Flow through a cylindrical pipe with a periodic array of fractal orifices

This content has been downloaded from IOPscience. Please scroll down to see the full text.

2013 Fluid Dyn. Res. 45 061405

(http://iopscience.iop.org/1873-7005/45/6/061405)

View the table of contents for this issue, or go to the journal homepage for more

Download details:

IP Address: 130.89.97.218
This content was downloaded on 08/01/2014 at 22:10

Please note that terms and conditions apply.
Flow through a cylindrical pipe with a periodic array of fractal orifices

P A J van Melick\textsuperscript{1} and B J Geurts\textsuperscript{1,2}

\textsuperscript{1} Fluid Dynamics Laboratory, Faculty of Applied Physics, Eindhoven University of Technology, NL-5300 MB Eindhoven, The Netherlands
\textsuperscript{2} Multiscale Modeling and Simulation, Faculty of Electrical Engineering, Mathematics, Computer Science, University of Twente, PO Box 217, 7500-AE Enschede, The Netherlands

E-mail: b.j.geurts@utwente.nl

Received 13 August 2012, in final form 28 February 2013
Published 14 August 2013
Online at stacks.iop.org/FDR/45/061405

Communicated by K Suga and S Laizet

Abstract
We apply direct numerical simulation (DNS) of the incompressible Navier–Stokes equations to predict flow through a cylindrical pipe in which a periodic array of orifice plates with a fractal perimeter is mounted. The flow is simulated using a volume penalization immersed boundary method with which the geometric complexity of the orifice plate is represented. Adding a periodic array of orifice plates to a cylindrical pipe is shown to increase the mixing efficiency of the flow in the laminar regime. The average stretching rate is shown to increase by a factor of up to 5, comparing pipe flow without orifice plates to flow passing through an orifice plate derived from the Koch snowflake fractal. The dispersion rate is shown to increase by a factor of up to 4. In laminar flow, the viscous forces are most important close to the walls, causing orifice geometries with the largest perimeter to exhibit the largest axial velocities near the centerline of the pipe and the largest pressure drop to maintain the prescribed volumetric flow rate. The immersed boundary method is also applied to turbulent flow through a 'fractal-orifices pipe' at $Re = 4300$. It is shown that the pressure drop that is required to maintain the specified volumetric flow rate decreases by about 15\% on comparing orifice plates with a circular opening to orifice plates with more complex shapes that contain several corners such as triangles, squares, stars and the Koch snowflake.

(Some figures may appear in colour only in the online journal)

1. Introduction

Flow through a cylindrical pipe has been studied intensively over many decades. It is one of the archetypal flow geometries in the tradition of fluid mechanics research, dating back to
early experiments by Reynolds (1883). Interest in pipe flow derives often from the many practical applications that require transport of liquids, gases and granular matter (Moody 1944). Pipe flow is also of fundamental interest in terms of understanding and optimizing laminar and turbulent mixing (Sreenivasan et al 1989). We investigate flow through a pipe to which a ‘flange’, also called an ‘orifice plate’, is added (Nicolleau et al 2010, 2011). The orifice plate locally reduces the area through which the flow can pass and finds a well-known application as a flowmeter in which the flange is used to create a pressure drop from which the flow rate is deduced. We focus on the influence that the shape of the perimeter of the orifice has on mixing and transport properties. We consider both laminar and turbulent flow conditions and establish that (i) increased laminar mixing and dispersion arise from flow through orifices with complex perimeters, and (ii) turbulent flow through an orifice with several corners requires a substantially lower pressure drop to maintain a specified flow rate compared to a circular orifice.

In this paper, we investigate flow through a cylindrical pipe to which an orifice plate is added. The flow is determined numerically, using direct numerical simulation (DNS) to solve the incompressible Navier–Stokes equations. One of the first high-precision simulations in this direction was reported in Moin and Kim (1982) and contained a study of turbulent channel flow. DNS dealing with turbulent pipe flow was discussed in Eggels et al (1994) where it was also compared with experiments. The simulations were extended later to higher resolutions in Fukagata and Kasagi (2002) using an energy-conserving discretization, and to higher Reynolds numbers in Wu and Moin (2008). In order to represent the orifice plate, we adopt a volume penalizing immersed boundary (IB) method as described e.g. in Lopez Penha et al (2011) and Mikhal and Geurts (2011, 2012). In this IB method, the total flow domain is represented as if it were cut from a rectangular solid domain enclosing it. A comprehensive discussion of IB methods may be found in Mittal and Iaccarino (2005). In a volume penalizing IB method the total domain, i.e. solid plus fluid parts, is represented on a Cartesian grid using a so-called ‘masking function’ which takes the value ‘1’ in a point in the solid and ‘0’ in points in the fluid. This approach allows us to approximate highly complex flow domains such as would emerge from adding an orifice plate whose perimeter is derived from a fractal curve. To the best of our knowledge, this is the first simulation study in which DNS-IB is adopted to laminar and turbulent flows in domains derived from truncating a fractal object. Such complexity of the flow domain would not be practical for more conventional body-fitted approaches in which the construction of a computational mesh, suitable for high-precision simulations, is a major stumbling block.

