Journal of Hydroinformatics | 04.1 | 2002

Fuzzy rules based model for solute dispersion in an open channel dead zone

Helen Kettle, Keith Beven and Barry Hankin

ABSTRACT

A method has been developed to estimate turbulent dispersion based on fuzzy rules that use local transverse velocity shears to predict turbulent velocity fluctuations. Turbulence measurements of flow around a rectangular dead zone in an open channel laboratory flume were conducted using an acoustic Doppler velocimeter (ADV) probe. The mean velocity and turbulence characteristics in and around the shear zone were analysed for different flows and geometries. Relationships between the mean transverse velocity shear and the turbulent velocity fluctuations are encapsulated in a simple set of fuzzy rules. The rules are included in a steady-state hybrid finite-volume advection–diffusion scheme to simulate the mixing of hot water in an open-channel dead zone. The fuzzy rules produce a fuzzy number for the magnitude of the average velocity fluctuation at each cell boundary. These are then combined within the finite-volume model using the single-value simulation method to give a fuzzy number for the temperature in each cell. The results are compared with laboratory flume data and a computational fluid dynamics (CFD) simulation from PHOENICS. The fuzzy model compares favourably with the experiment data and offers an alternative to traditional CFD models.

Key words | dead-zone, diffusion, dispersion, fuzzy, solute, turbulent

Helen Kettle (corresponding author)
Department of Geology and Geophysics,
Edinburgh University,
Edinburgh EH9 3JW,
UK
Tel: (44) 0131 650 8511
E-mail: *Helen.Kettle@glg.ed.ac.uk*

Keith Beven
Barry Hankin
Environmental Science,
Lancaster University,
Lancaster LA1 4YQ,
UK
Tel: (+44) 1524 593892
Fax: (+44) 1524 593985
E-mail: *K.Beven@lancaster.ac.uk*

INTRODUCTION

The dispersion of a solute in a natural channel is governed by molecular diffusion and the velocity field within the channel. Flow in natural channels is turbulent, and therefore mixing by molecular diffusion may be ignored since it is considered to be negligible compared with mixing by turbulent eddies. The turbulent velocity field is governed by the channel's shape and bed-surface characteristics. Large- and small-scale irregularities along the channel bed and banks may lead to the development of regions of slow moving or recirculating water known as dead zones, which greatly affect the dispersion of a solute. It is therefore important to understand more about the mixing processes at work in these areas. Water in the main river channel moves at a higher speed than that in the dead zone, creating a shear zone between these two areas. Momentum is diffused across this shear zone in the form of turbulent eddies. Velocity measurements obtained at a fixed point over time in a turbulent field indicate the presence of these eddies with a time series that fluctuates significantly about a mean value. The larger the magnitude of the average fluctuation, the more intense the turbulence.

Traditionally, the mixing of a solute in a turbulent fluid relies on the assumption that turbulent diffusion can be described in an analogous way to molecular diffusion (Taylor 1921). Molecular diffusion uses Fick's law, which states that the mass flux is proportional to the concentration gradient. The same approach is applied to turbulent diffusion, but the constant of proportionality, e_T, is very much larger, such that in one dimension, in the x-direction mass flux (in kg/m^{2}s) is given by

$$q_x = -e_t \frac{\partial C}{\partial x} \quad (1)$$

where C is the concentration and x is distance. The turbulent diffusion coefficient e_t (in m^2/s) is very hard to estimate since it must describe mixing over the many scales of motion present in a turbulent field. An alternative to Fick's law is to assume that since the tracer is dissolved in the water it will be transported directly by the turbulent fluctuations in the velocity field, so that mixing is modelled by the exchange of water back and forth across a boundary. The mass flux between two adjacent fluid regions is thus given by

$$q_x = u'_{n+1/2}\,(C_n - C_{n+1}) \tag{2}$$

where $u'_{n+1/2}$ is the velocity fluctuation at the boundary between regions n and $n+1$.

The mixing experiments in this study involve the measurement of a steady-state temperature field resulting from the continuous release of hot water into a dead zone in a laboratory flume. At a steady state the measured temperature field fluctuates around a mean value owing to the turbulent velocity fluctuations. In an attempt to predict the mean temperature field, a finite-volume model is formulated in which the mixing of the hot water is governed by a mean velocity field obtained from a k–ε turbulence model, and a fluctuating velocity field which is predicted by a set of fuzzy rules. The fuzzy rules are derived from velocity data collected in and around a laboratory flume dead zone for different geometries from those used in the mixing experiment. The input to the rules is the non-dimensionalized transverse mean velocity shear, which is used to calculate the mean longitudinal velocity fluctuation, u', as a fuzzy number. The fuzzy number for the cross-stream velocity fluctuation is related to this by a linear relationship derived from the flume data. Fuzzy numbers for velocity fluctuations are found for each cell boundary, and are combined using the single-value simulation method (Chanas and Nowakowski 1988) to calculate the temperature field resulting from the release of hot water in the corner of the dead zone. A fuzzy number for temperature is thus found for each cell and compared with the experiment data. The fuzzy sets for u' and v' represent the uncertainty with which the fluctuating field may be predicted, and this is carried through the model to produce a possible distribution for the mean temperature at each point in the domain.

Traditional computational fluid dynamics (CFD) models, such as the mixing-length model and the standard k–ε model (Launder and Spalding 1974), simulate the mixing of a scalar by relating turbulent diffusivity, e_t, to the kinematic turbulent viscosity, ν_t, such that

$$e_t = \frac{\nu_t}{\sigma_t} \tag{3}$$

where σ_t is a constant shown experimentally to be around 1. Both of these models rely on the use of Fick's law. The mixing-length model is not appropriate for recirculating flows, so only the k–ε model is used in this study for comparison with the fuzzy model. The k–ε model uses the rates of production and dissipation of turbulent kinetic energy (k and ε, respectively) to calculate ν_t from

$$\nu_t = \frac{C_\mu k^2}{\varepsilon} \tag{4}$$

where C_μ is a dimensionless constant. This model is well established and widely validated, but it is not easy to ascertain the uncertainty involved in the calculation of e_t. The fuzzy model provides a very simple and transparent approach to the problems of uncertainty in turbulent flow. By allowing the inclusion of more information into the model structure, more information is provided in the output. Where the k–ε-based model provides a single-value output for the transported scalar, the fuzzy model gives a distribution of values.

