A COMPARATIVE STUDY ON SOLUTION METHODS IN EXTENDED NEAR WALL ZONE OF TURBULENT FLOW

Sabah Tamimi
Al Ghurair University
UAE

ABSTRACT

In present work, a finite element based solution technique for general steady-state two-dimensional, incompressible confined turbulent flow in long parallel-sided channels with a one-equation model to depict the viscosity of turbulence has been developed and adopted. Particular attention is given to the important aspect of studying the flow behaviour in the 'near wall zone' of confined flow. It is particularly important that the transfer of mass and momentum within this zone is modelled accurately in order to obtain correct overall predictions. In this paper, two different methods of solutions have been tested when the near wall zone has been extended away from the solid wall.

Keywords: Turbulent flow; Pressure flow; Coupled and iterative methods; Extended near wall zone; Finite element method.

INTRODUCTION

Due to the growth of technology, the complexity of the Navier-Stocks equations which represent the fluid motion and the impossibility for an analytical solution to these equations, as a result of this much attention has been focused on computational fluid dynamics (CFD) for solving the resulting set of nonlinear partial equations which dominate the flow behavior processes. Numerous theoretical and experimental works are available on laminar flow [1-2], but this is not the case of turbulent flow. Since it has not been possible to obtain exact analytical solutions to such flows, an accurate numerical approach would be very beneficial to researchers. The finite element method (FEM) is one of these methods that have recently emerged as a powerful tool for solving the N-S equations. Within the computational domain (i.e. main domain), the finite element method is used to discretise the equations governing the fluid motion.

It is known that the values of the pertinent variables change from some initial profile to a fully developed form, which is thereafter invariant in the downstream direction when a fluid enters a prismoidal duct. The analysis of this region is known as developing region, which has been the subject of extensive studies.

An effective technique with significant grid refinement is required to model the variation of the pertinent variables near a solid boundary, where the variation in velocity and kinetic energy, in particular, is extremely large near such surfaces since the transfer of shear form the boundary into the main domain and the nature of the flow changes rapidly. Several solution techniques have been suggested in order to avoid such excessive refinement [3-5]. A more common approach is to terminate the actual domain subject to discretisation (main domain) at some small distance away from the wall, where the gradients of the independent variables are relatively small, and then a technique is a required to model the flow behaviour in the near wall element. In previous work different techniques were used. One of them was the use of an element technique, which is based on the use of the one dimensional finite element technique in one direction normal to the solid wall. The validity of the wall element technique has been tested and proved for developing and fully developed turbulent flow [6-7] when the near wall zone located at limited distance from the solid wall. Presently, the validity of this technique has been tested for developing and fully developed flow at extended near wall zone away from the wall using two different methods of solutions to simulate the turbulent flow in a smooth straight channel.

Mathematical Modelling

Steady - state incompressible two dimensional turbulent flow of a Newtonian viscous fluid with no body forces acting has been investigated and therefore the Navier-Stokes (N-S) equations associated with this type are,

$$\rho u_j \frac{\partial u_i}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu_e\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)\right] \text{ ……... (1)}$$

Where i,j= 1,2. u_i, p are the time - averaged velocities and pressure respectively, ρ is the fluid density, μ_e is the effective viscosity which is given by $\mu_e = \mu + \mu_t$, μ and μ_t are the molecular viscosity and turbulent viscosity, respectively. The flow field must satisfy the continuity equation, which may be written as:

$$\frac{\partial u_i}{\partial x_i} = 0 \quad \text{…………….………………………………. (2)}$$

Equation (1) and (2) cannot be solved unless a turbulence closure model can be provided to evaluate the turbulent contribution to μ_e. In the present work, a one equation model has been adopted so that,

$$\mu_t = C_\mu \rho k^{1/2} 1_\mu \quad \text{………………........................... (3)}$$

1_μ is the length scale of turbulence which is given by $1_\mu = 2.5\ 1_m$, 1_m is the mixing length based on the Prandtl hypothesis which has been specified algebraically for the present purposes as 0.4 times the normal distance from the nearest wall surface, C_μ is a constant and k is the time-averaged turbulence kinetic energy. The μ_t given by equation (3) requires that k to be known. This can be evaluated via a further transport equation given by:

$$\rho u_j \frac{\partial k}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + \mu_t \frac{\partial u_i}{\partial x_j}\left[\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right] - E \text{ .. (4)}$$

Where E= $C_D \rho k^{3/2} / 1_\mu$, μ_t / σ_k is the turbulent diffusion coefficient, σ_k is the turbulent Prandtl or Schmidt number and C_D is a constant.
The turbulence model based on equations (1-2) and (4) are called the one-equation (k-l) model. The above governing equations have been solved using a finite element method [8].

