Validation of Wall Element Technique of Turbulent Flow

Sabah Tamimi, College of Computing, Al Ghurair University, Dubai, United Arab Emirates**.,** sabah@agu.ac.ae

***Abstract-* An investigation based a finite element method (FEM) is carried out for the determination of confined turbulent flow with a one equation model used to depict the turbulent viscosity in a long straight channel. The effects of such flow in a zone close to a solid boundary has been investigated, by developing a finite element modeling technique based on near wall zone in which replaces the traditional techniques. The validity of the imposed technique is shown by comparison with other techniques.**

***Index Terms-* FEM, incompressible turbulent flow, pressure flow, wall element technique.**

I. INTRODUCTION

Fluid dynamics is one, among many important topics in applied computer science as well as engineering. Applications of fluid dynamics are continuing to grow as this advanced technology takes advantage of the increasing speed of computers and hardware capabilities and therefore, computational fluid dynamics (CFD), has became the interest of many researches. This development of CFD can offer a cost-effective to many problems one of them is the turbulent flow.

It is well known that when a fluid enters a prismoidal duct the values of the pertinent variables change from some initial profile to a fully developed form, which is thereafter invariant in the downstream direction. The analysis of this region, which is known as developing region, has been the subject of extensive studies. Numerous theoretical and experimental works are available on laminar flow [1-3], 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 [4] is one of these methods that have recently emerged as a powerful tool for solving the N-S equations. Within the main domain, the finite element method used to discretise the equations governing the fluid motion. A factor of consideration if that when using a numerical approach to analyze confined turbulent flow, an effective technique 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.

Consequently, if a conversational finite element subdomain is used to model the near wall zone (N.W.Z.), a significant grid refinement would be required. Indeed, in most situations this would be so fine as to be impractical.

Several solution techniques have been suggested in order to avoid such excessive refinement [5-7]. A more common approach is to terminate the actual domain subject to discretisation (main domain) at some small distance away form the wall, where the gradients of the independent variables are relatively small, and then use another technique to model the flow behavior in the near wall element. In this paper, a different near wall zone modeling techniques is used to simulate turbulent flow in a smooth straight channel.

Within the computational domain, the finite element method is used to discretise the equations governing fluid motion, namely continuity and the Navier-Stokes (N-S) equation. For present purposes the analysis is approached via a time averaged form of N-S equation with a spatially varying viscosity. For closure additional equations are required in order to evaluate local values of the turbulence kinetic energy, k, and Prandtl's mixing length. The additional equations must, therefore, depict the variation of both k and the mixing length. In the present work, the one equation model of turbulence is used, and a transport equation is derived which can be used to evaluate k.

II. MATHEMATICAL DESCRIPTION

The current investigation relates to steady - state incompressible two dimensional turbulent flow of a Newtonian viscous fluid with no body forces acting. For such a situation, the Navier-Stokes (N-S) equations associated with this type are,

DOI: 10.5176_2010-2283_2.1.147

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

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:

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

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 [8] which has limited application and therefore this model is not adopted in the present work, but an alternative (Prandtl [9]-kolmogorov [10]) model is used in which,

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

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:

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

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 is called the one-equation (k-l) model.

The above governing equations have been discretised by using the standard finite element method [11] and Galerking weight residual approach is adopted to solve the discretised equations. The flow domain is divided into quadratical 8-noded elements used to define the variations in velocities and turbulent kinetic energy, and linear 4-nodded elements used for the pressure. Greens theorem is used then to reduce the order of the equations to unity resulting in a “weak formulation” which resulted in non-linear equations matrix which is solved by either a coupled or an uncoupled method. Within the N.W.Z. universal laws concept [12], or one-dimensional parabolic elements, in a direction normal to solid wall is adopted.

III. NEAR WALL ZONE MODELING TECHNIQUES

Within the main domain, conventional two dimensional isoparametric elements are used to discretise the flow domain, and within the near wall zone (Figure 1) different techniques were used, these are as follows,

i) Conventional finite elements (i.e. 2-D elements up to the wall) are used to discretise the N.W.Z. and the variable values, following analysis, are used as reference data. However an excessive mesh refinement was needed which is expensive in computer time and memory.

ii) In order to avoid such excessive refinement, semi-empirical equations, known as “Wall Functions” or the so-called “Universal Laws”, are used to bridge from a solid boundary to the main domain.

iii) In the present work, a finite elements technique has been adopted, using one-dimensional (3-noded elements) normal to the wall (Figure 2).