METHODS

Experimental equipment and procedures

The experiments were conducted in two different-sized tilting flumes into which a plastic dead-zone structure was placed, as shown in Figure 1. The length of the dead zone was fixed at either 30 cm (as shown in Figure 1) or 15 cm. The channel bed and walls were smooth, and flow conditions were such that the variation in the free surface was small. Four different flow conditions were set up; the hydraulic conditions are shown in Table 1.

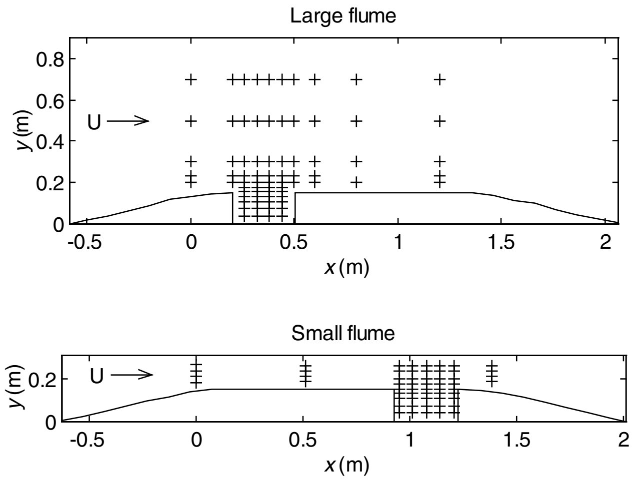

Figure 1 | Plan view of experimental conditions showing velocity data collection points, +, in large and small flumes.

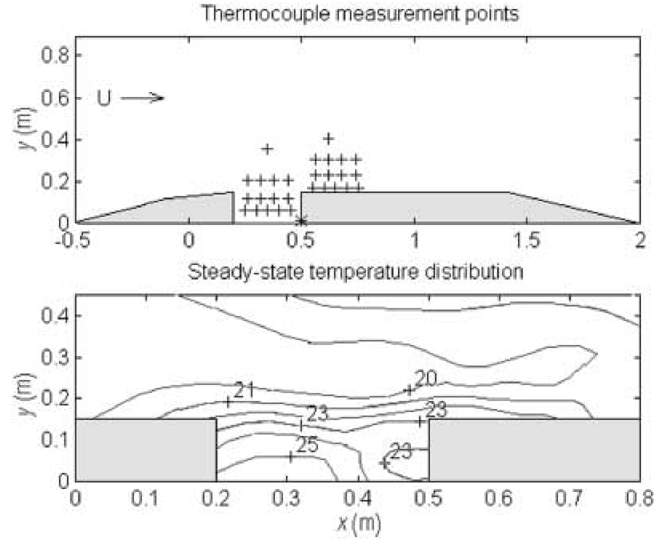

Figure 2 | Plan view of the steady state time-averaged temperature field in °C, resulting from hot water release at the point marked * in the corner of the dead zone.

Instantaneous velocity measurements were made using a 3-D acoustic Doppler velocimeter (ADV). This instrument measures instantaneous flow velocities at 25 Hz, in three dimensions, at a measuring volume of approximately 1 cc located 5 cm below the sensor head. The accuracy of this instrument was investigated by Lane *et al.* (1998), and was found to give good results for mean velocities. Velocity measurements were collected over a period of 100 s at each data point. The velocity time-series data were then decomposed into mean and fluctuating components, such that the mean of the fluctuating velocity was taken to be the sample standard deviation of the time series.

In the large flume with the conditions specified by Case 1 in Table 1, hot water was released at a constant rate and temperature into the corner of the dead zone at the point marked with an asterisk in Figure 2. Once a steady-state distribution was achieved, thermocouples located 5 cm above the bed at the positions shown in Figure 2 sampled the water temperature at 10 Hz for 5 min. The

Table 1 | Hydraulic conditions for flume experiments. U_f is the average longitudinal velocity in the main channel

Case	Flume	Dead zone width (cm)	U_f (cm/s)	Water depth (cm)	Height of velocity data above bed (cm)	Reynolds number, Re	Froude number, Fr
1	Large	30	28.7	16.3	5	48,800	0.237
2	Large	30	18.5	24	8	45,500	0.124
3	Small	30	22.6	28	5.6	67,100	0.145
4	Small	30	22.6	28	19.6	67,100	0.145
5	Small	15	21.3	28	19.6	117,400	0.253

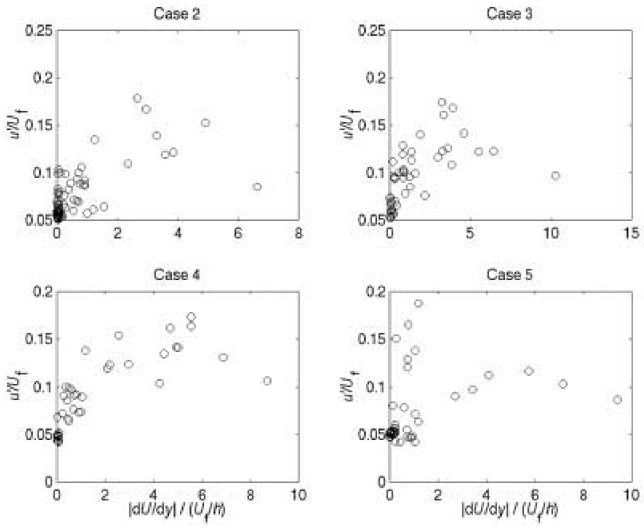

Figure 3 | Non-dimensionalized transverse velocity shear against non-dimensionalized longitudinal velocity fluctuations.

mean of the temperature time series for each thermocouple was then used to plot the average steady-state temperature field in the vicinity of the dead zone, as shown in Figure 2.

Mean transverse velocity shear and turbulence characteristics

In the following analysis, all velocity data except those specified by Case 1 in Table 1 were used for the fuzzy regression and to build the fuzzy inference system. The Case 1 data came from the same experiment as the hot-water tracing experiments and were excluded so that they could be used, together with the recorded temperature data, in an evaluation of the model predictions.

