COMPUTATIONAL FLUID DYNAMICS: DEVELOPMENT AND DELIVERY OF A FINAL YEAR M.ENG. MODULE

Mark A. Cotton[1]

Abstract *¾ A new Final Year M.Eng. (Level 4) module, Computational Fluid Dynamics or 'CFD', was delivered in the School of Engineering during the 2000/01 and 2001/02 sessions. CFD is used widely in the practical computation of the motion of liquids and gases and represents a pivotal component of modern engineering technology. The module was developed in order to offer engineering students the opportunity to gain familiarity with the field at a near-professional level. In keeping with the nature of industrial CFD, the module is very much 'hands on' and is based around four partially open–ended computational projects that provide the participant with experience of state-of-the-art commercial software for computational grid generation and CFD solution and post-processing. The direct practical content of the module is supported by a number of seminars examining the physical basis of an 'industry standard' turbulence model.*

Index Terms ¾ CFD, 'hands on', professional environment, project based, turbulence models

INTRODUCTION

The present paper describes the experience gained over the past two years in the development and delivery of a Level 4 module in Computational Fluid Dynamics (CFD). An early decision was taken that the module would be centred around commercially-available CFD packages and a teaching licence was agreed with Fluent Europe Ltd. It was clear from the outset that a traditional lecture-centred teaching format would be largely inappropriate in the context of a software based module. Instead, it was considered that the general ethos of a professional development course would be well suited both to the technical content of the module and to the experience and abilities of the Final Year M.Eng. participants. In parallel with this delivery structure it was necessary to maintain due academic rigour and also devise a method of assessment that could sit alongside other Level 4 options available to M.Eng. students.

The main body of the paper is divided into two parts. The first section, 'A New Structure for Delivery', expands upon the nature of the module and describes its day-to-day running and various aspects of design and assessment. In the second section, 'Simulation versus Reality', the focus shifts to some technical aspects of CFD and the turbulence models that are central to the CFD representation of the majority of practically-occurring fluid flows.

A NEW STRUCTURE FOR DELIVERY

An overriding consideration in the design of this Level 4 module has been the desire to create a near-professional environment within an advanced educational context. The module is structured to have a number of similarities with industrial development courses likely to be experienced by engineering graduates. This has been done particularly with 'first job' development courses in mind, but also with knowledge of the kind of training opportunities likely to arise at various career stages. The structure of the module was informed in part by the writer's own experience of attending a 4-day intensive course at Fluent Europe's headquarters prior to delivering the module for the first time (notably all other delegates on the course were practising engineers from UK/EU organizations). It is sought to create a professional environment in a number of ways. Thus, for example, students access the CFD packages ('Gambit' [1] and 'Fluent 5' [2]) from workstations located in a dedicated advanced computing suite; a high level of support is available from teaching personnel (two research students and the writer assist approximately 15 participants), and teaching support is provided in an informal and accessible manner.

Project Based Delivery of the Module

The module is centred around four computational projects – these become increasingly open-ended as the work progresses. The initial assignments are, however, relatively closely defined, and indeed the first project is concerned solely with grid generation. In the second project greater initiative is required as here the Project 1 study is extended to encompass a full CFD analysis and solution. Subsequent refinement of the solution is also required. Project 3 introduces three-dimensional computations and gives the student further exposure to post-processing techniques. The final element is an advanced exercise: Project 4 consists of a completely open-ended problem in which the student is expected to design and execute an entire CFD analysis of an engineering flow. In greater detail, the content of the four projects is as follows:

- CFD Project 1. 'Gambit: Modelling a Mixing Elbow' – Computational Fluid Dynamics solutions are not obtained continuously throughout a flow domain, but instead results are found at the discrete nodes of a computational grid or mesh. The grid generator supplied by Fluent Inc./Fluent Europe Ltd. is known as 'Gambit'

---

[1] School of Engineering, University of Manchester, Manchester M13 9PL, U.K. mark.cotton@man.ac.uk

[1]. CFD Project 1 is based upon Tutorial 2 of the Gambit documentation [3]. A two-dimensional representation of a mixing elbow (a 90^o pipe bend with a smaller pipe joining at the elbow of the bend) is to be created. The Gambit model consists of the physical walls of the mixing elbow and a quadrilateral mesh covering the internal fluid domain. The mesh generated in the course of this exercise is used subsequently in Project 2. Furthermore, the procedures developed here are used independently by the student in Project 4.

