Proceedings of the ASME 2010 3rd Joint US-European Fluids Engineering Summer Meeting and 8th International Conference on Nanochannels, Microchannels, and Minichannels
FEDSM-ICNMM2010
August 1-5, 2010, Montreal, Canada

FEDSM-ICNMM2010-31290

NUMERICAL INVESTIGATION OF TURBULENT MODELS IN THE DEVELOPMENT AND ESTABLISHMENT OF TURBULENT FLOWS WITH FLOW CONDITIONERS

Boualem LARIBI
Industrial Fluids Laboratory,
Measurements and Applications
University of Khemis Miliana, Algeria
E-mail : boualemlaribi@yahoo.com

Pierre WAUTERS
Mechanical department
Louvain Catholic University
Belgium
pierre.wauters@uclouvain.be

Abdelkader YOUCEFI
Mechanical department
University of sciences and
technology of Oran. Algeria
youcefi_a@yahoo.fr

ABSTRACT

The accuracy in measuring flow of fluids such as gas and oil has a great importance for the Algerian economy. The flows of fluids in non-standard conditions, presence of disturbances, in which there are flow meters in pipes, make a very important error. International standards ISO 5167 and AGA3 stipulate that the meter is installed in a fully developed flow. This article describes a numerical investigation of development and establishment of flows in the presence of a double bend 90° in perpendicular planes as a perturbation. The software used was code Fluent where different turbulence models are tested to better simulate and view the effectiveness of models in the description of the flow of fluid compared to flow behaviour cited in the standards and the experimental results. The numerical experimentation is done with air in a pipe of 100mm diameter at a Reynolds number 10^5. The numerical analysis is based on solving Navier-Stokes equation system with several turbulent models, k-ε, k-ω, RSM and its variants.

INTRODUCTION

The majority of the industrial flowmeters are gauged under conditions of perfectly established flow. This condition is actually very difficult to respect, being given the presence of the components of the system of control which are essential to its exploitation. So the presence of singularities such as, valve, elbow… etc, obviously constitute a source of error in the measurement of the flow [1]. In order to avoid these errors [2] the technology of the flow conditioners [3] did not cease developing these last years. They have as a main role the acceleration of the formation of the established flow, i.e., obtaining a fully developed velocity profile [4] over a length of the path of contact as reduced as possible. In industrial practice, considering the reasons of obstruction which do not make it possible to have rectilinear long distances of conduits, and in order to attenuate the disturbances of the flow one generally places between the flowmeters and the disturbance a device says flow straightener or flow conditioner [5]. These elements have the role to accelerate the development of flow and to ensure its establishment in a shorter distance (20 to 30 diameters of conduit). In this article it is not a question to concentrate themselves on the problem of the conditioners but rather to validate the turbulent models used by the code Fluent in order to apply best the latter in our research concerning the rectifiers of flow. The objective of this article is to carry out numerical simulations of the development of flow. Our system is a pipe with 100 mm of diameter at Reynolds number of 10^5. The disturbance of the flow is ensured by a 90°double bend in two perpendicular plans [6]. Several researchers regard this singularity as a standard disturbance which not only produced one very disturbed velocity profile, but also very disturbed turbulent intensity profile [7]. The numerical analysis is based on the resolution of the Navier-Stokes equations, with several systems of closing to knowing k-ε, k-ω and RSM [10]. The effect of flow conditioners will be discussed in paper FEDSM-ICNMM2010-31291.

NOMENCLATURE

D	inner pipe diameter
k	turbulent kinetic energy
Rey	Reynolds number
t	time
y	radial coordinate
z	axial coordinate

u mean velocity
U_{max} maximal velocity
ε dissipation rate of k
ω specific dissipation rate

TURBULENT MODELS INVESTIGATED

Basic equation

The general equation used in CFD and by the code Fluent is given by:

$$\frac{\partial(\rho\phi)}{\partial t} + div\left(\rho\phi\bar{U}\right) = div\left(\Gamma_{\phi}.grad\ \phi\right) + S_{\phi} \qquad (1)$$

Where ϕ is the general dependent variable which can be the mean velocity, the turbulent kinetic energy or the rate of dissipation of the turbulent kinetic energy.