To predict the average magnitude of the turbulent velocity fluctuation at a point in the flow domain, the local transverse velocity shear, $\partial U/\partial y$, was used. To establish a general relationship the variables were non-dimensionalized, using the mean longitudinal velocity U_f in the main channel to non-dimensionalize u', and U_f/h (where h is the water depth) to non-dimensionalize $\partial U/\partial y$. The relationship between the non-dimensionalized data from Cases 2–5 is shown in Figure 3. However, the relationship is not well defined, suggesting that for a given transverse velocity shear a range of u' is possible. This lends itself easily to a fuzzy rules-based system.

Fuzzy sets and fuzzy rules

A *fuzzy set* is a set of objects in which each object has a *membership function* assigned to it that indicates the extent to which the object belongs to the set. An example of a fuzzy set is the set of 'fast velocities'. Since there is no definite boundary between 'fast' and 'not fast', each velocity reading belongs to the set to a different degree. The membership function of a fuzzy set, A, is denoted by μ_A, and is a mapping from the universal set, X, to the interval of real numbers from 0 to 1 inclusive. The closer that $\mu_A(X)$ is to 1 the more X belongs to A, and the closer it is to 0 the less it belongs to A.

Each fuzzy rule used in this work contains only input and one output such that logical operators (e.g. AND or OR) are not incorporated. In Boolean logic, a rule can either apply with absolute certainty or not apply. In fuzzy logic, a rule may partially apply, so that there may be cases where a few different rules with different consequences can, to a certain degree, be applied to the same input. The level to which a rule applies to a given input is termed the degree of fulfilment (DOF) of a fuzzy rule and has a value in the interval [0,1]. Figure 4 shows how two rules may be used by one input, and how the consequence of each rule may be combined to produce the final output fuzzy set. There are many methods for combining the rule responses (Dubois and Prade 1991). In this work the sum method, is used in which the final output is simply the sum of all fuzzy sets given by the rules.

Fuzzy rules for *u′*

In order to develop fuzzy rules which will apply to all of the data sets collected, the data were initially non-dimensionalized using the water depth, h, and the mean channel velocity, U_f, as shown for the individual cases in Figure 3. A total of 245 measurements from Cases 2–5 were then used to establish the relationship between $(\partial U/\partial y)/(U_f/h)$ and u'/U_f, as shown in Figure 5. The shape

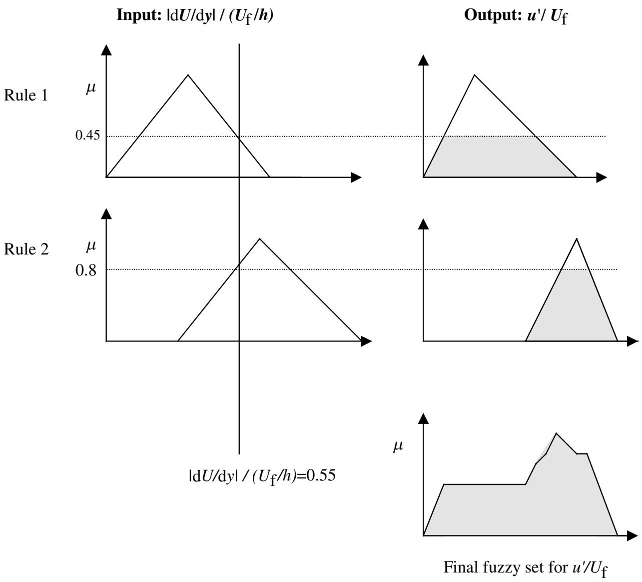

Figure 4 | A simplified example of how a fuzzy inference system uses a value for the non-dimensionalized transverse velocity shear in 2 rules to produce 2 fuzzy sets for the non-dimensionalized velocity fluctuation. These rule responses are then aggregated using the sum method to produce a single fuzzy set.

of this distribution indicates that the turbulence intensity can be high in regions of relatively low velocity shear such as in the dead zone where velocities are very low. Owing to the transportational nature of the flow, turbulent energy may be produced in the shear zone and then carried into regions of low velocity shear within the dead zone. This effect will occur, to varying extents, in any dead-zone geometry.

To generate the fuzzy sets to be used in the fuzzy rules, the support of the fuzzy set (i.e. the range for which the membership value is greater than zero) and the shape of the membership function must be estimated. The range for each variable is split into five parts, so that Range 1 for $(\partial U/\partial y)/(U_f/h)$ maps to Range 1 for u'/U_f and so on. These are then used as the supports of the five membership functions used to describe the mappings from $(\partial U/\partial y)/(U_f/h)$ to u'/U_f. The lines on Figure 5 show the chosen range of each support. These are chosen intuitively to give a simple and accurate description of the data. Once the

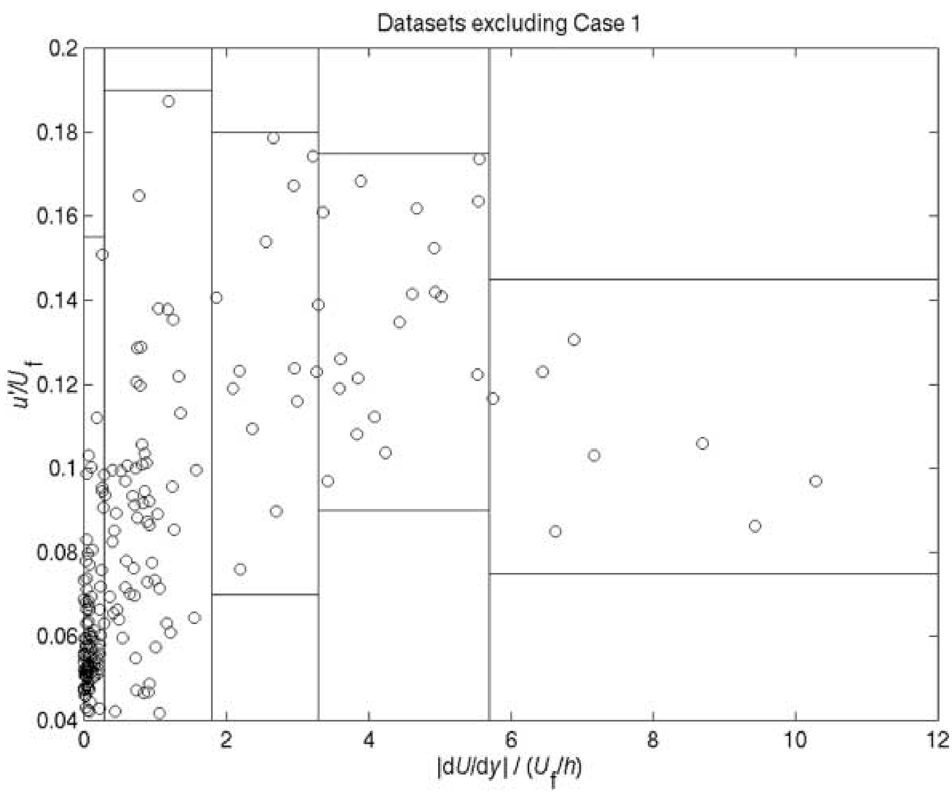

Figure 5 | Non-dimensionalized distribution of u' against non-dimensionalized transverse velocity shear. Straight lines show support of membership functions to be used in the fuzzy inference system.