- CFD Project 2. 'Fluent: Turbulent Flow and Heat Transfer in a Mixing Elbow' – The second exercise uses the CFD solution and post-processing package 'Fluent 5' [2]; it is based upon (and extends) Tutorial 1 of the Fluent 5 documentation [4]. Questions to be addressed here include turbulence model selection, the specification of fluid properties, and the application of boundary conditions. Higher order numerical discretization schemes and grid adaptation following initial solution are also investigated. ('Discretization' refers to the development of approximate forms of the specified set of partial differential equations prior to solution on the computational grid.) There are two extensions to the exercise provided in the documentation. First, the results obtained on a pre-generated triangular mesh supplied with the Fluent 5 tutorial are compared with those obtained on the quadrilateral mesh developed by the participant in Project 1. The tutorial uses the 'Standard k-e' turbulence model and, in the second extension made here, an alternative 'Realizable k-e' model is also examined. (k is the kinetic energy of turbulence and e its rate of dissipation by viscous action; turbulence models are discussed below under 'Simulation versus Reality'.)
- CFD Project 3. '3D Film Cooling using a Nonconformal Mesh' – Project 3 is a relatively short assignment based directly upon Tutorial 4 of the Fluent 5 documentation [4]. It is included in order to give the student experience of a fully 3-dimensional problem and also to provide further exposure to post-processing techniques (in particular the generation of x-y plots that permit 'hard' comparsions of CFD solutions with experimental data). Post-processing skills developed here will find application in relation to Project 4.
- CFD Project 4. 'Advanced Exercise. The Turbulent Round Jet' – The final exercise is completely open-ended, the participant being expected to use Gambit and Fluent 5 to make detailed comparison with the experimental data of [5] (selected as the standard reference case for the turbulent round jet by an independent international working group, [6]). It is expected that, based upon earlier experience, the student might examine different turbulence models and the effects of mesh type and refinement.

The module is assessed on a 'coursework only' basis with weightings attached to the four project reports in order to reflect the quantity and depth of work involved in each (respectively 10, 35, 15, and 40%). It is expected that the participants' reports will address relevant areas of theory and not simply present computational findings. Written feedback on the report is discussed individually with each participant within one to two weeks of submission and therefore plays a formative role as well as providing the basis for assessment.

The participants are encouraged to collaborate with one another in an interactive working environment (project reports, however, should reflect individual work). Expert advice, parallelling that from a more experienced colleague in industry, is provided by research postgraduates, themselves working in CFD, and the writer.

Learning Outcomes

The intended learning outcomes of the module cover both technical and broader aspects and may be listed under four headings:

- Knowledge and Understanding - The student should develop a good understanding of the physical concepts underpinning turbulence models in general, and the Standard and Realizable k-e models in particular. Some reference is made to more advanced turbulence models and to the numerical aspects of CFD such as discretization, the generation and refinement of the computational grid, and the establishment of criteria for the determination of an adequately converged flow solution.
- Intellectual Abilities – The essential intellectual step is the ability to abstract an engineering design or application to an appropriate CFD representation. The CFD Engineer must plan and execute a solution strategy and then evaluate the resulting solution in the light of any available experimental data and in consideration of the modelling approximations employed.
- Practical Skills – Practical skills are centred on the development of competence in the use of advanced software packages. The participant must develop familiarity with the Gambit pre-processor and the Fluent 5 solver and post-processor.
- General Transferable Skills – These fall into three groups. Students must develop interpersonal skills in terms of the ability to collaborate with colleagues in a 'real world' work/study environment, develop the ability to access practical information from on-line technical manuals, and form a critical appreciation of subject matter based upon information gathered from library and other sources.