In (iii) the momentum equations in direction normal to the wall surface, together with pressure equation and the kinetic energy equation are solved in the near wall zone. A pressure procedure, developed by Schneider [13], which implements the conservation of mass through the use of the pressure Poisson equation, in a direct manner has been utilized in this work. The inaccuracies associated with the use of one-dimensional elements in one direction normal to the wall, has also been investigated.

IV. BOUNDARY CONDITIONS

The actual domain is taken as a straight channel of with D, which is taken as 1.0, and length L. Particular attention is given to the important aspect of studying the viscous flow. It is particularly important that the location and imposition of boundary conditions on a downstream boundary is considered. For such condition the boundary conditions applied are as follows:

1. Velocities and pressure

i. Forced boundary conditions such as,

$\phi = \phi_\Gamma$ on the boundary $\Gamma_1 (\phi = \text{velocity})$

ii. Traction boundary conditions, where the traction’s are either defined or updated on boundary,

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

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

2. Turbulent kinetic energy (k) as per (i) above or updated Neumann conditions.

In this work, poisiuille flow is considered only and the boundary conditions imposed were as shown in Figure 1. Compatible fully developed velocity and kinetic energy profiles which look like parabolic curve were imposed at the upstream section when fully developed turbulent flow was considered at the first stage and the traction's were updated at downstream. These profiles were obtained by using the outlet values form each iteration as new approximations to the values at the inlet until a convergent condition is satisfied.

V. RESULTS AND DISCUSSION

The validity of the wall element technique (1-D elements in one direction) is tested by analyzing fully developed turbulent flow. Different Reynolds numbers based upon the width of D was investigated.

Convergent fully-developed turbulent velocity profiles are presented in Figures 3 which shows that the velocity values obtained by universal profiles have some discrepancy from those obtained from the advocated technique. Figures 4 clearly shows, the results obtained from the adoption of the presently advocated technique exhibits excellent agreement with the correct solution which resulted from the complete mapping. These are, superior to those obtained using universal laws.

Also, an excellent agreement between the developed technique and experimental results [14] shown in Figure 5.
Once more, the "correct" values are remarkably close to those obtained from the proposed technique, as shown in Figures 6-8 which refer to the velocity, kinetic energy and the turbulent viscosity.

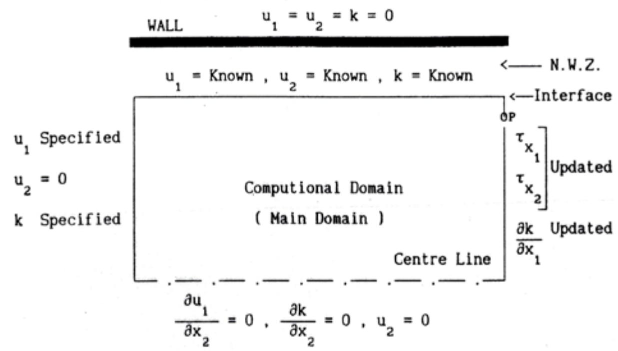

Figure 1: Boundary conditions when the mesh is terminated at small distance away from the wall.

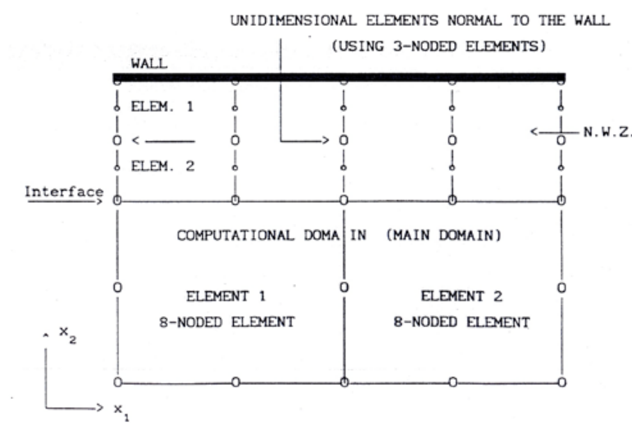

Figure 2: One-dimensional elements in one-direction normal to the wall used in the N.W.Z.

