International Journal of Computer Science
and Engineering (IJCSE)
ISSN 2278-9960
Vol. 2, Issue 2, May 2013, 115-122

International Academy of Science, Engineering and Technology
Connecting Researchers; Nurturing Innovations

COMPUTATIONAL MODELING TO SIMULATE NEWTONIAN VISCOUS FLOW

SABAH TAMIMI

College of Engineering & Computing, Al Ghurair University, Dubai, United Arab Emirates

ABSTRACT

This paper's proposal is to show some significant results obtained by analyzing the effect of the boundary layer on the fluid behavior. An effective technique called "wall element technique" based on the finite element method (FEM), has been modified and adopted in a zone close to the solid wall to depict Newtonian viscous flow in a smooth straight channel and well compared with other conventional techniques for determination of confined developing turbulent flow with a one equation model used to model the turbulent viscosity.

KEYWORDS: Developing Turbulent Flow, FEM, Modified Wall Element Technique, Pressure Flow

INTRODUCTION

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 publications on more general case of turbulent flow are still few [4-5]. Since it has not been possible to obtain exact analytical solutions to such flows, an accurate numerical approach would be very beneficial to researchers. 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 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 [6-8]. A more common approach is to terminate the actual domain at some small distance away from the wall, where the gradients of the independent variables are relatively small, and then another technique is needed to model the flow behavior in the near wall zone.

In previous work, the validity of the wall element technique, based on the use of the finite element in one direction normal to the solid wall has been tested and applied successfully to fully developed flow [9-10] but this is not the case for developing flow [11]. In the present paper, this technique which is based on finite elements method has been modified and adopted, by using one-dimensional elements in two directions normal and parallel to the solid wall.

The validity of the modified technique has been tested for developing flow, along with other techniques to simulate turbulent flow in a smooth straight channel.

MATHEMATICAL 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,

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

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 continuity equation can be written as:

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{2}$$

Equation (1) and (2) cannot be solved unless μ_e is provided. In the present work, a one equation model has been adopted so that,

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

1_μ is the length scale which is taken as 0.4 times the normal distance from the nearest wall surface, C_μ is a constant. In the present work, the one equation model of turbulence is used, and a transport equation is derived and used for evaluation of kinetic energy [12-13]. The distribution of the turbulence kinetic energy k can be evaluated by transport equation;

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

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 conventional two dimensional finite element method [14-16] to discretise the flow domain.

BOUNDARY CONDITIONS

For developing flow, constant values should be assumed at all variables upstream. So, the velocity imposed upstream was at 2.0 m/sec and the kinetic energy imposed as 0.02 m^2/sec^2 . No slip condition were imposed on solid walls and tractions updated downstream. The results are assumed to be converged when the relative change in any variables is less than 1%. Tractions are given by,

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

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

NEAR WALL ZONE

Within the near wall zone either universal laws concept or conventional finite elements (2-D elements up to the wall) were used as shown in Figure 1. In the present work, a wall element technique based on finite elements method has been modified and adopted, using one-dimensional (3-noded elements) in two directions normal and parallel to the wall as shown in Figure 2.

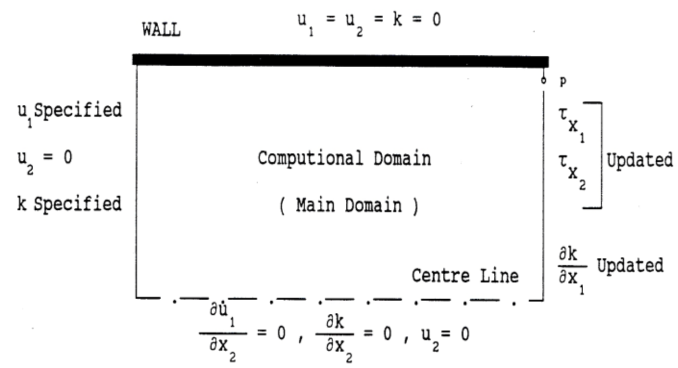

Figure 1: Boundary Conditions for Turbulent Flow Using Two-Dimensional Elements up to the Wall