Within the near wall zone either universal laws concept [9] or one dimensional parabolic element in a direction normal to solid wall is adopted. Within the main domain, conventional two dimensional isoparametric elements domain two dimensional elements up to the wall are used to discretise the flow domain.

BOUNDARY CONDITIONS

Constant values are assumed on all variables upstream was considered for developing flow and compatible fully developed velocity and kinetic energy profiles which looks like

ISSN 2056-5860

parabolic curve were imposed at the upstream section when fully developed pressure turbulent flow was considered at the first stage and the traction's were updated at downstream. In both cases no slip conditions were imposed on solid walls and tractions updated downstream as shown in Fig. 1. Tractions are given by,

$$\tau_{x_1} = -p + \frac{\mu_e}{\rho}\left(\frac{\partial u_1}{\partial x_1}\right) \qquad x_1\text{- parallel to walls}$$

$$\tau_{x_2} = \frac{\mu_e}{\rho}\left(\frac{\partial u_2}{\partial x_1} + \frac{\partial u_1}{\partial x_2}\right) \qquad x_2\text{- normal to walls}$$

Within the near wall zone universal laws or conventional finite elements (i.e. 2-D elements up to the wall) were used. In the present work, a wall element technique based on finite elements method has been adopted, using one-dimensional (3-noded elements) normal to the wall as shown in Fig. 2.

SOLUTION PROCEDURE

In the present work, two classes of solution methods have been used to solve the systems of algebraic equations. One is called coupled method (direct method). In this method, the matrix equations relating to the main domain and the near wall zone are combined together in one matrix and subsequently solved. The other method is an uncoupled method in which the matrix equations for the main domain and the near wall zone are solved separately.

In coupled method, the equations from the one dimensional elements normal to the wall in the N.W.Z. and the two dimensional elements in the main are assembled together to form one matrix. One-dimensional element numbering is continuous with those in the computational domain, such that more elements could be placed where higher gradients exist. Care was taken when assembling the equations to ensure that the same degree of freedom was being used at common nodes.

In uncoupled method, the interfacial matching between each domain is affected in a similar manner to that used when unidimensional elements normal to the wall (one-dimensional elements) is employed in the N.W.Z. and conventional two-dimensional elements employed in the main computational domain. At the interface, each nodal point associated with elements in the main domain corresponds to a set of unidimensional parabolic elements in the N.W.Z.

RESULTS AND DISCUSSION

Finer meshes distribution were used such that with further refinement no increase in accuracy was apparent when a parallel-sided duct of width D, which is taken as 1.0 in the present work, and the length L. Reynolds number based upon the width of the channel of 12.000 was considered when pressure flow was considered only.

All the results obtained previously [6-7] were when the mesh was terminated at 0.49D from the centre line (symmetric line). In the present work, the interface was located at 0.48D and 0.47D from the symmetric line, with values of Y^+=24 and Y^+=31, respectively. In this paper, two stages have been considered and in both two different methods of solutions (coupled and uncoupled) were used with extension of near wall zone further into main domain.

In the first stage the validity of the technique when developing turbulent flow was considered. A finer discretisation of the geometry consist of 56 elements, 199 nodes and the channel length was taken and 1-D elements normal to the wall are employed in near wall zone.

Figures 3 and 4 shows Developing velocity profiles for turbulent flow when the N.W.Z. is extended up 0.47D , at 10D downstream, L=10D, Re=12.000. Clearly, Fig. 4, the velocity plots became smoother when all element types are embodied within one over matrix, especially when the mesh is terminated at 0.47D as compared to the curve in Fig. 3 when an uncoupled method based on an iterative technique was employed. Fig. 5 and Fig. 6 present downstream kinetic energy profiles for turbulent flow, at 10D downstream, L=10D, Re=12.000, at interface 0.47D. Fig. 6 confirmed that the use of coupled method gave smoother results when the N.W.Z. was extended up to 0.47D and corresponding to Y^{+}=31 at this location.

The next stage was concerned again with the validation of the adopted wall element technique when fully developed turbulent flow was considered in an extended near wall zone using both coupled and uncoupled methods of solutions. Figures 7 and 8 Shows fully developed velocity profiles and kinetic energy profiles for turbulent flow, at 1.4D downstream, L=1.4D, Re=12.000, when the N.W.Z. is extended up 0.47D.
Now, it is quite obvious that better and smoother results are obtained for fully developed flow when the wall element technique was adopted in the extended near wall zone as shown in figures 7 and 8 compared with those obtained in figures 3 and 5 when developing flow was considered.