In recent years, flow past multiscale objects has received considerable attention in relation to the problem of generating broadband turbulence in a controlled way (Kuczaj and Geurts 2007). Turbulent flows have been generated experimentally mainly by producing flow through a regular grid of solid bars (Comte-Bellot and Corrsin 1966). This approach aims at introducing a wide range of dynamic scales into a fluid flow in order to investigate turbulence and its mixing properties. Even to date, much of the motivation behind a particular design of a grid with which to generate turbulence is based on classical geometry (Comte-Bellot and Corrsin 1971). Recently, the problem of introducing many dynamic scales into a flow has been addressed in a different manner by incorporating an active grid which perturbs the flow by moving its many vanes in a prescribed manner (Kuczaj et al 2006, Cekli and Van de Water 2010). Apart from this technically somewhat advanced approach, a new method for introducing many scales into a flow, which is technically very simple, has gained popularity. In the latter approach, flow is passed along geometrically complex, multiscale shapes, often following a fractal pattern (Hurst and Vassilicos 2007, Seoud and Vassilicos 2007). The flows generated in this ‘fractal’ way have been shown to contain a wide range of scales
and have even been reported to display scalings that differ from the classical Kolmogorov cascade (Mazellier and Vassilicos 2010). One advantage of using fractal-generated turbulence with respect to free turbulence is the increased level of control over the resulting turbulence. Using fractal-generated flow results in energy savings, increased reaction homogeneity and smaller equipment (Kearney 1999). In this category of controlled turbulence generation, we may also place flow through a cylindrical pipe with a mounted orifice plate that contains a hole for fluid to pass through with a possibly very complex perimeter. This model system was first considered experimentally in Nicolleau et al (2010). A remarkable reduction in the pressure drop needed to maintain a specified flow rate was reported in case orifices with a non-circular perimeter were adopted. This model system is investigated numerically using DNS-IB in this paper, confirming this experimentally observed property also at moderate Reynolds numbers.

Apart from a role in generating controlled broadband turbulence at high Reynolds numbers, fractal objects may have a role to play in enhancing mixing efficiency (Geurts 2001) under laminar flow conditions. In the case of laminar flow, creating complex stream patterns is of utmost importance in order to achieve rapid mixing in a compact volume (Chien et al. 1986). By passing flow through an orifice with a complex perimeter, one may expect that the streamlines also exhibit an increased local variation which may influence stretching and folding of fluid parcels and enhance dispersion of particles contained in the flow (Ranz 1979). To reduce the mixing length in microfluidics, it is important to introduce transverse flow components on top of the Poiseuille-type flow in the micro-tube. These transverse components will stretch and fold volumes of fluid, which will reduce the mixing length by decreasing the average distance over which diffusion must act to homogenize unmixed volumes (Stroock et al. 2002). We will study laminar mixing characterized by Lagrangian properties of fluid loops evolving in flow generated through complex orifice plates mounted in a pipe. For that purpose, we place a large number of test-particles on initially circular contours in the flow and follow the precise motion of each of these particles, thereby tracking the contours at later times. We determine the growth of the length of the fluid loops and the associated dispersion and show that the shape of the orifice perimeter has a strong influence on laminar mixing efficiency.

The organization of this paper is as follows. In section 2, we discuss the governing equations and the numerical method to simulate flow through complex orifice plates. We also introduce the orifices included in this study. Section 3 is devoted to the simulation of laminar flow through orifice plates and the enhanced mixing obtained by increasing the complexity of the orifice perimeter. Results pertaining to the relation between the shape of the orifice perimeter and the pressure drop required to maintain a prescribed flow rate for turbulent flow are discussed in section 4. Concluding remarks are collected in section 5.

2. Immerged boundary method for pipe flow

In this section, we first introduce the governing equations and describe the volume penalization IB method with which the flow through complex domains is simulated. Subsequently, we discuss the inclusion of an orifice and present the key shapes of the orifice plates that will be investigated in the context of pipe flow in this paper.

Incompressible flow is governed by the principles of conservation of mass and momentum. These can be expressed in terms of the Navier–Stokes equations (Batchelor 1967, Acheson 1990, Pope 2000). In non-dimensional form these equations are given by

\[ \nabla \cdot \mathbf{u} = 0, \]
Figure 1. Schematic representation of a solid body $\Omega_b$ with boundary $\partial \Omega_b$ embedded in a fluid domain $\Omega_f$. If $x \in \Omega_b$ then the masking function $H = 1$ while $H = 0$ if $x \in \Omega_f$. On a Cartesian grid, cells whose center is solid are shown in gray and all the other grid cells are white, approximating the fluid domain.

\begin{equation}
\partial_t u + (u \cdot \nabla)u - \frac{1}{Re} \nabla \cdot \nabla u + \nabla p = f,
\end{equation}

where $u \equiv (u, v, w)$ denotes the velocity vector and $p \equiv P/\rho$ where $P$ is the pressure and $\rho$ the mass density. In these equations, $\partial_t$ denotes the partial derivative with respect to time $t$ and $\nabla$ the derivatives with respect to the spatial coordinates $x \equiv (x, y, z)$. Finally, $Re$ is the Reynolds number and $f$ a forcing term that is used to represent the impenetrability of complex shaped solid walls, i.e. the no-slip condition at fluid–solid interfaces. The equations were made dimensionless using the radius $R$ of the pipe as the reference length and the bulk velocity $U_b = Q/(\pi R^2)$ in the pipe as the reference velocity, expressed in terms of the volumetric flow rate $Q$. The kinematic viscosity was chosen accordingly, to complete the definition of $Re$ (Geurts 2003).

