Progress of Solution Methods Based Boundary Layers of Turbulent Flow

SABAH TAMIMI
College of Engineering & Computing,
Al Ghurair University,
Dubai, United Arab Emirates.
sabah@agu.ac.ae

Abstract: - In the present work, an effective technique with significant grid refinement has been developed and adopted to model the variation of the pertinent variables near a solid boundary using finite element method based solution 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. Particular attention is given to the important aspect of studying the flow behaviour in the 'near wall zone' of confined flow. Accurate model within the near wall zone is used to obtain correct overall predictions which resulted from the transfer of mass and momentum within this zone. As a result of this some innovative techniques regarding the mapping of the near wall zone were developed. In the present work different methods of solutions have been tested when the near wall zone has been extended away from the solid wall.

Key-Words: - Developing turbulent flow, different solution methods, extended near wall zone, FEM.

1 Introduction

It is known that an analytical solution for the Navier-Stocks (N-S) equations which represent the fluid motion is impossible this is due to the complexity of these equations which presented by set of nonlinear partial equations, as a result of this much attention has been focused on computational fluid dynamics (CFD) to solve these equations which dominate the flow behavior processes. Theoretical and experimental works are available on laminar flow [1-4], but this is not the case of turbulent flow. Since it has not been possible to obtain exact analytical solutions to such flows, therefore an accurate numerical approach would be very beneficial to researchers. The finite element method (FEM) is the most powerful method for solving the N-S equations which is used to discretise the equations governing the fluid motion in the main domain. 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 [5-7]. The analysis of this region is known as developing region, which has been the subject of extensive studies.

An effective technique is required to model the variation of the pertinent variables in the near wall zone, 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 [8-10]. 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 [11], different wall element techniques were used. One of them was the use of empirical universal laws technique which has been approved not be acceptable any longer for both developing and fully developed turbulent flow. The second technique which was used is the conventional finite element. In this technique the 2-D elements of the main domain was extended up to the wall and proved not to be good enough because it is not economically viable which needs an excessive refinement to extend the main domain. The final

technique which was used, is the adopted technique (wall element technique), by using one dimensional element in one direction normal to the solid wall in the near wall zone. The validity of this of technique has been tested and proved to be superior to pervious techniques for fully developed turbulent flow. However, the accuracy of this technique when used in one direction is clearly not valid for developing flow.
As a result of this, the adopted technique has been modified by using one dimensional element in two directions normal and parallel to the solid wall and proved to be superior for developing turbulent flow [12]. In all case which has been investigated the near wall zone was located at limit distance from the wall.

As a continuation of this work and to consider all possibilities, in the present work, the use of the one dimensional element in two directions normal and parallel to the solid wall has been adopted and tested for developing flow when the near wall zone (N.W.Z.) has been extended away from the solid wall and different methods of solutions have been used to simulate developing turbulent flow in a smooth straight channel.

2 Governing Equations

The Navier-Stokes equations associated with steady state incompressible two dimensional turbulent flow of a Newtonian viscous fluid with no body forces acting are,

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

Where i ,j= 1,2. u_i, p are the 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. The continuity equation can be written as:

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

Equation (1) and (2) cannot be solved unless a turbulence closure model can be provided to evaluate the turbulent contribution to μ_e. The simplest model is via an algebraic formula [13] which has limited application and therefore this model is not adopted in the present work, but an alternative (Prandtl [14]-Kolmogorov [15]) model is used in which,

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

1_μ is the length scale of turbulence which is given by 1_μ = 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:

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

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 [16]. The above governing equations have been solved using the standard finite element method [17-19] and Galerking weight residual approach is adopted to solve the discretized equations governing the fluid motion. Within the near wall zone either universal laws concept [20] or one dimensional parabolic element in a direction normal to solid wall is adopted. Within the main domain, conventional two dimensional isoparametric elements domain up to the wall are used to discretise the flow domain.

3 Boundary Conditions

Since developing flow was tested constant values were imposed at the upstream section for the velocity and kinetic energy profiles are assumed on all upstream variables was considered, and 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 either conventional finite elements (2-D elements up to the wall) as shown in Fig. 1, or universal laws concept were used as shown in Fig. 2. In the present work, a wall element technique based on finite elements method has been adopted, using one-dimensional (3-noded elements) normal and parallel to the wall as shown in Fig. 3.