Evaluation in relation to Learning Outcomes

The four headings covering learning outcomes have been mapped on to the required coverage of the project reports on which the module is assessed. In 2000/01 the average overall mark achieved by the 14 participants was 63%, with 3 awarded marks at a 1st Class level, 9 at 2(i) level, 1 at 2(ii) level, and 1 at 3rd Class level. In 2001/02 the average mark of a further 14 participants was 66%; 4 students achieved 1st Class marks, 7 a 2(i) mark, and 3 a 2(ii) mark. Such figures indicate a satisfactory level of achievement of the required learning outcomes. Commenting anecdotally, Level 4 participants respond very positively to the module, and a high level of 'ownership' of the software and supporting material is evident.

It is hoped that the CFD module makes a wider contribution to the effectiveness with which the School of Engineering delivers relevant modern engineering content. The module quite naturally builds upon Level 1-3 modules in the area of fluid flow and is designed to provide a suitable 'launch pad' for those students wishing to specialize in the field after graduation.

Transferability of the Approach

A number of factors might be identified to indicate the transferability of the approach described here to other areas of engineering (and also possibly more widely). An essential feature is that the module is computer based, using a sophisticated software package that has widespread industrial take-up. A further defining aspect is the high level of specialist content. Largely following from those characteristics, the methods adopted are seen as being appropriate to Level 3 and 4 teaching – considerable demands are made upon the student in terms of intellectual and personal maturity. In view of the general nature of the module, and also its emphasis on rapid turn-around of coursework, it is considered to be best suited to small group teaching.

SIMULATION VERSUS REALITY

Fluid flows in both the natural environment and engineering systems are almost always turbulent, exhibiting a complex eddying motion that has both ordered and random characteristics. While module activities are centred on practical projects, a limited number of seminars is held to provide a Level 4 background in turbulent flow theory. The engineering computation of turbulent flows is frequently undertaken using an 'industry standard' mathematical model of turbulence known as the 'k-e' model. In the present module both 'Standard' and 'Realizable' variants of the k-e scheme are examined.

The Standard and Realizable k-e Eddy Viscosity Models

Practical methods for the calculation of turbulent flows are based upon a statistical approach. The process of time-averaging (or phase- or ensemble-averaging for unsteady flows) gives rise to unknown Reynolds stresses – this introduction of further unknowns without additional equations is termed the problem of 'closure', and it is the function of a turbulence model to supply values of the Reynolds stresses. The k-e group of turbulence models forms part of a broader class of 'Eddy Viscosity Models' (EVMs) in which the Reynolds stresses are related to gradients of the mean velocity via a scalar eddy, or turbulent, viscosity, $?_t$. Restricting attention to incompressible flows, the general form of the constitutive equation for all linear EVMs is given by

$$-\overline{u_i u_j} = n_t \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \frac{2}{3} d_{ij}\, k \qquad (1)$$

Cartesian tensor notation has been used in writing (1). The subscripts i and j may independently take the values 1, 2, or 3; x_i represents the coordinate direction (x_1 = x, x_2 = y, and x_3 = z), U_i is a mean velocity component and u_i the corresponding fluctuating component (thus instantaneous velocity in the i-direction is given by the sum of U_i and u_i). The term on the left of (1) represents any one of the six independent Reynolds stresses that exist in a general strain field. The overbar indicates a long-time average, appropriate to a flow that is 'statistically stationary', i.e. steady in the mean. The Kronecker delta, d_{ij} takes the value of 1 if i = j; otherwise it is zero (its introduction in (1) ensures that k is correctly equal to half the sum of the 3 normal stresses, $k = \frac{1}{2}(\overline{u_1^2} + \overline{u_2^2} + \overline{u_3^2})$).

