate advective and diffusive portions:

\[ \frac{\partial \omega}{\partial t} + \mathbf{u} \cdot \nabla \omega = 0 \]  
\[ \frac{\partial \omega}{\partial t} = \nu \nabla^2 \omega \]  

and the two expressions are solved sequentially. The transport of vorticity due to pure advection is simulated using a semi-Lagrangian discretisation of eqn. (5) whilst the diffusion eqn. (6) is solved analytically using Oseen vortices.

3.1 Advective transport

A uniform rectangular grid representing the flow domain is established with mesh spacings of \( \Delta x \) and \( \Delta y \). The vorticity of the \( n \)th vortex located at \((x_n, y_n)\) is allocated to the four surrounding mesh nodes according to the area-weighting scheme illustrated in Figure 1:

\[ \omega(k) = \frac{A_k \Gamma_n}{A^2}, \quad k = 1, 2, 3, 4 \]  

where \( \Gamma_n \) is the circulation of the \( n \)th vortex and \( A \) is the area of the cell. The stream function values at the mesh points are then obtained by discretising Poisson’s equation (3) using second-order finite-differences:

\[ \frac{(\psi_{i+1,j} - 2 \psi_{i,j} + \psi_{i-1,j})}{\Delta x^2} + \frac{(\psi_{i,j+1} - 2 \psi_{i,j} + \psi_{i,j-1})}{\Delta y^2} = -\omega_{i,j} \]  

where \( i,j \) are the nodal indices. Eqn. (8) is solved using Gauss-Seidel iteration in conjunction with successive-over-relaxation (S.O.R.).

The velocities of the elemental vortices are determined by evaluating the velocity components at each mesh point using central-difference approximations of eqn. (4):

\[ u_{ij} = \frac{(\psi_{i+1,j} - \psi_{i,j-1})}{2\Delta y} \quad , \quad v_{ij} = -\frac{(\psi_{i+1,j} - \psi_{i,j-1})}{2\Delta x} \]  

and then interpolating bilinearly onto the required vortex position:

\[ u_n = \sum_{k=1}^{4} \frac{u(k)A_k}{A} \quad , \quad v_n = \sum_{k=1}^{4} \frac{v(k)A_k}{A}. \]  

The positions of the vortices are then moved forward in time using a first-order scheme:

\[ x_n(t + \Delta t) = x_n(t) + u_n(t) \Delta t \quad , \quad y_n(t + \Delta t) = y_n(t) + v_n(t) \Delta t \]
where $\Delta t$ is the advection time step. Equations (7-11) form a consistent set of interpolation functions in the sense that an elemental vortex will not move in its own velocity field (Sarpkaya [6]). Vortices are introduced along the solid boundaries of the flow domain so as to annihilate the tangential slip velocity along the walls, as described by Lewis [7]. In addition, vortices transported across a solid-perimeter wall by the advection algorithm are 'reflected' back inside the flow domain to preserve the total circulation.

![Area-weighting scheme for the apportionment of vorticity to the four surrounding mesh nodes.](image)

**Figure 1:** Area-weighting scheme for the apportionment of vorticity to the four surrounding mesh nodes.

### 3.2 Diffusive transport

Vorticity diffusion is implemented using an analytical formulation involving Oseen vortices as described by Benson et al. [8]. For a vortex of circulation $\Gamma$ located at $z_0$ in an unbounded fluid, Batchelor [11] demonstrates that the evolution of the vorticity field can be described by

$$\omega(z, t) = \frac{\Gamma}{4\pi \nu t} \exp \left[ -\frac{|z - z_0|^2}{4\nu t} \right].$$  \hspace{1cm} (12)

Moreover, the linearity of the diffusion equation implies that the vorticity distribution produced by the diffusion of $n$ elemental vortices of circulation $\Gamma_i$ located at $z_i$ ($i=1,2,\ldots,n$) is simply given by the principle of superposition:

$$\omega(z, t) = \sum_{i=1}^{n} \frac{\Gamma_i}{4\pi \nu t} \exp \left[ -\frac{|z - z_i|^2}{4\nu t} \right].$$  \hspace{1cm} (13)
Assuming a diffusion time step of $\Delta t_d$ yields the updated vorticity field:

$$\omega(z, \Delta t_d) = \sum_{i=1}^{n} \frac{\Gamma_i}{4 \pi \nu \Delta t_d} \exp \left[ -\frac{|z - z_i|^2}{4 \nu \Delta t_d} \right].$$  \hspace{1cm} (14)