In the volume penalization IB method (Angot et al 1999, Kevlahan and Ghidaglia 2001, Mittal and Iaccarino 2005, Keetels et al 2007, Sarthou et al 2008) the forcing field $f$ is defined as

\begin{equation}
f = -\frac{1}{\varepsilon} H(x)u(x, t),
\end{equation}

where $\varepsilon \ll 1$ is a control parameter and $H(x)$ denotes the so-called masking function, a binary function used to distinguish fluid parts from solid parts of the domain. In fluid regions, we set $H = 0$ and solve the Navier–Stokes system. In solid regions, we set $H = 1$. In figure 1, we sketch the construction of the masking function for a solid body $\Omega_b$ immersed in a Cartesian grid. When $H = 1$ on a neighborhood of a point $x$, the governing equations reduce to $\partial_t u \approx -u/\varepsilon$ on this neighborhood, if $u$ is sufficiently differentiable and initially $|u| \gg \varepsilon$. Hence, any non-zero $u$ is exponentially sent back to 0 on a time scale $\varepsilon$. If $|u| \ll \varepsilon$ the forcing
is not dominant in the solid, but control over $|u|$ is already obtained, i.e. $|u|$ takes on negligible values in the solid. We take $\varepsilon = 10^{-10}$ as proposed in Lopez Penha et al (2011) and Mikhal and Geurts (2011, 2012). Such low values of the control parameter $\varepsilon$ imply that the forcing term effectively yields a Brinkman equation in which ‘porous’ regions are virtually impenetrable, i.e. solid material. A number of applications and further discussion of the IB method are provided in Iaccarino and Verzicco (2003) and Mittal and Iaccarino (2005).

The orifice plate and the cylindrical pipe are generated by specifying the masking function for this combined geometry. In figure 2, we sketch the rectangular block $L_x \times L_y \times L_z$ that encloses the total domain, together with the flow domain composed of the cylinder and the mounted orifice plate halfway across the domain. The pipe has radius $R$, which is taken as the reference length for the non-dimensional formulation. We adopt a uniform Cartesian grid for the computational model and use the rule that an individual grid cell can only be solid or fluid. The decision to assign the value $H = 0$ or 1 to a grid cell is taken on the basis of the property ‘solid’ or ‘fluid’ that applies at the center $x_c$ of the grid cell. As an example for a cylindrical pipe, in dimensional form, if $|x_c| \leq R$ then the grid cell is taken as a fluid with $H = 0$ and $H = 1$ otherwise. This implies that the actual interface between the fluid and the solid is approximated to within one grid cell accuracy. Moreover, since a grid cell is either entirely solid or entirely fluid, the actual interface is approximated by a ‘staircase’. Next to assigning the masking function for the cylindrical pipe segment of the domain, we require the specification of the masking function for the orifice plate. We turn to this next.

We consider a segment of pipe with length $L_x = 10$ in units $R$, the radius of the pipe, and adopt periodic boundary conditions in the streamwise direction. In order to enclose the pipe by a rectangular block we use, in non-dimensional notation, $L_y = L_z = 2 + 2h$ where $h$ denotes the mesh size in the $y$- and $z$-directions. This ‘padding’ of the computational domain with two grid cells is required by the numerical method in order to represent the no-slip boundary condition at the cylinder wall. In all simulations, we impose a desired volumetric flow rate $Q$ through the pipe, which is maintained by adjusting the pressure drop that is imposed over the streamwise extent, following the decomposition of the pressure term $p(x, t) = a(t)x + p'(x, t)$ where $p'$ is the correction to the pressure relative to the linear pressure profile $a(t)x$ and $a(t)$ is adapted such that constant $Q$ is achieved. The pressure correction $p'$ is assumed to be periodic (Moin and Kim 1982).

The orifice plate is placed exactly in the middle of the domain. It is given a thickness $\delta = L_x/16$. This represents a rather thick orifice plate, almost a short ‘fractal tube’. The reason to select this value for the current study is that it represents a fairly localized constriction

![Figure 2. Definition of the computational domain for a circular pipe with radius $R$ and centerline along the $x$-axis, enclosed by a rectangular volume $L_x \times L_y \times L_z$, with a complex orifice mounted halfway.](image-url)
in the pipe geometry, while allowing us at the same time to incorporate several levels of
grid refinement when assessing the accuracy of numerical predictions. In fact, in this paper
we include spatial resolutions with at least 16 grid points in the streamwise direction. This
coarsest resolution covers the width of the orifice plate by just one grid cell, which is the
principal minimum that can be accepted. Comparison of the coarsest results with higher
resolutions indicates a level of convergence—such grid refinement is conveniently executed
starting from a fairly thick orifice plate. In future studies, it will be interesting to investigate
the dependence of the flow on the width of the plate. We adopt grids that have \( n_x \times n_y \times n_z \)
grid cells in the \( x \)-, \( y \)- and \( z \)-direction, respectively, where we use powers of two for the
number of cells, i.e. \( n_x = 2^k \) and similarly for \( n_y \) and \( n_z \). By increasing the resolution at fixed
thickness \( \delta \), one may investigate convergence of the numerical solution—we illustrate this
for turbulent flow in a clean pipe in section 4. As an example, for a resolution of \( n_x = 128 \)
the orifice plate is located at \( 4.7266 \leq x \leq 5.2374 \). To further specify the orifice plate, we
introduce the porosity \( \phi \) of the plate by \( \phi = A_o / (\pi R^2) \), where \( A_o \) denotes the area of the fluid
cells at a cross-section through the orifice. In the following, we use \( \phi = 0.4 \) to obtain masking
functions that represent a considerable narrowing of the pipe at the orifice plate. We include as
basic shapes a circle, square, triangle and five-pointed star, such that the value of the porosity
is approximated as closely as possible on a selected resolution \( n_y \times n_z \). In figure 3, we present
masking functions for these basic orifice plates at a resolution of 64 \times 64. We also incorporate
a fractal geometry, obtained from the Koch snowflake. The Koch snowflake is constructed
by repeatedly applying the Koch curve construction on each side of a straight segment of
the perimeter in the previous generation. The zeroth generation of the Koch snowflake is an
equilateral triangle as in figure 3(c). The first generation is the ‘star’ shape in figure 3(d). In
figure 4, we show the second and third generation of this fractal. For convenience, we will also
refer to these geometries as Koch-2 and Koch-3 in the following. We may clearly recognize
the intended shapes as well as the staircase character of the approximations. With increasing
resolution the correspondence between the masking function and the intended shape obviously
will improve. The numerical porosity deviates slightly from the desired imposed porosity of
\( \phi = 0.4 \). We determined the actual porosity of the staircase representations at a resolution of
64 \times 64 and observed a relative error in the porosity of less than 2%.