In both the Standard and Realizable k-e models $?_t$ is prescribed as

$$n_t = C_m \frac{k^2}{e} \qquad (2)$$

Thus if k, e, and C_μ are known the problem of closure is resolved since (1) and (2) together now permit the determination of the Reynolds stresses in terms of the mean flow field. (This remark is made in a strictly mathematical sense; nothing is implied at this stage concerning the accuracy of either model.) In both models k and e are determined from additional transport equations, a feature that renders k-e models considerably more adaptable to complicating features in a flow than, say, the simple 'mixing length' EVM which relates the Reynolds stresses to the mean field using a fixed algebraic prescription of the turbulence length scale. A detailed discussion of the k- and e-equations is, however, beyond the scope of the present work and instead we focus upon the dimensionless quantity C_μ appearing in (2).

In the Standard k-e model [7] the value of C_μ is determined by reference to the 'logarithmic' region of wall-bounded flows where the ratio of the Reynolds shear stress $-\overline{u_1 u_2}$ to k is approximately uniform and the production and dissipation rates of turbulent kinetic energy are essentially in balance. Under such conditions it may be shown that C_μ is a constant equal to 0.09. In the Realizable model [8, 9], by contrast, C_μ is generalized to be a function of k, e, and the mean strain field. In the following sub-section we employ physical arguments (as developed in the CFD module) to explore the advantages of allowing C_μ to vary in response to local flow conditions.

Further Comments on EVMs

In order to simplify the following discussion consider an attached boundary layer flow in which the only element of the Reynolds stress tensor active in the mean flow equations is the shear stress $-\overline{uv}$ (or $-\overline{u_1 u_2}$). The dominant strain rate is formed by the gradient of the mean velocity in the principal flow direction with respect to the cross-stream (wall-normal) coordinate and therefore (1) and (2) supply the Reynolds shear stress as

$$-\overline{uv} = C_m \frac{k^2}{e}\frac{\partial U}{\partial y} \tag{3}$$

It is instructive to re-examine the k-e EVM in ‘structural’ terms (i.e. using dimensionless ratios of turbulence and mean field quantities). A simple manipilation of (3) yields

$$-\frac{\overline{uv}}{k} = C_m \frac{k}{e}\frac{\partial U}{\partial y} \tag{4}$$

Thus the k-e EVM may be viewed as postulating a linear dependence of the structural ratio $-\overline{uv}/k$ on the group $(k/\varepsilon)\partial U/\partial y$. Dimensional analysis immediately suggests a generalization of (4):

$$-\frac{\overline{uv}}{k} = g\left\{\frac{k}{e}\frac{\partial U}{\partial y}\right\} \tag{5}$$

or, in terms of a ‘damping function’ that may be introduced to the right hand side of (3),

$$-\overline{uv} = C_m\, f\left\{\frac{k}{e}\frac{\partial U}{\partial y}\right\}\frac{k^2}{e}\frac{\partial U}{\partial y} \tag{6}$$

f and g are functions related as

$$g = \left(\frac{k}{e}\frac{\partial U}{\partial y}\right)\times C_m\, f \tag{7}$$

The damping function $f\{(k/\varepsilon)\partial U/\partial y\}$ appearing in (6) modifies the standard constitutive equation in regions of a flow where the argument is large (here f is less than 1); in other regions f asymptotes to unity and the conventional form of (3) is recovered. (It is also possible for f to assume values greater than 1.) The ratio of k to e represents the time scale of large-scale turbulence; (?U/?y)$^{-1}$ is the time scale of mean field straining, and therefore (k/e) divided by (?U/?y)$^{-1}$, i.e. (k/e)?U/?y, provides a measure of the ability of turbulence to respond to changing mean conditions. An interpretation of the derivation of the conventional constitutive equation is that various structural parameters are implicitly assumed to be fixed constants, for example (k/e)?U/?y takes its ‘equilibrium’ value of $C_\mu^{-1/2} \approx 3.3$ (cf. the discussion of mixing length models in [10]). Strong support for the role of the group in characterizing shear-driven turbulent flows is provided by the Direct Numerical Simulation studies of [11]. (DNS involves the numerical solution of the exact instantaneous Navier-Stokes equations. In that sense it may be regarded as ‘real turbulence on a computer’.)