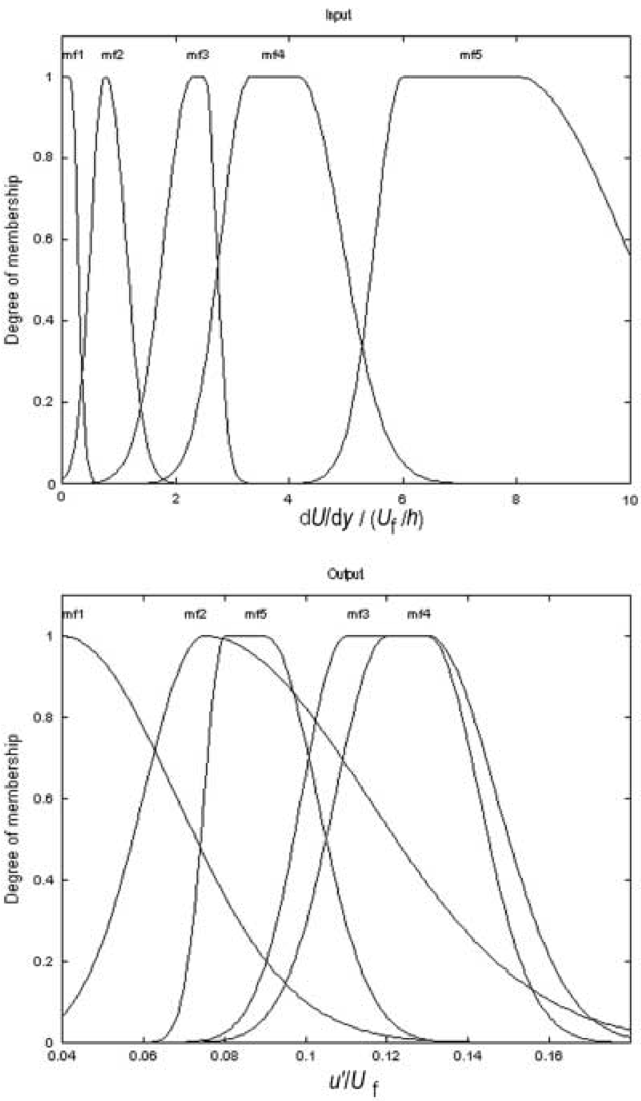

Figure 6 | Input and output membership functions for fuzzy rules relating non-dimensionalized transverse velocity shear and longitudinal velocity fluctuations.

supports have been established, the shape of the membership function is estimated based upon the frequency distribution of $(\partial U/\partial y)/(U_f/h)$ and the corresponding distribution of u'/U_f over the supports. The distributions are then scaled to give a maximum membership value of 1 in each range, as shown in Figure 6. For example, the fuzzy set for the lowest range of transverse velocity shear is calculated by first taking the frequency distribution of $|\partial U/\partial y|/(U_f/h)$ values in the interval [0,0.4]. This is then approximated to a Gaussian curve which overlaps with the next range of transverse velocity shear to ensure that the rule response is continuous. The support of the fuzzy set of u'/U_f to which this fuzzy set for transverse velocity shear maps is the range of u'/U_f values from 0.04 to 0.155. The shape of the membership function is then a Gaussian approximation of the frequency distribution of values in this range and can be seen to be weighted towards the lower values in the range (see Figure 6).

The fuzzy inference system consists of one input $(|\partial U/\partial y|/(U_f/h))$ and one output (u'/U_f). The membership functions for these are shown in Figure 6. The system is described by the five rules

If $(\partial U/\partial y)/(U_f/h)$ *is in fuzzy set 1 then* u'/U_f *is in fuzzy set 1,*

If $(\partial U/\partial y)/(U_f/h)$ *is in fuzzy set 2 then* u'/U_f *is in fuzzy set 2,*

. . . and so on.

The fuzzy inference system uses the product inference method (Bardossy and Duckstein 1995) to find the fuzzy set response of each rule. The outputs of each rule are aggregated using the sum method, as described above. The final fuzzy set of u' is then used in the fuzzy model.

The fuzzy inference system models a non-linear situation with a given transverse velocity shear producing a large range of turbulent velocity fluctuations which are represented by the output fuzzy set. Therefore, since the fuzzy inference system produces a fuzzy set for u' rather than a single value, it is difficult to quantify the validity of the rules as there is no one-to-one mapping between predicted and observed data sets. However, the accuracy of the fuzzy inference system was checked by comparing the minimum and maximum of the fuzzy set support and the mean of the values with a membership value of 1 with each observed value. It was found that the observed value lay within the range of the fuzzy set support for 95% of the measurements, and the correlation between the observed value and the mean of the predicted values with membership value 1 was 0.7.