For the numerical simulation of the momentum equations (2), we employ symmetry-
preserving finite volume discretization and use central differencing of second-order accuracy,
implemented on a uniform, staggered grid, distinguishing pressure points in the middle of
each grid cell, and velocity nodes in the center of each of the faces of the grid cells. This
discretization explicitly maintains the skew-symmetry in the discrete equations. Since the
energy is preserved under the convective operator, the skew-symmetric discretization allows
us to obtain a stable solution on any grid. For proper capturing of the solenoidal property
equation (1) of the velocity field, we approximate the gradient operator by the transpose of
the numerical divergence operator and use a positive-definite discretization of the viscous

The contributions of the convective, viscous and pressure gradient fluxes are integrated in
time using a generalization of the explicit second-order accurate Adams–Bashforth method.
Care is taken to accurately represent the skew-symmetry also in the fractional step time-
integration. Full incorporation would require an implicit time stepping, which, however,
is computationally too demanding. Instead, time integration starts from a modification of the
leapfrog method (Saylor 1988) with linear inter/extrapolations of the required ‘off-
step’ velocities and an implicit treatment of the incompressibility constraint. Optimization
for the largest stability region of the resulting scheme yields a particular so-called ‘one-
leg’ time-integration method, with a mathematical structure that is akin to the well-known
Figure 3. Masking functions of an orifice with a circle (a), square (b), triangle (c) and star (d) geometry. For the generation of these masking functions, grids with 64 × 64 points were used in the y- and z-directions.

Adams–Bashforth scheme. More details can be found in Verstappen and Veldman (2003). The treatment of the linear forcing term, which was introduced to arrive at the volume penalization IB approach, was done with an implicit Euler scheme in order to maintain computational efficiency also in the case of very small values of the forcing parameter ε (Lopez Penha et al 2011, Mikhal and Geurts 2011, 2012).

3. Increased laminar mixing

In this section, we present simulation results for flow through a complex orifice under laminar flow conditions at \( Re = 1 \), using a spatial resolution of 128 × 64 × 64, which was found to be adequate for capturing the dominant features of the laminar flow. The velocity field is discussed in section 3.1. We illustrate flow structures in the vicinity of the orifice plate and
pay attention to the gradual evolution of the laminar pipe flow in the region between two orifice plates. Subsequently, in section 3.2, we introduce a measure for mixing efficiency based on the rate of spreading of coherent groups of test-particles flowing with the flow. The rate of this spreading process will be determined for all orifice shapes in order to quantify the enhancement of laminar mixing compared to flow through a circular orifice.

3.1. Laminar flow characteristics

We begin the discussion of the flow through a pipe with a mounted orifice plate by considering the flow structures that develop in the velocity field. In figure 5, we sketched the geometry of the pipe and indicated the streamwise locations at which 2D cuts through the solution will be shown. In figure the axial (streamwise) velocity component \( u \) is presented for the Koch-2 orifice plate. We notice that sufficiently far from the orifice plate the contours of \( u \) are concentric circles, similar to what would arise in the absence of an orifice plate. At the upstream edge of the orifice plates at \( x = 4.6875 \) we recognize a sharp image of the Koch-2 shape imprinted on the velocity contours with relatively high velocities in the many substructures of the Koch-2 snowflake. This image blurs slightly as the flow passes through the orifice until \( x = 5.3125 \). Beyond this downstream edge of the orifice plates, the flow is seen to lose the sharper details quickly and already at \( x = 5.8594 \) much of the finest details have decayed away and only the coarsest six-fold symmetry of the Koch-2 shape can be
Figure 6. Axial velocity profiles at $Re = 1$ in the case of flow through an orifice plate with the shape of a second generation Koch snowflake. Contours are shown at a number of characteristic locations: $x = 0.78125$ (a), 2.3438 (b), 4.6875 (c), 5 (d), 5.3125 (e), 5.6594 (f), 7.8125 (g) and 9.2919 (h). The axial velocity is scaled by the bulk velocity.
discerned. Further away from the orifice plates the flow is more similar to the concentric contours characteristic of simple pipe flow. We also studied the structure of the streamwise velocity for other orifice plates and observed largely similar behavior. The shape of the orifice is well visible in the direct vicinity of the plate and quite rapidly resembles a Poiseuille-type flow within a distance of about two to three times the radius of the pipe.