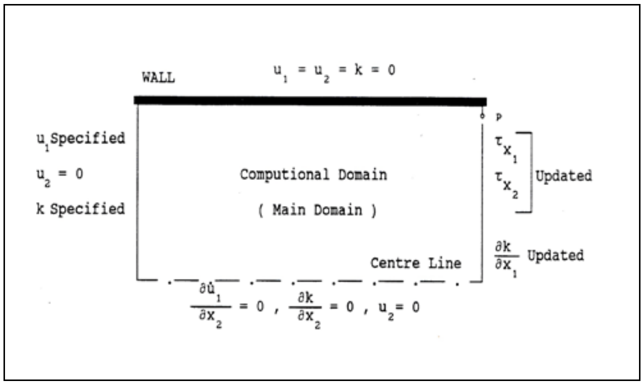

Fig.1 Boundary conditions for turbulent flow using two-dimensional elements up to the wall

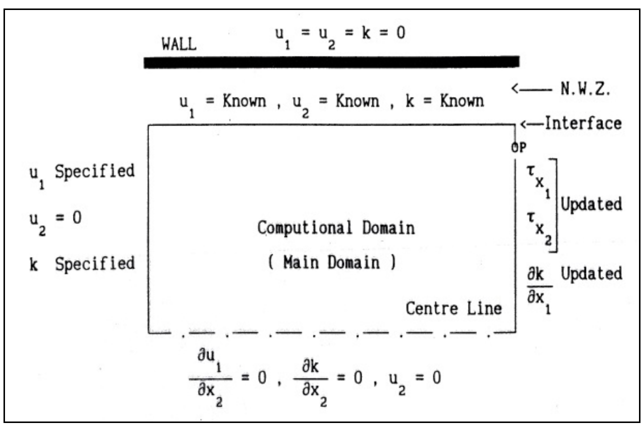

Fig. 2 Boundary conditions for turbulent flow using two-dimensional elements up to the wall

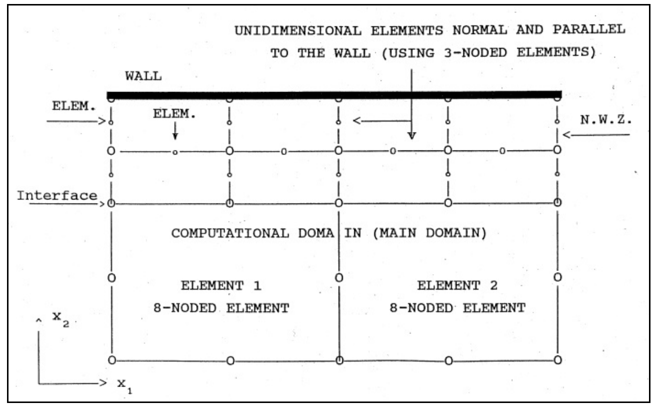

Fig. 3 One-dimensional elements in two-directions normal and parallel to the wall used in the N.W.Z

4 Results and Discussion

In the present work, two 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 coupled method, the matrix equations relating to the main domain and the near wall zone are combined together in one matrix and subsequently solved. So, the equations from the one dimensional elements normal and parallel 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 matrix equations for the main domain and the near wall zone are solved separately. So, the interfacial matching between each domain is affected in a similar manner to that used when unidimensional elements normal and parallel to the wall 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.

Finer meshes distribution were used consist of 56 elements, 199 nodes 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 (i.e. L=10D), and length L. The Reynolds number based upon the width of the channel of 12.000 when pressure flow was considered only.

All the results obtained previously [11-12] 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.

This stage was concerned with the validation of the adopted wall element technique when developing turbulent flow was considered in an extended near wall zone using both coupled and uncoupled methods of solutions.

The velocity profiles u_1 obtained in Fig. 4, proved the adopting of 1-D in two directions in the N.W.Z. still valid for turbulent developing flow when evaluating the longitudinal velocity in extended near wall zone.

The results obtained in Fig. 5, shows that 1-D elements in one direction are not acceptable for developing flow when the N.W.Z. is extended and 1-D in two directions is superior.