Fuzzy sets for v'

In order to model mixing in two dimensions, the cross-stream velocity fluctuations v' must also be known. Fuzzy

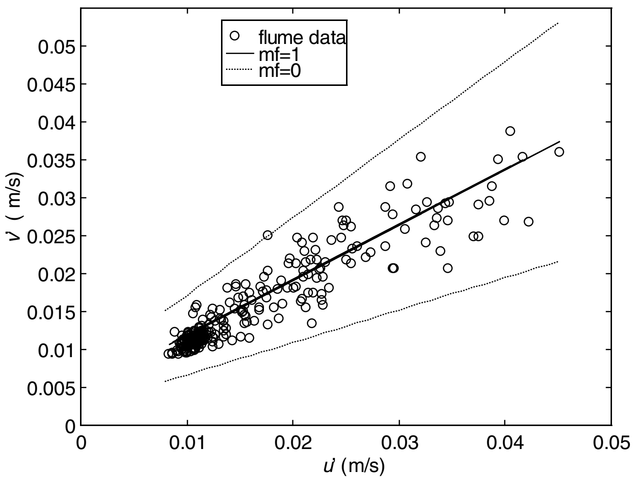

Figure 7 | Fuzzy regression for cross stream and longitudinal velocity fluctuations with v' shown as a fuzzy number whose most likely value is shown by the line where the membership function (mf) is one. The support of the fuzzy number is defined by the zero membership lines.

regression (Bardossy *et al.* 1990) was used to establish a relationship for v' based on u' using the data sets of Cases 2–5. In fuzzy regression, the regression parameters are fuzzy numbers and the dependent variable is also expressed as a fuzzy number. The aim is then to find the regression model which minimizes the fuzziness of the dependent variable. Figure 7 shows a fairly strong linear relationship with a correlation coefficient of 0.9. For a membership value of 1, fuzzy regression gives

$$v' = 0.7864u' + 0.0038 \quad (5)$$

Thus for a given value of u' the range of v' lies between the bounds of the zero membership values with the most likely value given by Equation (5). This describes a fuzzy number for v', and a single value of v' is chosen from this distribution according to the single-value simulation method for use in the finite-volume model.

Application to the steady-state mixing of hot water in an open channel with a dead zone

The release of hot water into the flume is a case of a 3-D flow with buoyancy effects. However, the flow is relatively shallow and the vertical mixing is strong, and therefore the buoyancy effects are considered to be negligible and a 2-D model is used as an approximation to the system. In order to model dispersion, the mean movement (advection) of the transported variable must be known. Since a grid scheme is to be used, it is necessary that the law of mass conservation is satisfied for each cell. To ensure this the velocity field must satisfy the continuity equation. Therefore a CFD code, rather than the experimental velocity data, was used to generate the mean velocity field. The fuzzy rules-based dispersion model requires values for mean longitudinal and transverse velocities everywhere in the flow domain in order to predict fuzzy values of u' and v'. In this study, the rules have been implemented into a finite-volume representation of the flow domain using a mean velocity field generated by the PHOENICS CFD code using the same spatial discretization. PHOENICS is a general-purpose computer code for the simulation of fluid flow, heat transfer, chemical reactions and related phenomena (Spalding 1972). For the present application the k–ε turbulence model with the hybrid numerical solution scheme was invoked within PHOENICS version 1.5. The region of the flume to be simulated was 2.5 m in the longitudinal direction, 0.9 m in the cross-stream direction and 0.16 m vertically. PHOENICS was used to produce a 2-D velocity field over a domain divided into 56 cells in the longitudinal direction and 41 cells in the cross-stream direction. The grid was irregular, with smaller cells in areas of high-velocity shear. The results of the fuzzy model were compared with both the measured temperature field and the temperature field generated by PHOENICS.

The relevant fluid properties in the velocity field simulation are the kinematic laminar viscosity, ν, and the turbulent Prandtl numbers, σ. These were set such that $\nu = 1.138 \times 10^{-9}\ \mathrm{m^2/s}$ (Batchelor 1967), $\sigma(k) = 1.00$ and $\sigma(e) = 1.3$, where k and ε are the production and dissipation rates of kinetic energy per unit mass, respectively. The mean longitudinal velocity at the inlet was assumed to obey the turbulent log law, and values of turbulent kinetic energy there were estimated from the experimental data. The inlet and outlet boundaries are used to specify the pressure drop over the region (which drives the flow) by setting a fixed mass flux at the inlet and fixing the pressure to be zero at the outlet. The effective roughness height for the flume walls was set at 0.5 mm. The surface of the water

 Journal of Hydroinformatics | 04.1 | 2002

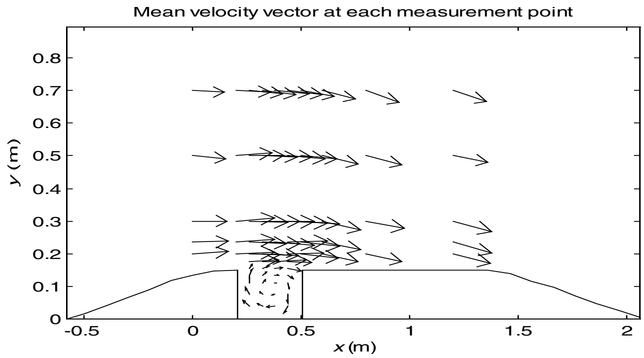

Figure 8 | Velicity vectors measured with ADV for Case1. Maximum is 0.31 m/s.

was taken to be a rigid zero-flux symmetry boundary, which was a reasonable approximation for the flow in the large flume. Figure 8 shows the mean velocity field measured by the ADV probe for experimental Case 1. Figure 9 shows how the PHOENICS velocity field compares with the experimental data. On average, the longitudinal velocity, *U*, is over-predicted and the cross-stream velocity, *V*, is under-predicted.

Simulation of temperature field using PHOENICS

To simulate the temperature field in PHOENICS, the same parameters used for the velocity field were taken with the addition of the turbulent and laminar Prandtl numbers (σ_T and σ_L) for temperature, such that $\sigma_T = 0.9$ (Versteeg and Malalasekera 1995) and $\sigma_L = 7$ (Batchelor 1967). The input of hot water is represented by a patch of fixed temperature. The diffusion of heat is controlled by the Prandtl numbers, which indicate the ratio of eddy viscosity to thermal diffusivity. Since eddy viscosity is based on the production and dissipation rates of turbulent kinetic energy, the thermal diffusivity will vary over the domain.