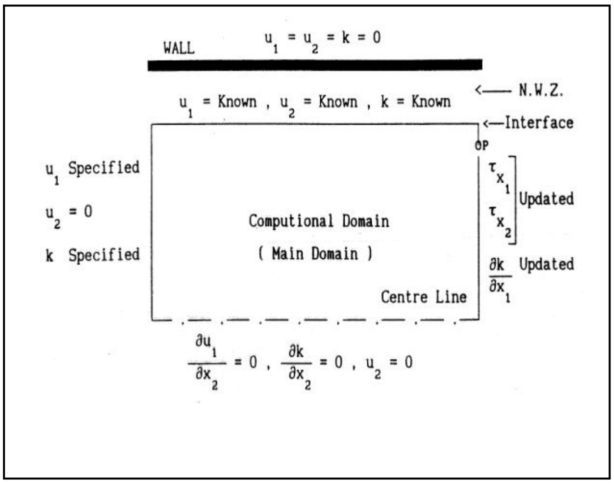


Fig.1 Boundary conditions when the mesh is terminated at small distance away from the wall N.W.Z.

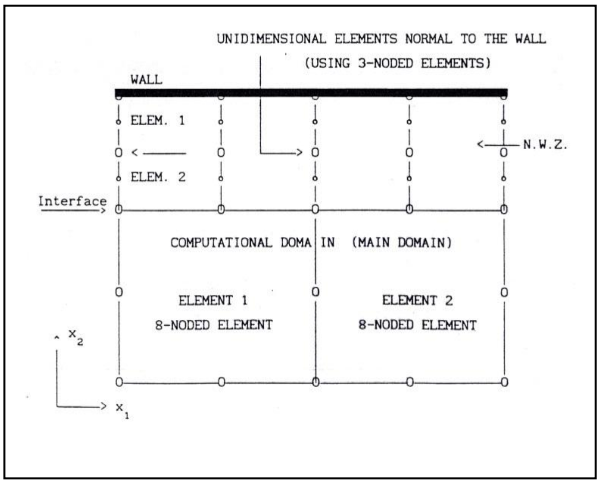


Fig. 2 One-dimensional elements in one-direction normal to the wall used in the

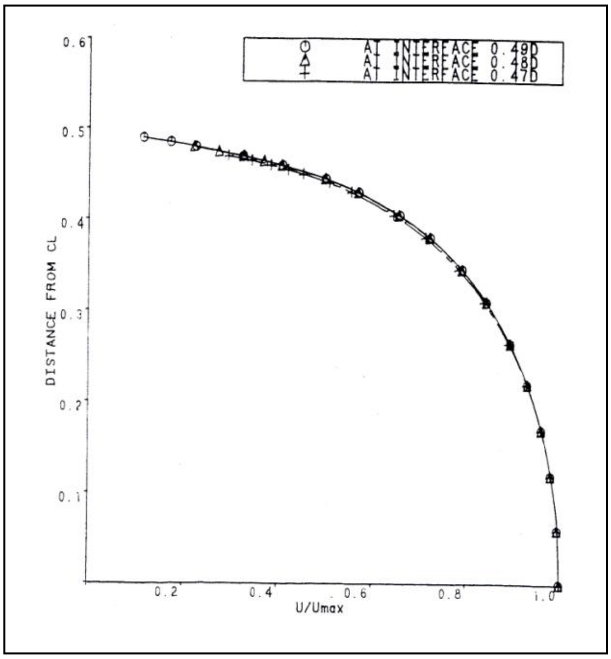


Fig. 3 Developing velocity profiles, equations equations solved using iterative technique