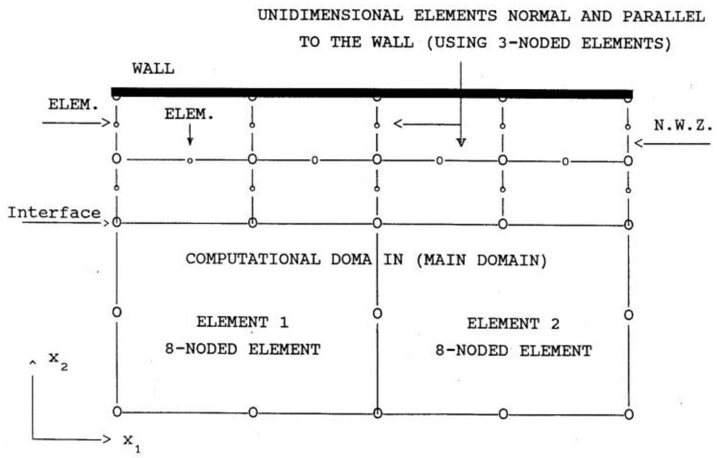

Figure 2: One-Dimensional Elements in Two-Directions Normal and Parallel to the Wall Used in the N.W.Z

RESULTS AND DISCUSSIONS

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 length L. Reynolds number based upon the width of the channel of 12.000 was considered when pressure flow was considered only. The validity of the modified adopted wall element technique (1-D elements in two directions) has been tested for developing turbulent flow, and compared with other accepted techniques.

Figure 3 clearly shows that the velocity profiles obtained by universal laws have some discrepancy from those obtained when the wall element technique (i.e. 1-D in two directions normal and parallel to the wall) was used. In fact the disparity between the advocated technique and universal laws is now even greater than those obtained previously for fully developed flow [9, 10]. So far, the results obtained for fully developed and for developing flow proved that, the wall element techniques using elements in the N.W.Z. are always better than those obtained when using universal laws. Also shows an excellent agreement with the correct solution which resulted from the complete mapping (i.e. 2-D up to the wall).

Figures 4-6 represents the downstream velocity $u_{1,}$ kinetic energy and turbulent viscosity profiles, respectively. In summary these show that the results obtained from the used of 1-D elements in two directions are superior to those obtained when 1-D elements in one direction only was used. This was known, conceptually, but the variation has now been demonstrated. These results prove that, the use of one dimensional element in 2-directions is a valid technique for developing flow and one can avoid both the mapping of 2-D elements up to the wall and the use of 1-D elements in one direction.

The velocity, the kinetic energy and turbulent viscosity at different cross sections of the channel length are shown in Figures 7-9, respectively. Clearly, these figures have shown different profiles at different cross sections for developing flow, this because the change in the velocities u_1 and u_2 in both directions x_1 and x_2. Each curve in Figures 7-9 exhibits different characteristics for developing flow, since the flow still develops gradually when the channel length ≤ 40 [15]. Figure 10 shows an excellent agreement between the adoptive technique and the experimental results [17].

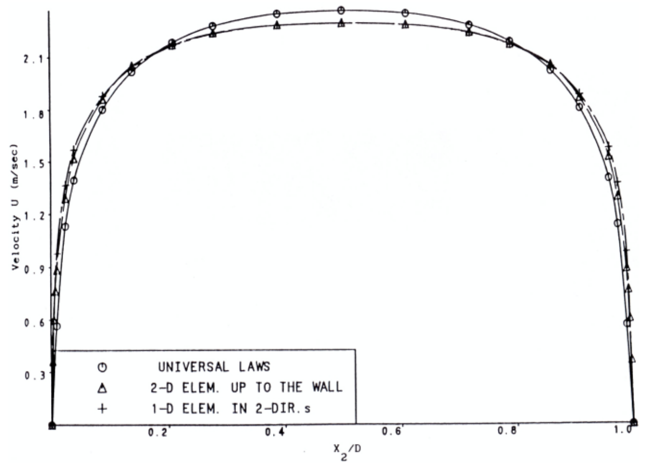

Figure 3: Turbulent Velocity Profiles for Developing Flow, at 10D Downstream, L=10D, RE=12.000