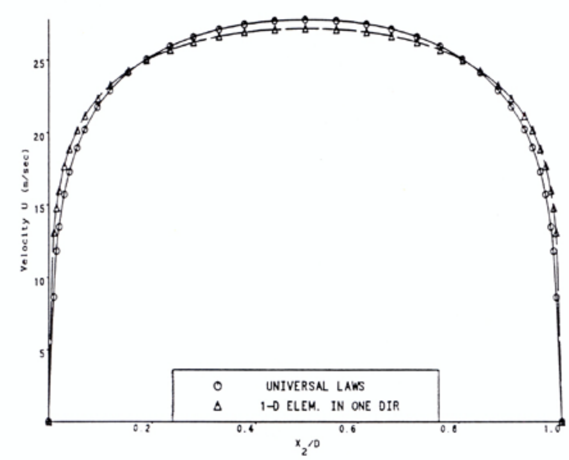

Figure 3: Turbulent velocity profiles for fully-developed flow, at 8D downstream, L=8D, Re=50.000

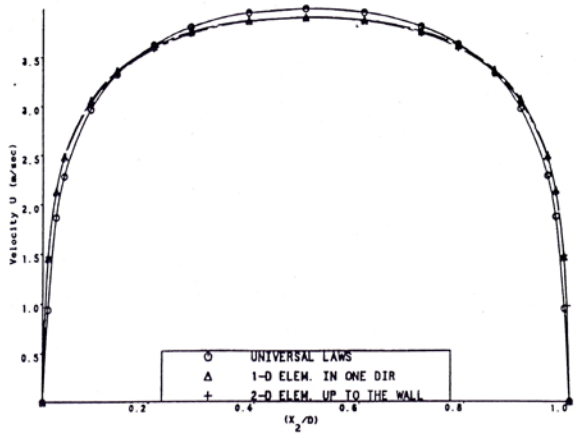

Figure 4: Turbulent velocity profiles for fully-developed flow, at 8D downstream, L=8D, Re=12.000

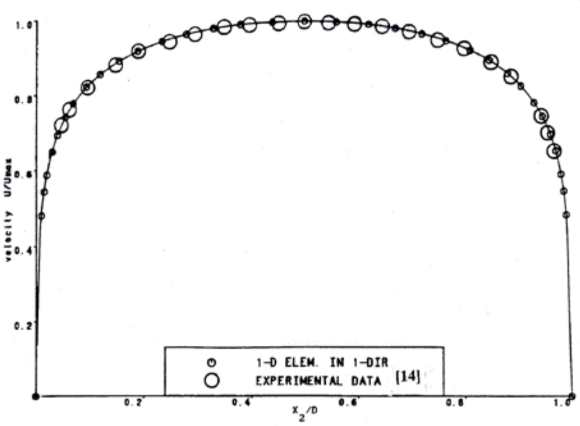

Figure 5: Turbulent velocity profiles for fully-developed flow, at 8D downstream, L=8D, Re=50.000

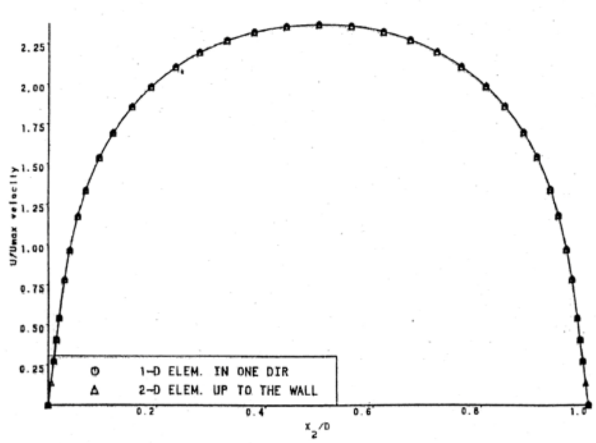

Figure 6: Turbulent velocity profiles for fully-developed flow, at 1.4 D downstream, L=1.4D, Re=1.000