Extension to Rapidly-Varying Flows

A somewhat different concept was pursued in [12] and [13]. While related to the idea that (k/e)?U/?y may be viewed as the ratio of turbulence and mean strain time scales, the group was instead interpreted as total strain (as opposed to strain *rate*) truncated on the large-scale turbulence time scale. Informed by the results of a formal approximation to the Navier-Stokes equations known as ‘Rapid Distortion Theory’ [14] an ad hoc equation for homogeneous flow was advanced in [13]:

$$\frac{\partial b}{\partial t} = \frac{\partial U}{\partial y} - \frac{b}{T} \tag{8}$$

Thus, with initial condition ß = 0 at t = 0, and for constant ?U/?y and relaxation time scale T:

$$b = T\frac{\partial U}{\partial y}[1 - exp(-t/T)] \tag{9}$$

Approximations to (9) at short (t/T = 1) and longer times (t/T = 1) are t.?U/?y (actual total strain) and T.?U/?y (truncated total strain), respectively. If T is equated to the large-scale turbulence time scale the final expression may be re-written as (k/e)?U/?y. The parameter ß may be used as the argument of a damping function in a manner similar to (6). The difference now, however, is that the ‘memory’ of turbulence for past straining events is introduced in the formulation.

The proposals of [12] and [13] were developed in [15] in order to produce a model applicable to inhomogeneous as well as homogeneous flows. In place of (8) there is substituted a second ad hoc equation for a ‘strain parameter’, S:

$$\frac{DS}{Dt} = \frac{k}{e}\left(\frac{\partial U}{\partial y}\right)^2 + \frac{\partial}{\partial y}\left(\frac{n_t}{s_S}\frac{\partial S}{\partial y}\right) - \frac{S}{(k/e)} \qquad (10)$$

where s_S is a turbulent Prandtl number for the diffusion of S; the diffusive term is included following [12] in consideration of the fact that fluid elements arriving at a given point will experience different individual strain histories. The term mimics the effect of differing flow realizations within a time- or ensemble-averaged turbulence closure. An equation with mathematical properties similar to (10) was advanced in [16] as an auxiliary equation for an inverse time scale. While those authors were concerned not with a strain parameter but with the development of an improved representation of the dissipative processes, it was noted in [16] that the solution of equations similar to (10) takes the form of an integral back along a mean streamline to a given point. The zone of influence is wider in regions further removed from the point in question and the integral is weighted to have a receding memory of more distant events. Returning to the special case of homogeneous flow, we note that the long-time (t/(k/e) = 1) solution of (10) is S = ((k/e)?U/?y)^2, i.e. the square of the corresponding solution of (8) (where k/e has been assumed to be constant for simplicity). The model developed in [15] has been applied in [17] to flows characteristic of those occurring in the post-trip operation of nuclear reactors.

CONCLUDING REMARKS

The present paper has described the writer's experience in connection with a new CFD module for Level 4 engineering students. The process has been two-way and much has been learnt from the participants in the course of numerous exchanges.

The first part of the paper was concerned with various aspects of module structure and the creation of a learning environment akin to that of a professional development course. The 'taught' seminar material outlined in the second part of the paper was included to help students develop an improved physical insight into the likely fundamental strengths and weaknesses of turbulence models of the k-e type. Given a sound grasp of these basics, participants would then possess a grounding in turbulence theory that would enable them to develop a fuller understanding of more advanced elements of turbulence modelling methodology.