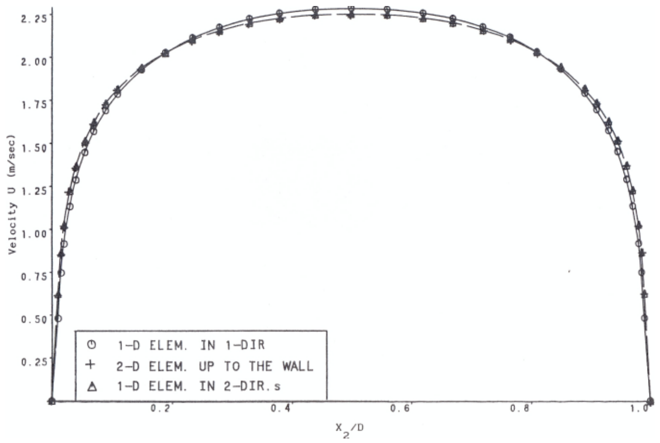

Figure 4: Developing Velocity Profiles for Turbulent Flow, at 10D Downstream, L=10D, RE=12.000

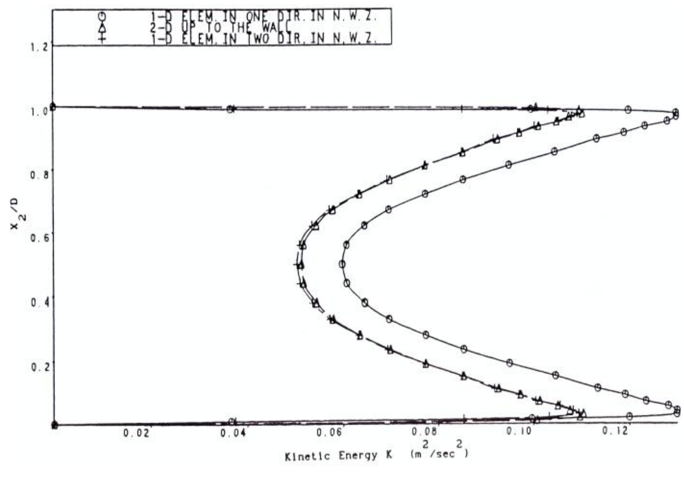

Figure 5: Developing Kinetic Energy Profiles for Turbulent Flow, at 10D Downstream, L=10D, RE=12.000