Simulation of the temperature field using a fuzzy-rules hybrid finite-volume model

The domain is divided into an orthogonal grid with varying cell sizes that match the output of the PHOENICS simulation to ensure the continuity of the velocity field. In

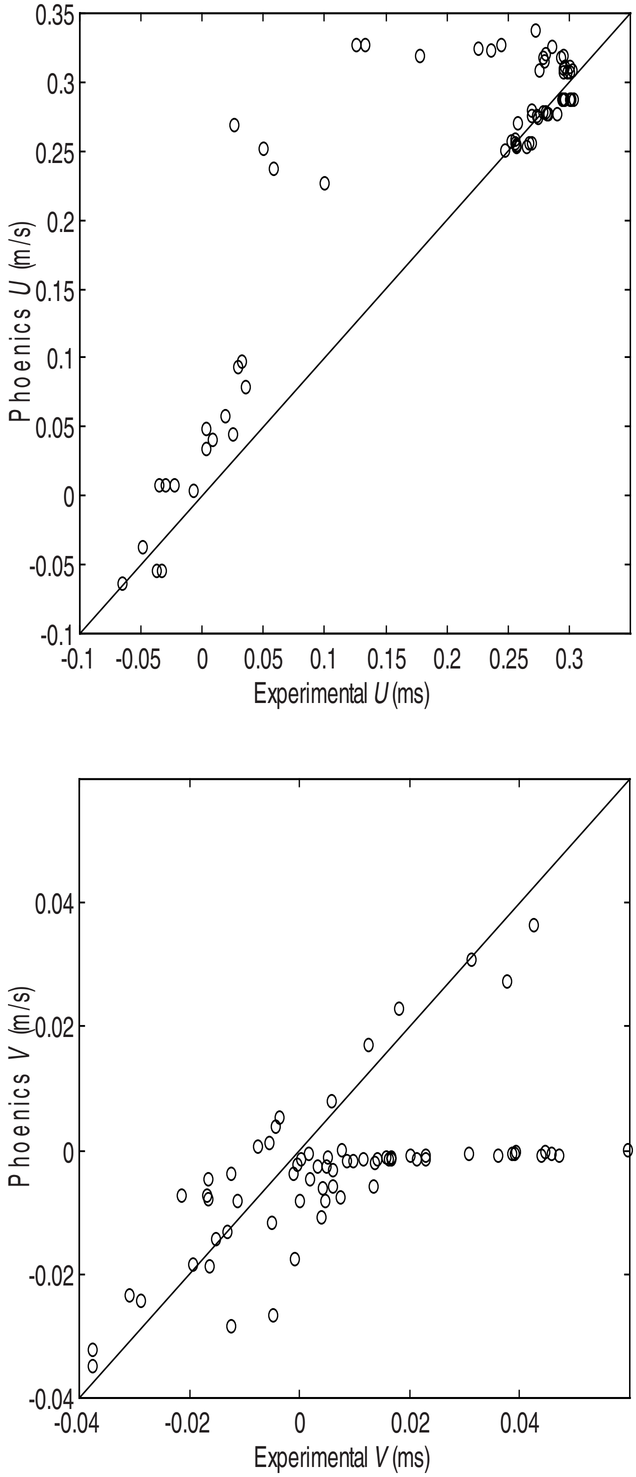

Figure 9 | Comparison of PHOENICS U, V velocity fields with the experiment data.

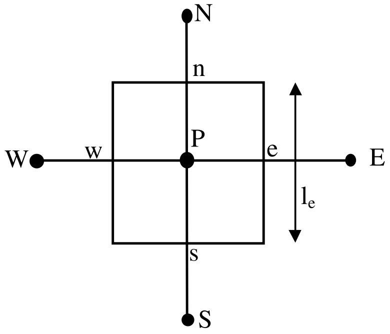

Figure 10 | A control volume around node P.

the absence of sources, the steady 2-D advection and diffusion by turbulent velocity fluctuations of a transported scalar property, φ, are governed by Equation (6). In the finite-volume formulation, as shown in Figure 10, the diffusive component is conceptualized as an equal and opposite exchange of fluid across a cell boundary. For example, fluid from cell P will move out of the cell at a speed u'_e into cell E, and fluid from cell E will move into cell P at a speed u'_e. The advective component is described by the mean movement of fluid across the cell centre, such that the total scheme is given by

Rate of change of φ by advection = rate of change of φ by diffusion
i.e.

$$\frac{\partial}{\partial x}(U\phi)\bigg|_p + \frac{\partial}{\partial y}(V\phi)\bigg|_p = u'_w \frac{\partial \phi}{\partial x}\bigg|_w + u'_e \frac{\partial \phi}{\partial x}\bigg|_e + v'_n \frac{\partial \phi}{\partial y}\bigg|_n + v'_s \frac{\partial \phi}{\partial y}\bigg|_s \quad (6)$$

with the flow satisfying the continuity equation. Upper-case subscripts denote node points (cell centres) and lower-case subscripts denote cell face values. Integrating over the control volume in Figure 10, with φ_i considered to be constant along the cell face i and flow into the cell taken as positive, gives

$$\varphi_w l_w U_w - \varphi_e l_e U_e + \varphi_s l_s V_s - \varphi_n l_n V_n = u'_w l_w(\varphi_P - \varphi_W) + u'_e l_e(\varphi_P - \varphi_E) + v'_s l_s(\varphi_P - \varphi_S) + v'_n l_n(\varphi_P - \varphi_N) \quad (7)$$

where l is the length of the cell boundary corresponding to the subscript. This is the general equation for the numerical scheme.

Hybrid finite-volume scheme

The values of φ at the cell faces in Equation (7) are calculated according to the hybrid scheme (Spalding 1972). This scheme uses a combination of the upwind and central differencing schemes, and exploits the transportational qualities of the upwind scheme and the increased accuracy of central differencing. If the central-difference scheme is used to find the values of φ at the cell faces, then the discretization scheme is only stable and accurate if the ratio of advection to diffusion is less than 2. In the work presented here, this non-dimensional value is given by $|U/u'|$ and is equivalent to the Peclet number used in traditional finite-volume schemes. This is defined as $Pe = |U\Delta x/e_t|$, where Δx is the cell length and e_t is the turbulent diffusion coefficient. Therefore, the hybrid scheme utilizes the central-difference scheme to calculate φ at the cell face if $|Pe_T| < 2$ at that face (where $Pe_T = U/u'$), otherwise the upwind scheme is employed. For example, if the east cell face has $|Pe_T| < 2$, the hybrid scheme uses a central-difference approximation to estimate φ at the east cell face, such that

φ_e = linear interpolation between φ_P and φ_E

However, if the east cell face has $|Pe_T| \geq 2$, then the upwind scheme is used, such that

$$\varphi_e = \varphi_P \text{ if } U_e > 0$$

$$\varphi_e = \varphi_E \text{ if } U_e < 0$$

This is easily applied in two dimensions by treating each cell boundary in the way described above. Once the values of φ at the cell faces have been found in terms of the cell-centre values of φ, the general Equation (7) is rearranged in the form

$$a_p\varphi_P = a_W\varphi_W + a_E\varphi_E + a_S\varphi_S + a_N\varphi_N + S_u \quad (8)$$

with

$$a_P = a_W + a_E + a_N + a_S + (U_e - U_w) + (U_n - U_s) - S_p \quad (9)$$

where the coefficients a_E, a_w, a_N, a_S and a_P depend upon the magnitude of the Peclet number at faces e, w, n and s, and the source term is a function of the dependent variable such that the source over the cell volume is given by $(S_u + S_p\varphi_P)$. The source term is defined so that it is possible to set the variable φ at node P to a value φ_{fix} by assigning S_p a very large arbitrary value Z and setting S_u to $Z\varphi_{fix}$. The value of Z must be large enough to render the other terms on the right-hand side of Equation (8) negligible, so that $\varphi_P = \varphi_{fix}$ (Versteeg and Malalasekera 1995).