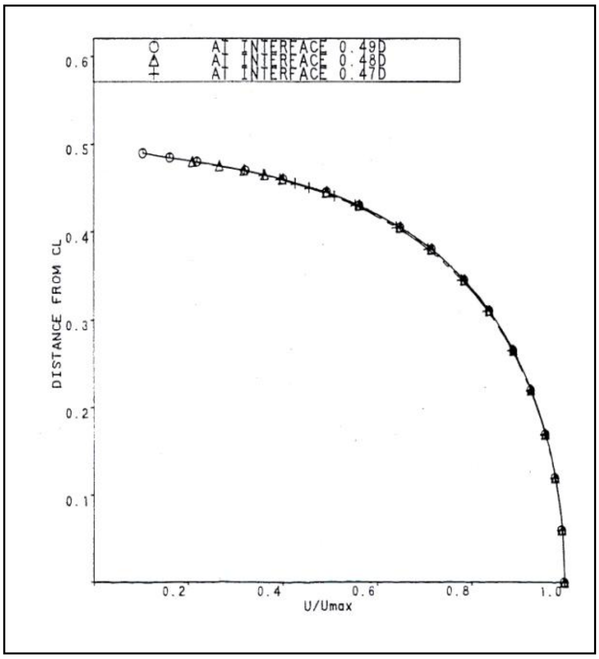


Fig. 4 Developing velocity profiles, solved in one matrix

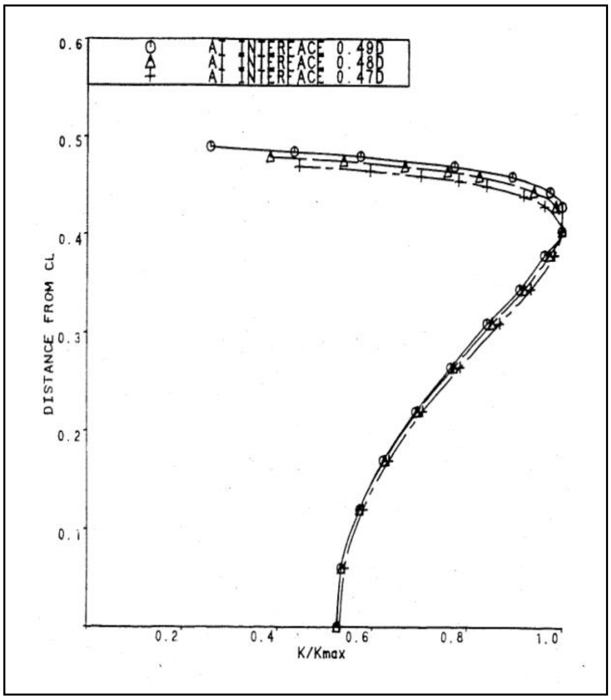


Fig. 5 Downstream kinetic energy profiles for developing turbulent flow using iterative technique

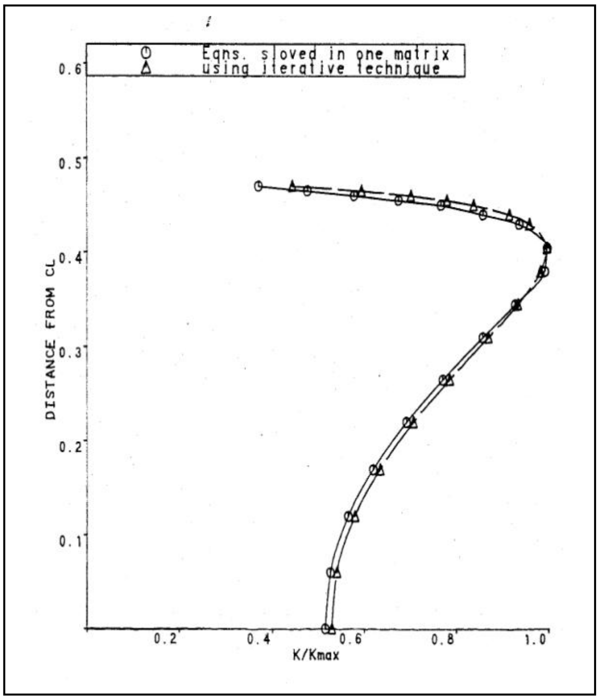
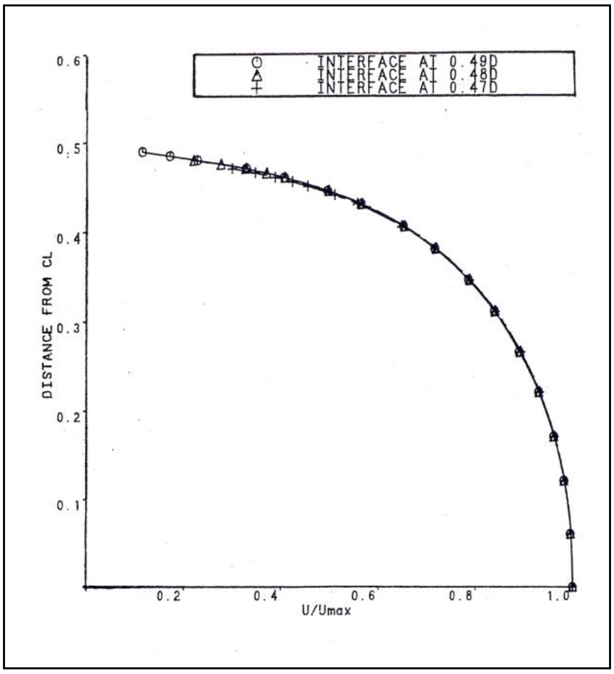
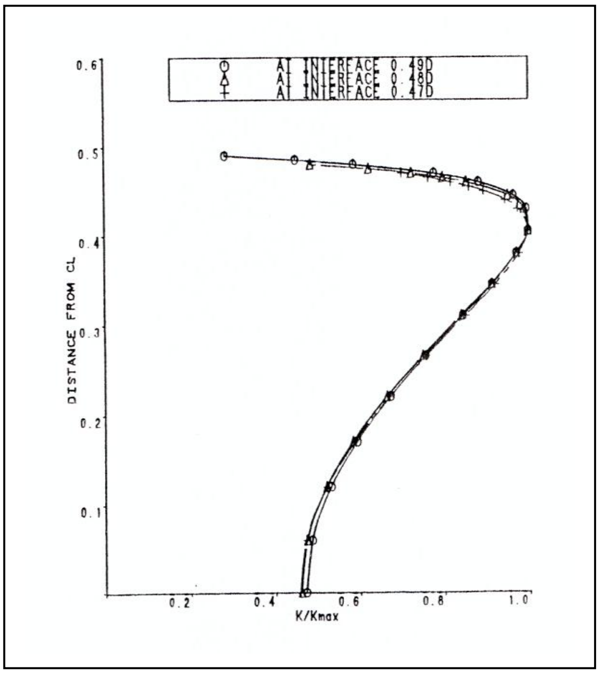