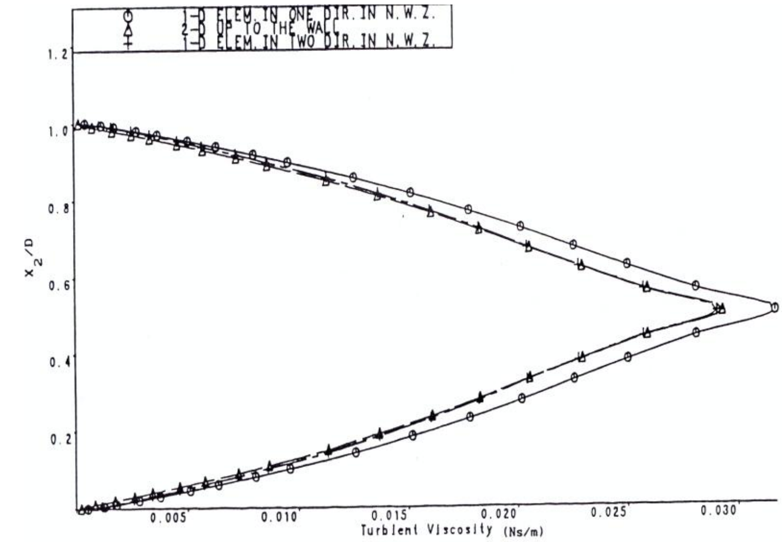

Figure 6: Viscosity Profiles for Developing Turbulent Flow, at 10D Downstream, L=10D, RE=12.000

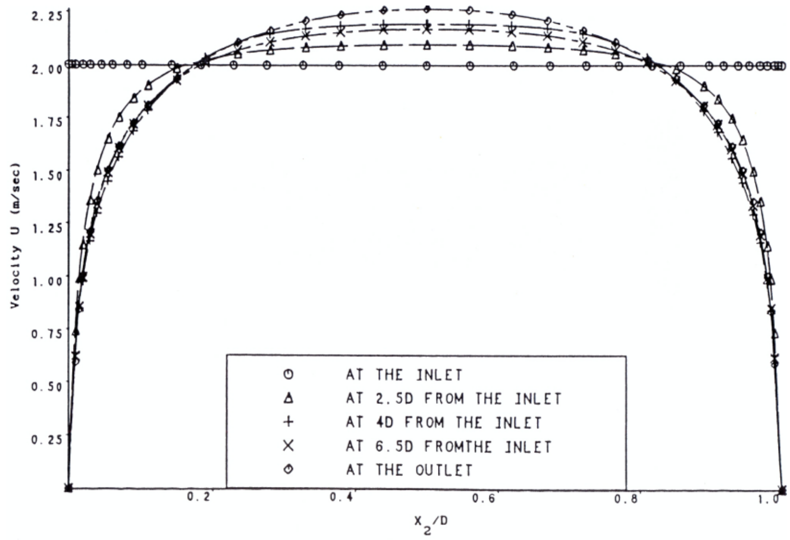

Figure 7: Developing Velocity Profiles at Different Cross Sections (at the Inlet,2.5D,4D,6.5D and at the Out Let) for Turbulent Flow Using 1-D Element in Two Directions in the N.W.Z., L=10D, RE=12.000

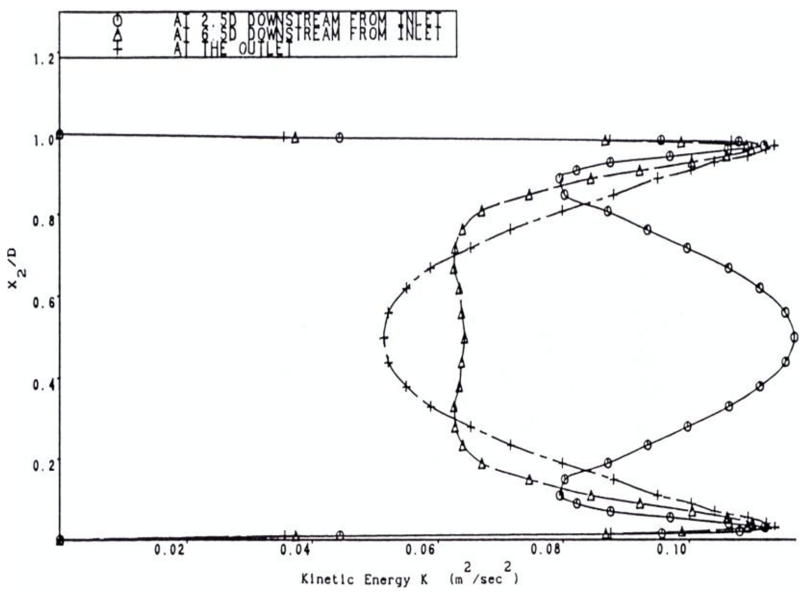

Figure 8: Developing Kinetic Energy Profiles at Different Cross Sections (at 2.5D, 6.5D and at the Out Let) for Turbulent Flow Using 1-D Element in Two Directions in the N.W.Z., L=10D, RE=12.000