In figure 7, we show the radial velocity component \( u^{(r)} \) defined as

\[
u^{(r)} = \frac{y}{r} v + \frac{z}{r} w
\]

in the point \( x = (x, y, z) \) where \( r^2 = y^2 + z^2 \). The value of the radial velocity was found to change considerably in the pipe—hence each of the figures is shown with its own colormap. A positive radial velocity corresponds to an outward radial velocity, while a negative value denotes motion toward the center. Just before the orifice plate we observe quite high values of inward velocity with an amplitude of about the bulk velocity, as would be expected to allow passage of the flow through the more narrow opening in the orifice. Between the orifice plates we observe striking patterns of alternating regions of relatively high and relatively low radial velocity. The specific pattern that develops is directly related to the shape of the orifice plate; we observed different but equally striking radial velocity patterns associated with the other shapes. The alternating regions of high and low radial velocities disappear quite rapidly away from the orifice plate to return to values around zero when the gradual approach back to Poiseuille-type flow is more complete.

The specific shape of the orifice plate has a unique effect on the flow that develops in its vicinity. In figure 8, we compare the enstrophy \( \xi \) for all geometries studied. In particular, we plot

\[
\xi = \omega_x^2 + \omega_y^2 + \omega_z^2
\]

where the vorticity \( \omega = (\omega_x, \omega_y, \omega_z) = \nabla \times u \). Capturing the enstrophy with an IB method requires accurate capturing of the derivatives of the velocity field—for laminar flow the dominant structures could be well predicted with our method at a modest resolution of \( 64 \times 64 \). The structure of the enstrophy field clearly connects to the shape of the orifice plate that was used, as is clearly visible at the downstream edge of the plates. High values of the enstrophy are observed close to the perimeter of the orifice, while in the middle of the pipe \( \xi \) is rather small. Moreover, we notice that, seen from the center of the pipe, ‘convex’ sections along the perimeter show relatively low values of enstrophy and higher values are seen near smooth segments (circle, square, triangle) and near the exposed ‘concave’ sections (star, Koch-2). Apart from obvious differences in the structure of the enstrophy field, we observe that the amplitude of the enstrophy increases rapidly with increasing number of sharp corners in the perimeter of the orifice. Where maximal values of \( \xi \) around 100 are seen for flow without a flange, the introduction of a circular orifice plate at porosity 0.4 yields a ten-fold increase in the maximum. This maximum increases further from 1200 to 1600 when introducing a square or a triangular shape, and even to 2500 and 3000 for the star and Koch-2 orifice plates.

A more quantitative comparison of the influence of the shape of the orifice plate on the flow is presented in figure 9(a) in which the pressure drop \( \Delta p \) across the length of the pipe is shown. This pressure drop is required to maintain the imposed volumetric flow rate \( Q \), and specifies the power \( P \) required to keep the flow going, since \( P = Q \Delta p \). In our simulations, we impose a fixed flow rate and measure the pressure drop. The pressure drop in the steady laminar flow is seen to increase with increasing complexity of the perimeter of the orifice, similar to the increasing maximal enstrophy as observed in figure 8. We observe up to
Figure 7. Radial velocity profiles at $Re = 1$ in the case of flow through an orifice plate with the shape of a second generation Koch snowflake. Contours are shown at a number of characteristic locations: $x = 0.78125$ (a), 2.3438 (b), 4.6875 (c), 5 (d), 5.3125 (e), 5.8594 (f), 7.8125 (g) and 9.2919 (h). The radial velocity is scaled by the bulk velocity.
a 30% higher value for Koch-2 compared to the case with a circular orifice. The values obtained for the other shapes are between these limits. This ‘order’ in the value of the pressure drop is also seen in the steady pressure profiles in figure 9(b); the linear pressure dependence for the clean pipe is seen to change into a much larger pressure gradient near the orifice plate, next to regions of comparably constant pressure. The simulated flow does not represent passage through a single orifice plate. In fact, the length $L_x = 10$ of the domain is seen to be too short for a full recovery of the flow to Poiseuille flow, as would arise if the periodic length is sufficiently large. Consequently, the investigated flow is sensitive to the length of the domain. Effects associated with the dominant passage near the orifice plate, and some of the gradual approach to a Poiseuille-type flow between two copies of the orifice plate.
Figure 9. Pressure drop as a function of time to maintain constant volumetric flow rate (a) and pressure profile averaged over time as a function of the streamwise coordinate (b) for laminar flow at $Re = 1$, comparing orifices with the shape of a circle, square, triangle, star and no orifice.

seem to be incorporated in the simulated flow. However, a systematic variation of $L_x$ would be needed to assess the influence of the periodicity length.

3.2. Laminar mixing efficiency

We may also quantify the effect of the orifice plate in terms of its influence on the mixing efficiency of the flow that develops. There is no universal definition of a mixing rate in the literature. Here we focus on measures for the stretching and folding of fluid loops. Specifically, we define initially circular loops with their center in the middle between two copies of the orifice plates, by placing a large number of test particles on these loops. Each of these particles on each of the loops is then tracked as they move through the domain. The test particles do not have any inertia and precisely follow the streamlines without altering the flow. Tracking all particle paths of the particles that were initially placed on the same loop defines the evolution of that loop. As the contours travel through the pipe, they will reach the end of the flow domain; in that case we apply periodic conditions which allow us to track the loops as they move several times through an orifice plate. This allows us to compute the dynamic properties of the evolving loops such as the contour length and the dispersion relative to the center of gravity of the loop.