Fig. 6 present downstream kinetic energy profiles for developing turbulent flow, at 10D downstream, L=10D, Re=1.000, at interface 0.47D when an uncoupled method based on an iterative technique was employed.

Fig. 7 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 kinetic energy profiles presented in Fig. 6 indicate that the coupled matrix solution technique is more advantageous resulting in a better correlation as distance from the wall increases.

Although this was expected the extent of the variation has now been demonstrated. In addition, it appears that the coupled method of solution is superior to that when interfacial matching is used.

Another important observation is that the stringent limits placed on Y^+ when universal laws are used no longer apply when the advocated technique is used. This makes the solution of most problems very much easier and the like hood of having to re-mesh due to too large Y^+ values is considerably diminished.

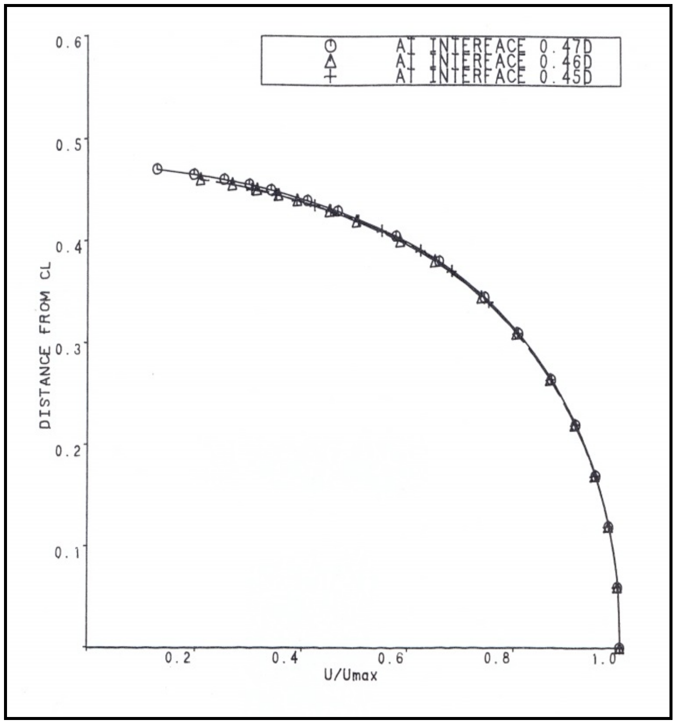

Fig. 4 Developing velocity profiles for turbulent at different flow at 10 downstream, using 1-D elements in two directions in extended near wall zone

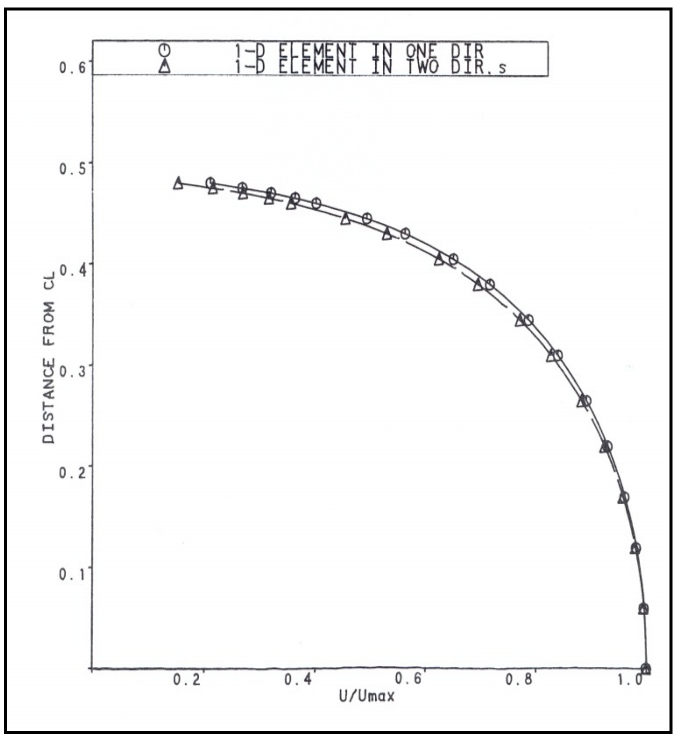

Fig. 5 Developing velocity profiles for turbulent, at 10D downstream, L=10D, at interface 0.48D, equations solved in one matrix.