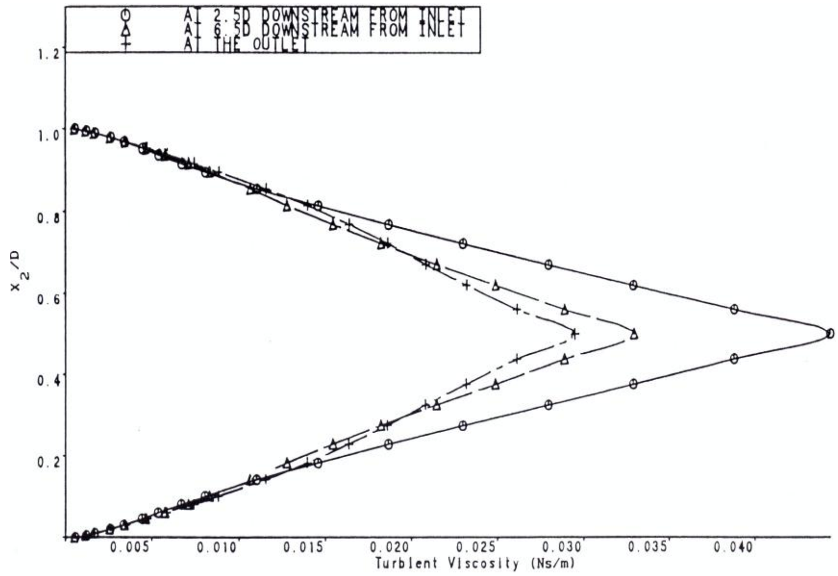

Figure 9: Developing Viscosity Profiles at Different Cross Sections (at 2.5D, 6.5D and at the Out Let) for Turbulent Flow Using 1-D Element in Two Directions in the N.W.Z., L=10D, RE=12.000

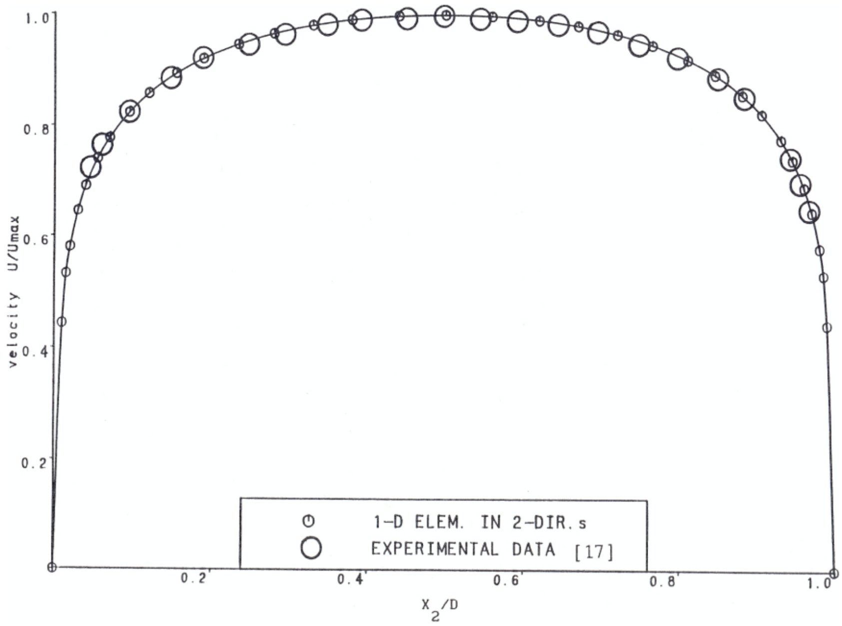

Figure 10: Velocity Profiles Obtained When 1-D Elements in 2-Directions Used in the N.W.Z. along with Experimental Results, L=10D, RE=50.000

CONCLUSIONS

The utilization of empirical universal laws is not valid since these laws are only really applicable for certain unidimensional flow regimes. 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 FEM.