Velocity fluctuations

The mean velocity field produced by PHOENICS is used to calculate the transverse velocity shear that is the input to the fuzzy-rule system. The rules then produce a fuzzy number for u'. A single value for u' is generated based on the single-value simulation method of fuzzy variables (Dou *et al.* 1997), termed the fuzzy-numerical simulation method by Chanas and Nowakowski (1988). This is done by generating a value t of the membership function on the interval [0,1]. This is the membership value at which the membership function for the fuzzy set u' is bounded below, so that a new fuzzy set (known as the t-level set) is created in which each member u'_t has a membership value greater than t. A value from the t-level set is then randomly chosen to be the single value for u'. The fuzzy number for v' is then found from a fuzzy regression relationship. A single value for v' is found in a similar way such that the same value of t is used to create the t-level set for v' and then a value from v'_t is randomly chosen. The single values of u' and v' are then used in the finite-volume model to calculate dispersion. The resulting temperature field gives the distribution at the chosen t-level. Multiple runs give possible temperature fields for each t-level, from which a fuzzy number for temperature may be reconstructed.

Heat source

To set the source of hot water in the corner of the dead zone, the value of φ in this cell is fixed at the appropriate temperature, φ_{fix}. This is done by setting the source terms such that

$$S_u = 10^{30}\varphi_{fix}$$

$$S_p = -10^{30}$$

Once the a_P coefficient for each node in the domain has been calculated, a large system of linear equations in the form of Equation (8) is obtained. Since there is an equation for each node, solving these equations simultaneously yields the value of φ at each node. These discretized equations are solved using the iterative tri-diagonal matrix algorithm (TDMA) as described in Versteeg and Malalasekera (1995).

RESULTS AND DISCUSSION

The model was run 200 times to give a distribution of possible temperatures at every cell in the domain, and these were then compared with the temperature field predicted by PHOENICS and the measurements made by the thermocouples in the flume. The thermocouples recorded temperatures over approximately 5 min, and large temperature fluctuations were found. This variability is due to temporal variations in the flow field, e.g. possible periodic motion caused by the dead zone geometry. The modelling work presented here does not incorporate these time-dependent events since the fuzzy variables are used only to represent the uncertainty in the calculation of the turbulent velocity field at a steady state. The fuzzy rules are used in this work to show that a given transverse velocity shear can produce a distribution of possible turbulent velocities, but they do not attempt to capture the time-varying nature of the turbulence. Therefore, the thermocouple data were averaged over time to give the mean temperature in an approximately steady-state situation and the measured temperature fluctuations are ignored. The possible range of temperatures at the thermocouple sites, given by the minimum and maximum predictions from the fuzzy model, are shown in Figure 11, and the location of the thermocouples is shown in Figure 12. Figure 11 shows that the temperatures recorded by the thermocouples in the dead zone (numbers 1–9) generally lie within the range predicted by the fuzzy model. However, PHOENICS constantly under-predicts the temperatures in this region, which may be because PHOENICS is underestimating the amount of turbulent diffusion in the dead zone. Since PHOENICS calculates the turbulent

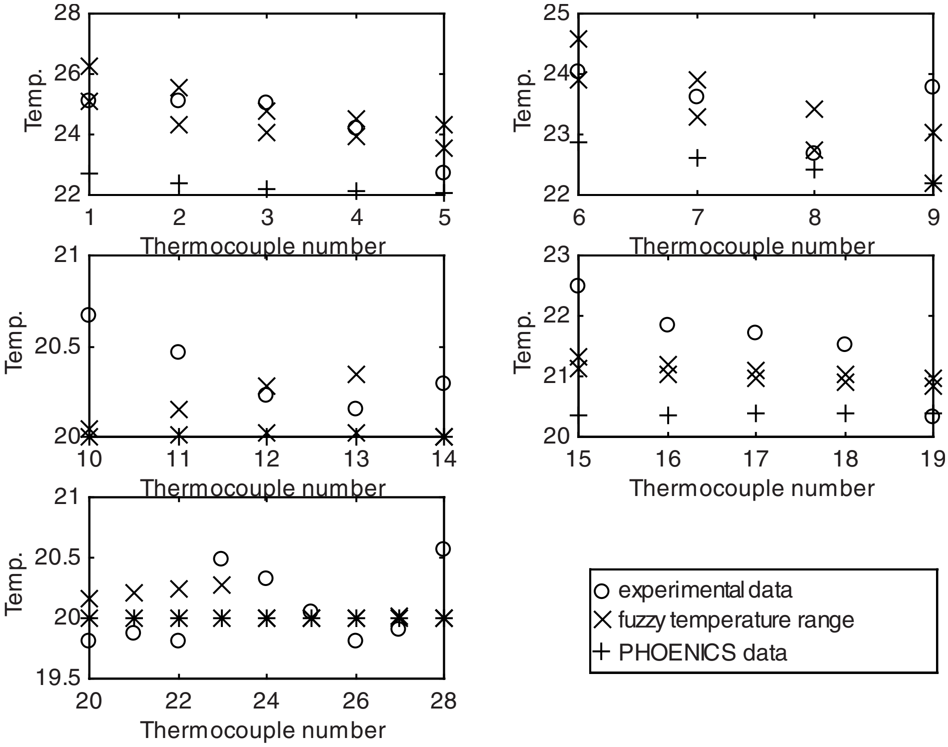

Figure 11 | Predicted and observed temperatures in °C for all thermocouple sites.

diffusion from the predicted production and dissipation of turbulent kinetic energy, this implies that PHOENICS is underestimating the turbulent energy within the dead zone. This mistake is not made by the fuzzy model since the fuzzy rules are designed to allow high turbulence intensities to occur in areas of low velocity shear. Outside the dead zone, the accuracy of the predictions from both the fuzzy model and PHOENICS is reduced as a result of the inaccurate mean velocity field (shown in Figure 9). At thermocouple sites 24, 25, 26 and 28, the fuzzy model produces a single value for the temperature. This is because the model predicts that the hot water will not reach into this part of the flow so the water temperature is unaffected by the heat source. Although the predicted range at each thermocouple does not always capture the observed data, on the whole the fuzzy predictions are nearer the observed value than the PHOENICS predictions, which is owing to the detailed information on turbulent diffusion provided by the fuzzy rules.