S_{ϕ} is the term source of the variable ϕ

Γ_{ϕ} is the coefficient of diffusion of ϕ

Models used in the simulation[10]

The code Fluent presents several models of turbulence, and for the lightening of the text we will not reproduce the equations in this article. The reader can consult literature of code Fluent for more details. Here we only present a summary of the models.

The k-ε models

The k-ε model is the simplest model known as two equations model. This model assumes that the turbulence regime is fully developed throughout all the section of pipe and the effects of molecular viscosity are negligible compared to the turbulent viscosity (far wall). It is based on the Boussinesq assumption. It comes in three forms:

K-ε Standard model: it is a semi empirical model. Two transport equations are used: one for the turbulent kinetic energy k and the other for its dissipation rate ε.

K-ε RNG model: Based on a mathematical technique called re-normalization (hence the acronym RNG: Re-Normalization Group), this variant is characterized in practice, in the equation of ε basis, by a coefficient $C_{\varepsilon 2}$ depends on k/ε, therefore variable. This helps cushion the turbulence in regions of high strain rate (turbulence overestimated by the standard model). The quality of results is improved for the flow downstream of a step, areas of detachment-gluing and the vortex flow.

K-ε Realizable model: The concept of the model introduced by Lumley means that the model must respect asymptotic situations. For example, k and ε should never be negative. This model seems well suited for circular jets, boundary layers with strong adverse pressure gradient, flows with high curvature and vortex flows.

The K-ω models

Competitor of the k-ε model, the k-ω model uses the same guiding principles, but replaces the equation in ε by a balance of turbulent vorticity. It comes in two forms:

K-ω Standard model: The k-ω standard model proposed by FLUENT is based on the model of Wilcox [10]. Its structure is similar to k-ε model. This model involves two equations of transport: one for the turbulent kinetic energy k and another for the specific dissipation rate ω.

K-ω SST model (Shear-Stress Transport): The SST model or Shear-Stress Transport developed by Menter [10], is derived from the k-ω Standard model. This model combines the robustness and accuracy of the formulation of the k-ω model in the region near the wall with the k-ε model and all features mentioned above for free flow defined far wall.

The model of Reynolds stress (RSM)

The RSM model or Reynolds Stress Model is a model of closed second order. In some cases (curved boundary layers, swirling flows, rotating flows), the approximation based on the Boussinesq hypothesis to represent the Reynolds stress is not applicable. It comes in three forms:

RSM-1 Linear pressure-strain model: Fluent in default, the term pressure-tension in the exact transport equation is modeled after the proposals of Gibson and Launder [10].

RSM-2 Quadratic pressure-strain model: A model of pressure-tension optional proposed by Speziale, Sarkar and Gatski [10] is provided in Fluent, This model has been shown to give performance in upper range of shear flow. This accuracy should be improved interesting for complex flows, especially those with curvature natural flow.

RSM-3 Low-Re Stress-Omega Model: This model is based on the equations of omega and the LRR model [10]. This model is ideal for modeling flows on curved surfaces and vortex flow.

The closure coefficients are identical to the k-ω model, however, there are additional factors closure. The RSM-3 looks at k-ω model due to its excellent predictions for a wide range of turbulent flows. In addition, the modification low Reynolds and boundary conditions for rough surfaces are similar to the k-ω model.

EXPERIMENTAL DEVICE SIMULATION

Our facility includes a pipe with diameter D equals 100mm, and length 10D followed by a 90° double bend out of plane followed a length pipe of 40D [7]. Figure 1.

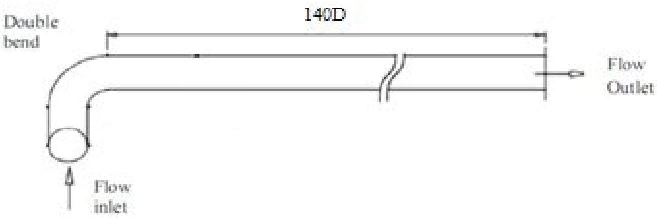

Figure 1. Conduit

RESULTS AND DICUSSION

References profile

Velocity profile: To validate the numerical predictions obtained, the adimensional velocity profile u/U_{max}, are compared with ±5% of the theoretical fully developed velocity profile according to ISO 5167. In a turbulent stationary flow, the fully developed velocity profile of flow is determined by the power law [9]:

$$\frac{u}{U_{max}} = \left(1 \pm 2\frac{y}{D}\right)^{1/7} \quad (2)$$

Where n is an integer depends of Reynolds number. In our case Re = 10^5 which gives n=7.