To compute the stretching and dispersion characteristics of the flow, we introduce circular loops in the $x$, $y$, and $z$ plane with centers in the middle of the pipe and radii $(i/16)L_y$ with $i = 1, 2, 3, 4, 5$. In total, we include 15 loops, 5 in each coordinate plane, and 1000 particles on each loop to obtain suitably averaged mixing characteristics. We refer to the five loops with a constant $x$-coordinate as $x$-loops, and similarly identify $y$-loops and $z$-loops according to whether the loop is in a plane normal in the $y$- or $z$-direction, respectively. All loops are localized near the centerline, within the inner 1/3 part, i.e. well separated from the cylinder wall in order to limit possible symmetry problems due to numerical errors arising from the IB representation of the solid–fluid interface on a Cartesian grid. In the case of flow through a simple cylindrical pipe along the $x$-direction, all particles on a specific $x$-loop have virtually the same numerical streamwise velocity, as should be in view of the dependence of the streamwise velocity on the radial coordinate only in Poiseuille flow. Due to the staircase representation of the solid–fluid interface, slight variations of less than $10^{-4}$, relative to the
maximum velocity in the center, could be observed in the streamwise velocity of all particles on \( x \)-loops. Distortions of the \( x \)-loops in Poiseuille flow due to these small numerical errors are negligible compared to the natural physical distortions of the \( y \)- and \( z \)-loops in the same flow, for which there is considerable variation of the streamwise velocity over the loop. In this case, the mixing efficiency is due to distortions in the \( y \)- and \( z \)-loops; this is expressed by a drift in e.g. the average contour length taken over all 15 \((x, y, z)\)-loops, in the case without an orifice plate. In case orifice plates are introduced all \( x \)-, \( y \)- and \( z \)-loops contribute significantly to the mixing efficiency and the increase in the contour length is much more pronounced.

The loops are introduced after the simulation has reached the steady state. We track the mixing as a function of time and use as the reference time-scale the flow-through time \( T_{\text{pipe}} \), i.e. the time a particle would take to flow a distance \( L_x \) with the bulk velocity. From figure 10(a), we observe that the orifice geometries that possess the largest number of sharp corners exhibit the fastest growth of the contour length. This result is not surprising since the presence of more sharp corners causes larger differences in the transverse velocities, which allows the contour to deform. We can also record the center of gravity of the fluid loops and use this to quantify a measure for average dispersion by monitoring the growth of the average distance of the particles on a loop to the center of that loop. The dispersion characteristic as shown in figure 10(b) is obtained by averaging over all 15 loops. The average stretching rate in figure 10(a) is seen to increase by a factor of up to 5, comparing pipe flow without an orifice plate to flow passing through an orifice plate derived from the Koch snowflake fractal. The dispersion rate is seen to increase by a factor of up to 4, cf figure 10(b). Generally speaking, the introduction of an orifice introduces considerable increase in the mixing rate, as seen from the difference between a clean pipe and a circular orifice. In addition, adding fine structure to the perimeter of the orifice leads to a further increase of the mixing rate.

4. Reduced turbulent pressure drop

In this section, we apply the volume penalization IB method to simulate turbulent flow in a (fractal-orifice) pipe flow. First, we investigate the convergence of the numerical solution with increasing spatial resolution. Then we apply the method to determine the
pressure drop needed to sustain a prescribed volumetric flow rate through the pipe in the turbulent regime.

In order to assess the accuracy with which turbulent flow through a cylindrical pipe can be computed on the basis of our volume penalization IB method, we compare results obtained on a set of resolutions with results from the literature using conventional body-fitted finite volume discretizations employing cylindrical coordinates. Results of the literature are taken from turbulent pipe flow at $Re = 5300$ based on the pipe radius. We compare these with simulations by Eggels et al and Unger et al, using a spatial resolution of $256(x) \times 128(\phi) \times 96(r)$, and Kasagi et al, who adopted a grid resolution of $512(x) \times 128(\phi) \times 96(r)$ for a twice as long pipe compared to Eggels et al and Unger et al. In figure 11, we show the grid convergence of the time-averaged mean (a) and rms (b) velocity profiles using four different grids. The mean and rms profiles are obtained by adopting time averaging over sufficiently long time intervals and averaging over $\phi$ after tri-linear interpolation from the Cartesian grid on which the IB simulations were conducted. We observe a clear convergence of IB results to the data found in the literature. Even at rather modest spatial resolutions fair agreement is observed between the finite volume IB approach and the body-fitted simulations, consistent with plane channel results reported in Verstappen and Veldman (2003). In fact, the first grid point off the wall was found to lie at $y^+ \approx 100$ at the $16 \times 8 \times 8$ grid, which decreases to $y^+ \approx 4$ in case the resolution was increased to $128 \times 64 \times 64$. The use of a skew-symmetric, energy-preserving spatial discretization, implemented on a staggered grid, appears essential to this. The friction velocity $u_\tau$ was obtained from its correlation with the bulk velocity $U_b$, i.e. $U_b/u_\tau = 14.7$. This correlation was established for pipe flow (Eggels et al 1994) as well as for flow in a square duct (Gavrilakis 1992). It was compared to a more direct evaluation using the shear stresses estimated at the wall using tri-linear interpolation—good general agreement was observed for the highest resolution. As can be expected, this convergence is more explicit in the case of the mean velocity profile, while for the rms profiles higher resolutions would be advantageous in order to achieve full agreement with the literature. This was also observed in relation to the influence of discretization errors in large-eddy simulations in which the role of subgrid resolution was investigated (Vreman et al 1994, 1996). The agreement is deemed sufficient and convincing given the fact that the IB results do not employ a non-uniform grid that is finer close to the cylinder wall.