Fig. 6 Downstream kinetic energy profiles for developing turbulent flow when coupled and uncoupled methods used





CONCLUSION

The adopted wall element technique (1-D element in one direction normal to the wall) which is based on the use of the finite element methods has been applied successfully to developing and fully developed turbulent flow in extended near wall zone, and shows excellent results for fully developed flow, since each 1-D string of elements is analysed individually, this saved computer memory and time required. Also, the use of coupled method was better than the iterative method when the near wall zone was extended. This applies to both developing and fully developed flow. However, for low Y+ values the uncoupled method was still applicable.

REFERENCES

1. Mehrotra, A. K. and Patience. G. S.(1990) *Unified Entry Length for Newtonian and power law fluids in Laminar pipe flow, J. Chem. Eng*., 68, 529-533.
2. D.M. Hawken, H.R. Tamaddon-Jahromi, P. Townsend and M. F. Webster (1990) *A Taylor-Galerkin based algorithm for viscous incompressible flow*, Int. Journal Num. Meth. Fluids.
3. Graft T, Gerasimov A, Iacovides H, Launder B. (2002) *Progress in the generalization of wall-function treatments*, Int. Journal for heat and fluid flow. 148-160.
4. Haroutunian, and Engelman, S. (1991) *On modeling wall-bound turbulent flows using specialized near-wall finite elements and the standard k-ε turbulent model*. Advances in Num. simulation of Turbulent flows, ASME, 117, 97-105.
5. Launder, B. E. and Shima, N. (1989) *Second moment closure for near wall sublayer: Development and Application*, AIAA Journal, 27, 1319-1325.
6. Sabah Tamimi (2012) *An Analysis of Developing Turbulent Flow*, International Journal of Computer Science & Communication (IJCSC), ISSN: 0973-7391, Vol. 3, No.1, 11-13.
7. Sabah Tamimi (2012) *Validation of wall element technique of turbulent flow*, GSTF International Journal on Computing (JoC), Vol. 2, No. 1, 177-181.

8. Griffiths DV and Smith IM (2006) *Numerical Mtghods for Engineering*, Chapman and Hall/CRC 2nd Edition, New York.
9. Davies, T. J. (1972) *Turbulent phenomena*, Academic Press.

BIOGRAPHY

SABAH TAMIMI is an Associate Professor at the College of Engineering and Computing, Al Ghurair University, Dubai, UAE. He earned his M.Sc. (Computer Science, 1988) and Ph.D. (Applied Computer Science, 1992), both degrees from University of Wales, UK. He is working continuously as a full-time academic faculty for more than 24 years at university level. In addition, the last 11 years, he has been involved in administrative sector working as Deputy Dean and Dean as well as faculty. His interests are in computer modelling and simulation, computational flow dynamics, software testing techniques, computer graphics and databases. He has a very good number of publications in international journals and conferences, and reviewed many papers for international journals and conferences. Also, he is an editorial board member of different international journals.