The computational efficiency of redistributing every vortex in the flow field onto new zero-aged vortices at the mesh nodes can be improved by neglecting the vorticity contribution outside a user defined core region. A convenient cut-off point containing 99.34% of the original circulation (Benson et al. [8]) is to select a core defined by $r \leq r_m$ where $r_m$ is twice the radius of maximum velocity:

$$r_m = 4.4836 \sqrt{\nu \Delta t_d}. \hspace{1cm} (15)$$

To redistribute the circulation of an individual vortex, P, it is possible to avoid the unnecessary expense of testing its proximity to all mesh points in the flow domain by defining the local significant zone of influence to be

$$I_P - K_1 \leq i \leq I_P + K_1 \hspace{1cm} \text{and} \hspace{1cm} J_P - K_2 \leq j \leq J_P + K_2 \hspace{1cm} \text{(16)}$$

where $I_P$ and $J_P$ are the nodal co-ordinates of the closest mesh point to P, and $K_1$ and $K_2$ are the smallest integers such that:

$$\left(K_1 - \frac{1}{2}\right) \Delta x > r_m \hspace{1cm} \text{and} \hspace{1cm} \left(K_2 - \frac{1}{2}\right) \Delta y > r_m \hspace{1cm} \text{(17)}$$

Benson et al. [8] found that the diffusion algorithm produces an accurate solution provided $r_m/\Delta x \geq 2$ and $r_m/\Delta y \geq 2$. The diffusion time step is therefore chosen so as to allow the vortex influence to spread over a radius of at least two cell lengths per time step. Since the required diffusion time increment is invariably larger than that necessary for accurate simulation of advection, Benson et al. suggested that the diffusion time step should be defined as a multiple of the advection time step i.e. $\Delta t_d = m \Delta t$ where $m$ is a suitable integer.

The analytical diffusion algorithm has to be modified in the vicinity of the no-slip boundaries to ensure there is no net flux of vorticity across the solid walls. This is accomplished by representing diffusion near a solid boundary by additional vortex images of the same sign located outside the computational domain:

$$\omega(z, \Delta t_d) = \frac{\Gamma}{4 \pi \nu \Delta t_d} \exp \left[ -\frac{|z - z_0|^2}{4 \nu \Delta t_d} \right] + \frac{\Gamma}{4 \pi \nu \Delta t_d} \exp \left[ -\frac{|z - \overline{z}_0|^2}{4 \nu \Delta t_d} \right] \hspace{1cm} \text{(18)}$$

where $z_0$ is the position of the real vortex and $\overline{z}_0$ is the location of the image vortex. It should be noted that only vortices closer to a boundary than the core radius ($r_m$) need to be considered in eqn. (18).

4 Model validation

The backward-facing step is one of the most fundamental geometries causing flow separation and has been extensively investigated in both the laboratory (Denham &
Computational Methods and Experimental Measures

Patrick [12]; Armaly et al. [13]) and as a standard 'bench-mark' test for numerical simulations (Ghoniem & Gagnon [14]; Guj & Stella [15]; Barton [16]). The present study first considers the flow geometry originally proposed by Denham & Patrick [12] who described an experimental study of laminar flow over a confined backward-facing step placed in a two-dimensional channel containing water. Denham & Patrick used a laser Doppler anemometer to obtain detailed velocity measurements of the flow expansion at various Reynolds numbers in the laminar regime. All measurements were of the streamwise (x-direction) velocity component, \( u \), and were made in the central plane of the duct to ensure the observed flow conditions were as close to two-dimensional as possible. Velocity profiles were measured at several transverse sections upstream and downstream of the step, at four different Reynolds numbers (\( Re=73, 125, 191 \) and 229). The Reynolds number, \( Re \), was defined by Denham & Patrick [12] as:

\[
Re = \frac{U_i h}{\nu}
\]  

(19)

where \( U_i \) is the average inlet velocity, \( h \) is the step height and \( \nu \) is the kinematic viscosity of the fluid.

The hydrodynamic model was used to simulate the flow regime investigated by Denham & Patrick. The flow domain consisted of a backward-facing step in a rectangular channel of length 420mm, inlet height 30mm and step height, \( h=15\text{mm} \), giving a 2:3 expansion ratio. A 141x31 node finite-difference mesh was employed in the Eulerian calculation, giving cell sizes of \( \Delta x = 3\text{mm} \) and \( \Delta y = 1.5\text{mm} \). Following Denham & Patrick, the kinematic viscosity of water was taken to be \( 9.97\times10^{-7}\text{m}^2\text{s}^{-1} \). The diffusion time step was specified as 2.0s whilst the advection time step was set to 0.2s.