![Figure 11. Convergence of mean (a) and rms (b) velocity profiles in turbulent pipe flow at $Re = 5300$ comparing IB results at resolutions $16 \times 8 \times 8$, $32 \times 16 \times 16$, $64 \times 32 \times 32$ and $128 \times 64 \times 64$ with finite volume discretization results on body-fitted grids in cylindrical coordinates.](image)
Turbulent flow through a pipe with a mounted orifice plate is conducted at $Re = 4300$. We selected this slightly lower Reynolds number as we still observed turbulent flow at this value and the resolution requirements can be met more fully given the selected resolution of $128 \times 64 \times 64$. To quantify the effect of the orifice plate under turbulent conditions, we consider the turbulent kinetic energy (TKE) defined as

$$TKE = (u^{rms})^2 + (v^{rms})^2 + (w^{rms})^2,$$

(6)

where the rms fields were obtained using time averaging only. The TKE is a measure of the energy of the velocity fluctuations. Without an orifice plate, the highest TKE is found in the vicinity of the wall. In case an orifice plate is present, the TKE maps are shown in figure 12 at the upstream edge of the orifice plates and in the middle of the orifice plates. We observe that at the upstream edge of the orifice plates highest turbulent velocity fluctuations are found near ‘corners’ in the perimeter of the orifice. Moving through the thickness of the orifice plates, we observe a rapid spreading and high TKE values all along the perimeter.

Turbulent transport through an orifice shows an interesting reduction in the pressure drop required to maintain the imposed volumetric flow rate. In figure 13(a), we plotted the pressure drop as a function of time for all situations with an orifice, as well as for the flow without a flange. We observe that the square, triangle and star exhibit approximately the same pressure drop, while the Koch-2 geometry leads to a pressure drop that is approximately 5% larger. The largest pressure drop is observed in case a circular orifice is employed, which is approximately 15% larger than the pressure drop for a square, triangle or star geometry. In contrast to the laminar flow (cf figure 9) where the fractal orifice was found to lead to the largest pressure drop and the circular orifice to the lowest, the situation is now reversed and on average the highest pressure drop is seen to arise for the circular orifice. This interesting reversal of the order from lowest to highest pressure drop was considered in more detail.

1. In the case of laminar flow, viscous effects are dominant—in such situations the use of a fractal orifice at constant porosity implies that a relatively larger fraction of the perimeter is closer to the cylinder wall, compared to the circular orifice. Hence, the region where decelerating viscous forces are strongest is larger with fractal orifices. In order to maintain the prescribed overall flow rate, the velocity near the center needs to be correspondingly higher for fractal orifices. The combination of stronger viscous friction near the perimeter of the orifice and higher velocities near the center implies generally larger velocity gradients and in turn an increased pressure drop for laminar flow in the case of fractal orifices.

2. In the case of turbulent flow, viscous effects are dominant only very close to walls. The circular orifice was found to induce considerable back flow directly downstream of its location, and correspondingly also considerable forward flow in the central region in order to maintain the imposed flow rate. This gives rise to large velocity gradients and an associated large pressure drop. The region of back flow for a circular orifice was found in the shape of a ring adjacent to the pipe wall. The other orifice geometries did not show such a continuous ring but rather a region of back flow that was ‘punctured’ behind locations where the orifice showed sharp protruding corners in its geometry. Particularly in turbulent flow this puncturing of the back flow ring is significant since quite a fraction of the flow goes through the protrusions. As a result, the overall back flow is reduced in the case of a fractal orifice, and hence also the forward flow does not need to be as strong to maintain the imposed flow rate. Likewise, the use of a circular orifice implies that on average the flow has to curve inward more than in the fractal case, in order to pass
Figure 12. TKE profiles at $x = 4.6875$, at the upstream edge of the orifice plates, and at $x = 5$, in the middle of the plate, for $Re = 4300$ and different orifice plates: circle (a, b), square (c, d), triangle (e, f), star (g, h) and second generation Koch snowflake (i, j).
Figure 13. Pressure drop as a function of time to maintain constant volumetric flow rate (a) and pressure profile averaged over time as a function of streamwise coordinates (b) for turbulent flow at $Re = 4300$, comparing orifices with the shape of a circle (maximal pressure drop), square, triangle, star, Koch-2 and no orifice (lowest pressure drop).

through the opening. On average the velocity gradients associated with a fractal orifice are lower than for the circular case, leading to lower pressure drop in turbulent flow for a circular orifice.

This sequence of results is also seen in the time-averaged pressure profiles shown in figure 13(b). Comparing the turbulent pressure profiles with the laminar results in figure 9(b), we observe that the profiles are quite similar, indicating that the presence of an orifice presents mainly a geometric partial blocking to the flow, likely to be determined by the porosity as the primary contribution. Also, the difference between flange/no-flange is much more pronounced.

5. Concluding remarks

In this study, we used DNSs in combination with an IB method for the simulation of pipe flow with an orifice plate. This computational model was found to be suitable to generate the dominant flow structures in the highly viscous laminar regime at $Re = 1$ as well as in turbulent flow conditions at $Re = 4300$. For turbulent flow the energy-preserving skew-symmetric discretization was found to yield good convergence with increasing spatial resolution; reliable simulation results for the mean velocity were observed already on uniform grids at a resolution of $64 \times 32 \times 32$ and an accurate agreement with body-fitted simulation results in cylindrical coordinates was seen at a resolution of $128 \times 64 \times 64$.