The accuracy of this technique when used in one direction is clearly not valid for developing flow since the assumption of unidirectional flow is unacceptable. Whilst, the use of 1-D elements in two directions has been applied successfully and proved to be superior to other techniques and, can be used with confidence for developing turbulent flow.

REFERENCES

1. Wiginton, C. L. and Dalton, C., Incompressible laminar flow in the entrance region of a rectangular duct, J. Apple. Mech., Vol. 37, 1970, pp. 854-856.
2. Sparrow, E. M., Hixoin, C.W. and Shavit, G., Experiments on laminar flow development in rectangular duct, *J. Basic Eng.*, Vol. 89, 1967,pp. 116-124.
3. Garg K. Vijay, Developing flow in a rectangular duct, *Proc. Int. Conf. Num. in Laminar and Turbulent Flow, Seattle*, 1983, pp. 58-68.
4. Taylor, C., Hughes, T. G. and Morgan, K., A numerical analysis of turbulent flow in pipes, *Int. J. Computers and fluids*, Vol. 5, 1977, pp. 181-204.
5. Taylor, C., Harper, J., Hughes, T. G. and Morgan, K., An analysis of developing turbulent flow in a circular pipe by the finite element method, *Proc. Int. Conf. Num. Meth. in Laminar and Turbulent Flow*, New York, 1978, pp. 341-350.
6. Launder, B. E. and Shima, N., Second moment closure for near wall sublayer: Development and Application, *AIAA Journal*, Vol. 27, 1989, pp. 1319-1325.
7. 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.
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. Sabah Tamimi, Progress in Boundary Element Technique of Viscous Fluid, 2012 IEEE International Conference on Computer science and Automation Engineering, Vol.2, pp. 714-718.
10. Sabah Tamimi, On turbulent flow near a wall, *Journal of Communication and Computer*, ISSN: 1548-7709, Vol. 9, No. 10, 2012, pp.1104-1109.
11. Sabah Tamimi, An Analysis of Developing Turbulent Flow", *International Journal of Computer Science & Communication (IJCSC)*, ISSN: 0973-7391, Vol. 3, No.1, 2012, pp.11-13**.**
12. Wolfshtein, M., Some solutions of the plant turbulent impinging jet trains, *ASME, J. Basic Eng*., Vol. 92, 1970, pp. 915-922.
13. 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.
14. Zienkiewicz, O.C., The *Finite element method*, 3rd Ed., McGraw Hill, 1977.
15. Taylor, C. and Hughes, T. G., *Finite element programming of the Navier-Stokes equation*, Pineridge press, 1981.
16. Griffiths DV and Smith IM, Numerical Methods for Engineers, *Chapman and Hall/CRC*, 2nd Edition, New York, 2006.

17. 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 Transport technology, Loughborough university of Technology*, 1973.

AUTHOR DETAILS

Brief Biography of the Author: Sabah Tamimi was born in Baghdad, Iraq on 9th April 1956. He was western educated. He holds a M.Sc. in Computer Science (1988) and a Ph.D. in Applied Computer Science (1992), both of them from University of Wales, UK.

He is working continuously as a full-time academic faculty for more than 21 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. He is currently an Associate Professor at the College of Engineering and Computing, Al Ghurair University, United Arab Emirates. His research interests include: computer modeling and simulation, computational flow dynamics, computer graphics and databases. He has a very good number of publications in International journals and conferences proceeding (of IEEE, WSEAS, etc.), and reviewed many papers for international journals and conferences. Also, he is an editorial board member of different international journals.