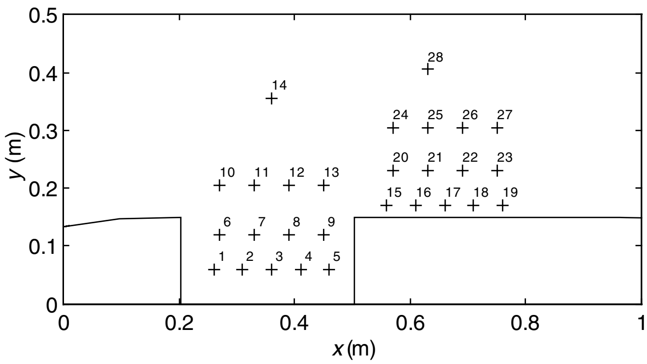

Figure 12 | Location and numbering of thermocouples.

CONCLUSIONS

In this paper an alternative approach to traditional CFD methods, based on fuzzy rules, has been presented to express the imprecise nature of mixing in a non-homogeneous turbulence field, and to incorporate this uncertainty into the model structure. For simplicity the model was developed in two dimensions and is therefore is only a rough approximation to the situation in the flume, which is 3-D due to the buoyancy of the hot water. However, given this loss of the vertical dimension, the model performs surprisingly well. The velocity field generated by the CFD package PHOENICS was not a perfect match with the measured velocity data but did provide a 2-D mean flow field which satisfied the continuity equation, as required by the mixing model. The fuzzy rules were implemented into a hybrid finite-volume scheme, and the model was run 200 times to achieve a distribution of possible temperature fields. Since both the fuzzy model and PHOENICS use the same mean velocity field and the same numerical scheme, the differences in temperatures predicted by these models imply that the temperature distribution is very dependent upon the estimated turbulent diffusion. It has been shown that the mixing can be adequately described by the fuzzy velocity fluctuations, and thus uncertainty can be incorporated directly into the model with no loss of accuracy. Future work is intended to apply this methodology to the more complex situations found in modelling transport in a natural river geometry (see Kettle *et al.* 2001).

NOMENCLATURE

a_E	coefficients of φ_E
a_N	coefficients of φ_N
a_P	coefficients of φ_P
a_S	coefficients of φ_S
a_W	coefficients of φ_W
C	concentration (kg/m^3)
e_t	turbulent diffusion coefficient (m^2/s)
ε	rate of dissipation of kinetic energy per unit mass (J/s/kg)
e	east cell face
h	water depth (m)
k	rate of production of kinetic energy per unit mass (J/s/kg)
l_e	length of cell face e (m)
μ	membership function of fuzzy set
ν	kinematic laminar viscosity (m^2/s)
ν_t	kinematic turbulent viscosity (m^2/s)
n	north cell face
P	centre cell node
Pe_T	Peclet number used in model
Pe	traditional Peclet number
σ_T	turbulent Prandtl number
φ	value of property per unit mass
S_p	source term
S_u	source term
s	south cell face
t	time (s)
u'	fluctuating longitudinal velocity (m/s)
U	mean longitudinal velocity at a point in the flow domain (m/s)
U_f	mean longitudinal velocity for main flow (m/s)
v'	cross-stream fluctuating velocity (m/s)
V	mean cross-stream velocity (m/s)
w	west cell face
x	distance (m)
y	transverse distance (m)

REFERENCES

Bardossy, A. & Duckstein, L. 1995 *Fuzzy Rule-Based Modelling with Applications to Geophysical, Biological and Engineering Systems*. CRC-Press, Boca-Raton, FL.

Bardossy, A., Bogardi, I. & Duckstein, L. 1990 Fuzzy regression in hydrology. *Wat. Resources Res.* **26**(7), 1497–1508.

Batchelor, G. 1967 *An Introduction to Fluid Dynamics*. Cambridge University Press, Cambridge.

Chanas, S. & Nowakowski, M. 1988 Single-value simulation of fuzzy variables. *Fuzzy Sets Syst.* **25**, 43–57.

Dou, C., Woldt, W., Bogardi, I. & Dahab, M. 1997 Numerical solute transport simulation using fuzzy sets approach. *J. Contaminant Hydrol.* **27**, 107–126.

Dubois, D. & Prade, H. 1991 Fuzzy sets in approximate reasoning (Parts 1 and 2). *Fuzzy Sets Syst.* **40**, 143–244.

Kettle, H., Hankin, B. & Beven, K. 2001 Fuzzy rule-based model of solute dispersion in a river dead zone. *J. Hydroinformatics* (in press).

Lane, S., Biron, P., Bradbrook, K., Butler, J., Chandler, J., Crowell, M., McLelland, S., Richards, K. & Roy, A. 1998 Integrated three-dimensional measurement of river channel topography and flow processes using acoustic Doppler velocimetry. *Earth Surface Processes Landforms* **23**(13), 1247–1267.

Launder, B. E. & Spalding, D. B. 1974 The numerical computation of turbulent flows. *Comput. Methods Appl. Mech. Engng.* **3,** 269–289.

Spalding, D. 1972 A novel finite-difference formulation for differential expressions involving both first and second derivatives. *Int. J. Numerical Methods Engng.* **4,** 551–559.
Taylor, G. I. 1921 Diffusion by continuous movements. *Proc. Math. Soc., Ser.* **2**(20), 196–212.
Versteeg, H. & Malalasekera, W. 1995 *An Introduction to Computational Fluid Dynamics—The Finite-Volume Method.* Longman Harlow, UK.