![Figure 2: Comparison of non-dimensionalised streamwise velocity profiles for \( Re=73 \).](image-url)
Figures 2 and 3 compare predicted velocity profiles across the width of the channel against the experimental results published by Denham & Patrick [12] for Reynolds numbers of 73 and 229. The non-dimensionalised velocity, $u/U_i$, and the non-dimensionalised distance downstream of the expansion, $x/h$, are used in order to allow a direct comparison of results. For both Reynolds numbers the agreement between the predicted and observed velocities is very good, demonstrating the feasibility of the analytical vortex diffusion scheme. Additional simulations using a hydrodynamic mesh of twice the resolution (281x61 nodes) yielded almost identical velocity profiles.

The second validation considers the backward-facing step originally studied by Armaly et al. [13] and later investigated by Guj & Stella [15] and Barton [16]. The flow geometry is illustrated schematically in Figure 4 and consists of a confined backward-facing step with an expansion ratio of 1:2. As an aside, it should be noted that Armaly et al.'s original experimental investigation employed a slightly different expansion ratio of 1:1.94. At low Reynolds numbers the flow separates at the sharp corner and then reattaches itself to the lower boundary further downstream forming a single primary recirculating eddy. The reattachment length increases almost linearly with Reynolds number, the slight non-linear trend being attributed to viscous drag along the upper boundary (Barton [16]). However, at higher Reynolds numbers ($Re > 400$), the adverse pressure gradient along the upper boundary is strong enough to promote a secondary recirculation zone attached to the upper wall which causes a reduction in the growth of the lower eddy. The determination of the separation and reattachment locations offers a severe bench-mark test for any hydrodynamic model because of the highly non-linear flow kinematics in the lee of the step.
For consistency with Armaly et al. [13], the Reynolds number, $Re$, in the present series of tests is redefined as:

$$Re = \frac{U_i 2h}{\nu}$$

where $U_i$ is the average inlet velocity, $h$ is the height of the inlet channel and $\nu$ is the kinematic viscosity of the fluid. The use of $2h$ as the characteristic length scale can be attributed to the fact that Armaly et al. [13] based the Reynolds number on the hydraulic diameter of the inlet channel.

A fully-developed parabolic $u$-velocity profile was prescribed at the inlet of the hydrodynamic model whilst the outflow boundary, consisting of zero-normal velocity derivatives, was located a distance of $30h$ downstream of the step. The location of the outflow boundary was chosen sufficiently far downstream so as not to affect the position of the upstream recirculation zones. A high resolution $301 \times 101$ node finite-difference mesh was employed in the Eulerian calculation, giving cell sizes of $Ax = 0.1h$ and $Ay = 0.02h$. Following Benson et al. [8], the diffusion time step was chosen to allow the diffusing vortices to spread over a core radius of at least two cell lengths per time step.

Computed non-dimensionalised reattachment and separation lengths are presented in Figure 5 together with the experimental data of Armaly et al. [13]. The present numerical simulations agree satisfactorily with the experimental results up to $Re = 400$. Armaly et al. observed three-dimensional flow effects at higher Reynolds numbers which probably explains the deviations above $Re = 400$. Figure 5 also includes the numerical predictions obtained by Barton [16] who employed a two-dimensional finite-volume approach. The excellent agreement between the two numerical schemes demonstrates the feasibility of the analytical vortex diffusion algorithm.

![Figure 4: Confined backward-facing step and recirculation zones for a high Reynolds number flow (after Barton [16]).](image-url)
Figure 5: Comparison of reattachment and separation lengths.

5 Conclusions

A discrete vortex model has been described for simulating two-dimensional laminar recirculating flow patterns behind a backward-facing step. The model uses a semi-Lagrangian method to track the evolution of the vortex positions due to pure advection whilst an analytical diffusion scheme employing Oseen vortices is implemented during the diffusive calculation. The analytical diffusion algorithm is free from the artificial fluid oscillations created by random-walk procedures and is therefore ideally suited to low Reynolds number steady flows. Predictions from the numerical model have been compared against experimental data of laminar flow past a backward-facing step. In addition, computed reattachment and separation lengths have been compared against alternative numerical predictions. The results give realistic velocity profiles and reattachment lengths downstream of the step demonstrating the feasibility of the proposed discrete vortex scheme.
References