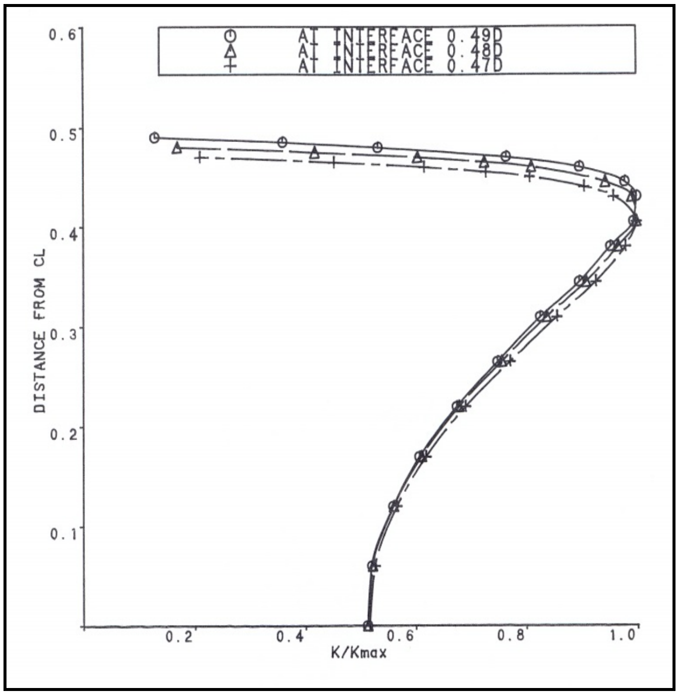

Fig.6 Turbulent kinetic energy profiles for developing flow, using 1-D in 2-directions at 10D downstream, L=10D. Equations solved using iterative technique

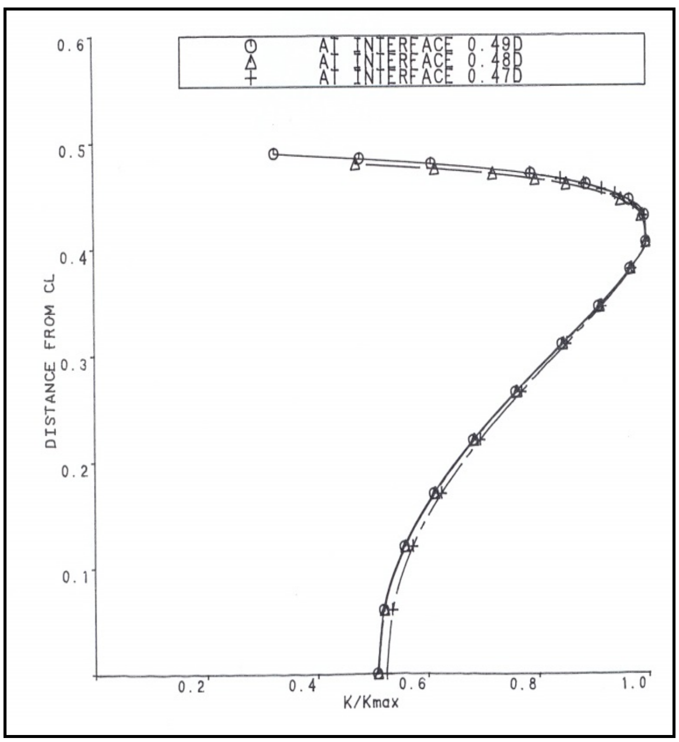

Fig. 7 Turbulent kinetic energy profiles for developing flow, Equations solved in one matrix using 1-D in 2-directions at 10D downstream, L=10D

5 Conclusions

It has been approved that, the use of empirical universal laws technique is not be acceptable any longer for both developing or fully developed turbulent flow, since these laws are only really applicable for certain unidimensional flow regimes. Also, the use of the conventional finite element when the 2-D elements of the main domain were the extended up to the wall proved not economically viable because it needs an excessive refinement to extend the main domain.