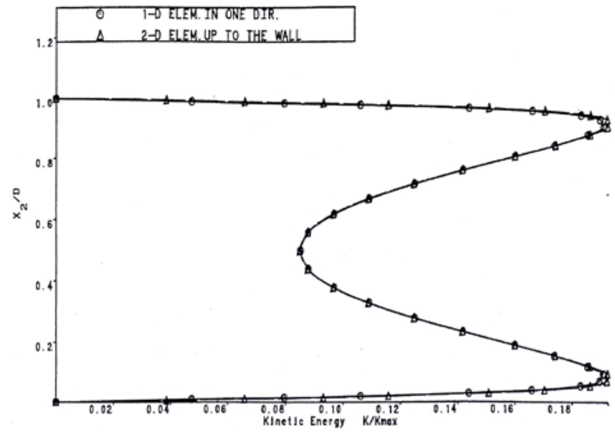

Figure 7: Kinetic energy profiles for fully-developed turbulent flow, at 1.4 D downstream, L=1.4D, Re=1.000

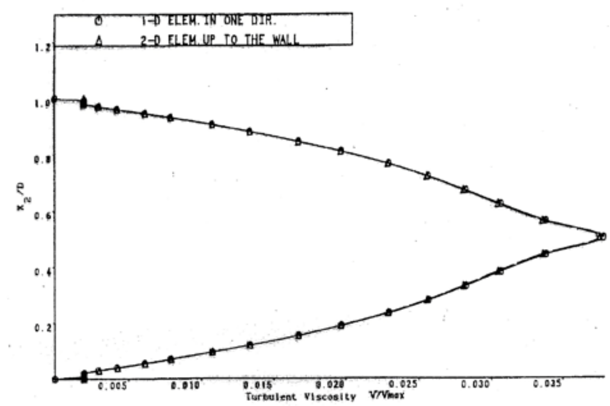

Figure 8: Viscosity distribution profiles for fully-developed turbulent flow, at 1.4D downstream, L=1.4D, Re=1.000

VI. CONCLUSIONS

1. The utilization of empirical universal laws is not valid since these laws are only really applicable for certain one-dimensional flow regimes.
2. The general use of 2-D elements up to the wall is not economically viable. Therefore to avoid such an excessive refinement, these methods have been replaced by introducing a wall element technique, based on the use of the F.E.M.
3. The use of the wall element technique in one direction has shown an excellent results when the fully-developed flow considered.

Therefore, this technique can be used with confidence for turbulent fully-developed case.

REFERENCES

[1] Wiginton, C. L. and Dalton, C., Incompressible laminar flow in the entrance region of a rectangular duct, *J. Apple. Mech*.,Vol. 37, pp. 854-856 (1970).

[2] Sparrow, E. M., Hixoin, C. W. and Shavit, G., Experiments on laminar flow development in ractangular duct, *J. Basic Eng*., Vol. 89, pp. 116-124 (1967).

[3] 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).

[4] Zienkiewicz, O.C. and Taylor, R. L., The finite element method. *McGraw-Hill* (1988).

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

[6] 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, pp. 97-105, (1991).

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

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

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

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

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

[12] Davies, T. J., Turbulent phenomena, *Academic Press*,(1972).

[13] Schneider, G. E., Raithby, G. D. and Kovanovich, M., Finite element analysis of incompressible fluid flow, incorporating equal order pressure and velocity interpolation, Proc. *Int. Conf. Num. Meth. in laminar and turbulent flow*, pentech press, London, pp. 89-102, (1978).

[14] Nayak, U.S.L. and Stevens, S.J, An experimental study of the flow in the annular gap between a long vehicle and a low close-fitting tunnel, *Report: Dept. of Technology, Loughborough University of Technology*, (1973).

Sabah N. Tamimi was born in Baghdad, Iraq on 9th April 1956. He was western educated. He received his M.Sc. in Computer Science and the Ph.D. degree in Applied Computer Science, both of them from University of Wales, UK in 1988 and 1992, respectively.

He is working continuously as a full-time in educational sector at university level for more than 20 years. In addition, the last 10 years, he has been involved in administrative sector working as Deputy Dean and Dean as well as faculty. He is currently an Associate Professor and Dean at the College of Computing, Al Ghurair University, United Arab Emirates. His research interests include software testing techniques, computational flow dynamics, computer simulations, computer graphics and databases.