The volume penalizing IB method assigns the property ‘fluid’ or ‘solid’ to each control volume in the computational domain. The advantage of the IB method is that also very complex boundaries are incorporated easily—the computational effort does not depend on the complexity of the domain. However, more complex domains will require a higher spatial resolution in order to also capture the smaller geometrical structures. The disadvantage of the IB method is that for every IB that does not align with the grid, the position of the solid–fluid interface is accurate only to the mesh spacing. The current IB approach is only first-order accurate (Lopez Penha et al 2011, Mikhal and Geurts 2011, 2012) and employs a simple staircase approximation for the solid–fluid interfaces since each grid cell is allowed to take only one property, i.e. solid or fluid. More work is needed to take the current
volume penalization IB method to a higher order approach that allows for sub-grid-cell resolution of the interface—this is a subject of ongoing research extending work in Sarthou et al (2008).

Laminar flow through a range of orifice plates was studied, including circular, square, triangular, star and Koch-2 shapes at a porosity of 0.4. At low Re, we observed an increase in the pressure drop that maintains the flow rate, with an increasing number of sharp corners along the perimeter. The simulations show that a large number of sharp corners result in a large enstrophy close to the orifice perimeter. A longer perimeter implies that a large part of the orifice area lies close to the wall, implying that the domain at which viscous effects have a significant influence becomes larger and ultimately results in a larger shear stress and a larger pressure drop. This resulted in an increase of up to 30% in the pressure drop associated with a Koch-2 geometry, compared to a circular orifice. In the laminar regime, we also observed a significant increase in mixing efficiency with increased complexity of the perimeter of the orifice. This can have practical implications for laminar mixers without any moving parts.

For the turbulent case of $Re = 4300$, the highest axial velocities do not always occur at the center of the pipe. Consequently, the viscous effects have less influence and in some parts of the domain in the vicinity of the orifice plate, the velocities and TKE are higher near the wall. The pressure drop required to maintain the flow was found to be highest for a circular orifice and up to 15% lower for shapes with a number of sharp corners.

In this study, we concentrated on flow structures around a ‘fractal’ orifice plate. The question that remains is whether the fractal orifice can be useful for industry. There is no uniform answer to this question, since it depends on the flow that will be used. For laminar flow, we found that the pressure drop is larger for a ‘fractal’ geometry of the orifice compared to the pressure drop across a circular orifice. So when one wants to achieve a constant mass flow in the laminar regime, the ‘fractal’ orifice will cost more energy than the traditional circular orifice. However, when the focus of the process lies on mixing of laminar flows, then the ‘fractal’ orifice is more favorable than the circular orifice. An increase in the generation of the fractal will result in a higher pressure drop and higher mixing rates.

For turbulent flow, we found that the ‘fractal’ geometry exhibits a smaller pressure drop than the traditional circular orifice. The TKE patterns showed largest values near the perimeter of the orifice. Particle tracking in a turbulent flow obtained with IB resulted in some cases in particles getting caught in the solid wall. A consistent particle-tracking method needs to be developed that avoids numerical collision of massless particles with a solid wall. This development will allow us to investigate the influence of the orifice shape on turbulent mixing, which is the subject of future research.

It is still an open question whether the perimeter of the orifice should be drawn from a fractal construction in order to have optimal conditions for mixing or (turbulent) transport. For that purpose, dedicated physical experiments should be incorporated as well, in the same vein as in Nicolleau et al (2010), in order to reach realistically high Reynolds numbers with current simulation techniques. Research into IB simulations is also continued in this direction by developing higher order IB approaches and high-performance computing.

**Acknowledgments**

The simulations for this paper were made possible through a grant from NCF under project number SH-061. This support is gratefully acknowledged.
References


Angot Ph, Bruneau C H and Fabrie P 1999 A penalization method to take into account obstacles in incompressible viscous flows Numer. Math. 81 497–520


Cekli H E and Van de Water W 2010 Tailoring turbulence with an active grid Exp. Fluids 49 409–16


Geurts B J 2001 Mixing efficiency in turbulent shear layers J. Turbul. 2 17


Geurts B J and Holm D D 2006 Leray and LANS-α modelling of turbulent mixing J. Turbul. 7 1–33

Hurst D and Vassilicos J C 2007 Scalings and decay of fractal-generated turbulence Phys. Fluids 19 035103


Kuczaj A K and Geurts B J 2007 Mixing in manipulated turbulence J. Turbul. 7 1–28


Mazellier N and Vassilicos J C 2010 Turbulence without Richardson–Kolmogorov cascade Phys. Fluids 22 075101

Mikhail J and Geurts B J 2011 Bounding solutions for cerebral aneurysms New Arch. Math. 5 163–70


Moody L F 1944 Friction factors for pipe flow Trans. ASME 8 671–84


Ranz W E 1979 Applications of a stretch model to mixing, diffusion and reaction in laminar and turbulent flows AIChE J. 25 41–7

Reynolds O 1883 An experimental investigation of the circumstances which determine whether the motion of water in parallel channels shall be direct or sinuous and of the law of resistance in parallel channels Phil. Trans. R. Soc. 174 935–82

Seoud R E and Vassilicos J C 2007 Dissipation and decay of fractal-generated turbulence Phys. Fluids 19 105108
Wu X and Moin P 2008 A direct numerical simulation study on the mean velocity characteristics in turbulent pipe flow J. Fluid Mech. 608 81–112