The module is research-led, turbulence modelling forming the writer's principal research interest. Research students working in CFD have made a valuable contribution to the module and, once again, the work of the Level 4 participants has made some input into the research students' own activities. In the medium term it is envisaged that the overall design of the module will remain similar to the present structure. It is, however, planned to introduce some changes in terms of the detailed content of the four CFD projects that constitute the assessed material of the module.

ACKNOWLEDGEMENT

The writer would like to record his gratitude to Mr. K. R. Kim and Mr. D. Reedha, research postgraduates who have contributed to the delivery of the module. Thanks are also due to the participants over the past two years who have made valuable comments that will inform future developments in CFD teaching.

REFERENCES

[1] Fluent Inc., "GAMBIT User's Guide", 1998.

[2] Fluent Inc., "FLUENT 5 User's Guide", Vols. 1-4, 1998.

[3] Fluent Inc., "GAMBIT Tutorial Guide", 1998.

[4] Fluent Inc., "FLUENT 5 Tutorial Guide", Vol. 1, 1998.

[5] Hussein, H.J., Capp, S.P. & George, W.K., "Velocity measurements in a high-Reynolds-number, momentum-conserving, axisymmetric, turbulent jet", *Journal of Fluid Mechanics,* Vol. 258, 1994, pp. 31-75.

[6] AGARD (Advisory Group for Aerospace Research and Development), "A selection of test cases for the validation of large-eddy simulations of turbulent flows", *AGARD Advisory Report 345,* 1998.

[7] Launder, B.E. & Spalding, D.B., "The numerical computation of turbulent flows", *Computer Methods in Applied Mechanics and Engineering,* Vol. 3, 1974, pp. 269-289.

[8] Reynolds, W.C., "Fundamentals of turbulence for turbulence modeling and simulation", *Lecture Notes for Von Kármán Institute, AGARD Report 755,* 1987.

[9] Shih, T.-H., Liou, W.W., Shabbir, A., Yang, Z. & Zhu, J., "A new k-e eddy viscosity model for high Reynolds number turbulent flows", *Computers and Fluids,* Vol. 24, 1995, pp. 227-238.

[10] Tennekes, H. & Lumley, J.L., "A First Course in Turbulence", MIT Press, 1972.

[11] Lee, M.J., Kim, J. & Moin, P., "Structure of turbulence at high shear rate", *Journal of Fluid Mechanics,* Vol. 216, 1990, pp. 561-583.

[12] Townsend, A.A., "Entrainment and the structure of turbulent flow", *Journal of Fluid Mechanics,* Vol. 41, 1970, pp. 13-46.

[13] Maxey, M.R., "Distortion of turbulence in flows with parallel streamlines", *Journal of Fluid Mechanics,* Vol. 124, 1982, pp. 261-282.

[14] Hunt, J.C.R. & Carruthers, D.J., "Rapid distortion theory and the 'problems' of turbulence", *Journal of Fluid Mechanics,* Vol. 212, 1990, pp. 497-532.

[15] Cotton, M.A. & Ismael, J.O., "A strain parameter turbulence model and its application to homogeneous and thin shear flows", *International Journal of Heat and Fluid Flow,* Vol. 19, 1998, pp. 326-337.

[16] Lumley, J.L., Berkooz, G., Elezgaray, J., Holmes, P., Poje, A. & Volte, C., "Fundamental aspects of incompressible and compressible turbulent flows", in *Simulation and Modeling of Turbulent Flows* (Eds. Gatski, T.B., Hussaini, M.Y. & Lumley, J.L.), Oxford University Press, 1996, pp. 5-78.

[17] Cotton, M.A., Ismael, J.O. & Kirwin, P.J., "Computations of post-trip reactor core thermal hydraulics using a strain parameter turbulence model", *Nuclear Engineering and Design,* Vol. 208, 2001, pp. 51-66.