The accuracy of adopted wall element technique when used in one direction normal to the wall proved to be superior to pervious techniques for fully developed turbulent flow, since each 1-D string of elements is analyzed individually, this saved computer memory and time required. However, this is not the case for developing turbulent flow since the assumption of unidirectional flow is unacceptable. Whilst, the use of 1-D elements in two directions based on the use of the finite element methods has been has been applied successfully for developing turbulent flow in extended near wall zone, and shows excellent results. 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] A.K. Mehrotra, and G.S. Patience, Unified Entry Length for Newtonian and Power Law Fluids in Laminar Pipe Flow, *J. Chem. Eng.*, Vol. 68, 1990, pp.529-533.

[2] Wiginton, C. L. and Dalton, C.,Incompressible laminar flow in the entrance region of a rectangular duct, *J. Apple. Mechanics, vol. 37*, 1970, pp. 854-856

[3] Garg, K. Vijay, Developing flow in a rectangular duct, *Proc. Int. Conf. Numerical in Laminar and Turbulent Flow, Seattle*, 1983, PP. 58-68

[4] D.M. Hawken, H.R. Tamaddon-Jahromi, P. Townsend and M. F. Webster, a Taylor-Galerkin based algorithm for viscous incompressible flow, *Int. Journal Num. Meth. Fluids*, 1990

[5] B. J. McKeon, J. Li, W. Jiang, J. F. Morrison, and A. J. Smits, Further observations on the mean velocity distribution in fully developed pipe flow, *J. Fluid Mech*., 501_135, 1999.

[6] E.S. Zanoun, F. Durst, and H. Nagib, Evaluating the law of the wall in two-dimensional fully developed turbulent channel flows, *Phys. Fluids 10, 1*, 2003

[7] X. Wu and P. Moin, A direct numerical simulation study on the mean velocity characteristics in turbulent pipe flow, *J. Fluid Mech. 608, 81*, 2008.

[8] Graft T, Gerasimov A, Iacovides H, Launder B., Progress in the generalization of wall-function treatments, *Int. Journal for heat and fluid flow*, 2002, P. 148-160.

[9] Haroutunian, and Engelman, S., 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*, Vol. 117, 1991, pp. 97-105.

[10] Launder, B. E. and Shima, N.,Second moment closure for near wall sublayer: Development and Application, *AIAA Journal, vol. 27, 1989*, pp. 1319-1325.

[11] Sabah Tamimi, A Study on Computational of Flow Modeling, *the 4th IAJC/ISAM Joint Int. Conference, Orlando, Florida, USA*, ISBN: 978-1-60643-379-9, 2014, paper # 55.

[12] Sabah Tamimi, Boundary Layers Modeling for Developing Turbulent Flow, *the 11th International Conference on Fluid Mechanics & Aerodynamics (FMA '13), Athens, Greece*, ISBN: 978-1-61804-183-8, 2013, pp. 41-45.

[13] Launder, B. E. and Spalding, D. B, Lectures in mathematical models of turbulence. *Academic Press*, 1972

[14] Prandtl, L., Uber ein neues forelsystem fur die ausgebildete turbulenze, *Nachr. Akad. Der wissenschafft , Gottingesn*, 1945.

[15] Kolmogrov, A. H., Equations of turbulent motion of an incompressible fluid, IZV. *Akad Nauk, SSSR Ser. Phys*, 1942, Vol. 1-2, pp. 56-58.

[16] Taylor, C., Hughes, T. G. and Morgan, K., A finite element model of one and two equation models of turbulent flow, *Recent Advances in Numerical Methods in Fluid*, Vol. 1, Pineridge press, 1980, pp. 311-344.

[17] Taylor, C. and Hughes, T. G., Finite element programming of the Navier-Stokes equation, Pineridge press, 1981.

[18] Zienkiewicz, O. C. and Taylor, *R. L., The finite element method*, *McGraw-Hill, 1*988.

[19] Griffiths DV and Smith IM, Numerical Methods for Engineering , *Chapman and Hall/CRC, 2nd Edition*, *New York*, 2006

[20] Davies, T. J., Turbulent phenomena, *Academic Press*, 1972