Turbulent intensity profile: For the turbulence intensity, the reference profile is determined by the Lawn experimental profile [7].

Results

Velocity profile development: The development and establishment of flow is shown in figures 1 to 9 at various axial stations z/D downstream the double bend. These profiles are compared to the theoretical power law profile with a tolerance of ±5% as recommended by ISO 5167. Eight models of turbulence are considered.

From figure 1, at station z/D = 0.8 downstream the double bend, we find that the double bend causes a disturbance velocity profile entirely outside the theoretical profile. It also seems that all the turbulence models give an identical solution at this station. We can see an acceleration of the flow for radial positions between y/D=0 and y/D=0.5, and a deceleration of the flow between y/D=0.5 and y/D=1. These results are obvious given the asymmetry of velocity profile due to the presence of double bend, faster flow on the outer radius and a deceleration of the inner radius.

Figures 2 and 3 shows that the flow has a deficit in the central region of conduct contrary to the profile which has a theoretical maximum for this region. These results are in perfect agreement with the experimental results obtained by Laribi and al [7] Merzkirch and al [8].

The development of flow mainly continuity consequence is apparent in figures 2 to 9. All models tested seem to follow the same development. Remarkable results are obtained by k-ε and RSM-3 models at stations z/D=130 and z/D=140, for which the velocity profile fits the theoretical profile remarkably.

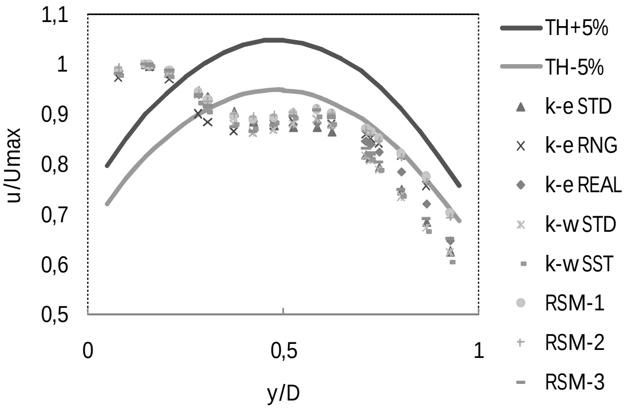

Figure 1. Velocity Profile at z/D=0.8

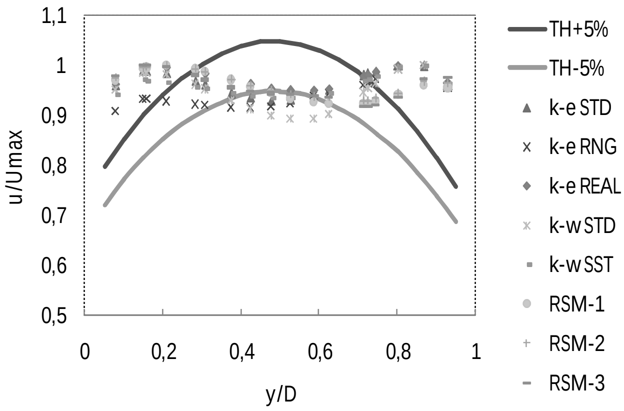

Figure 2. Velocity Profile at z/D=6

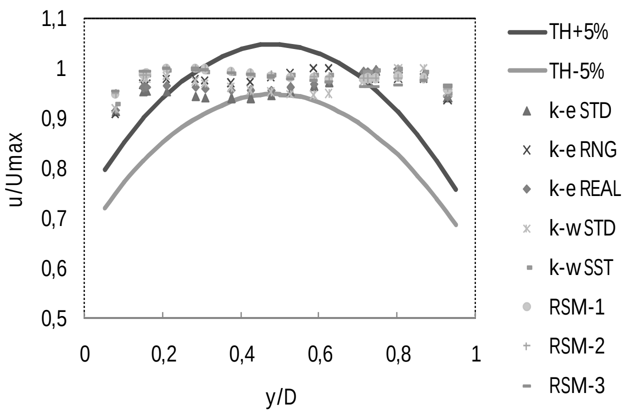

Figure 3. Velocity Profile at z/D=9

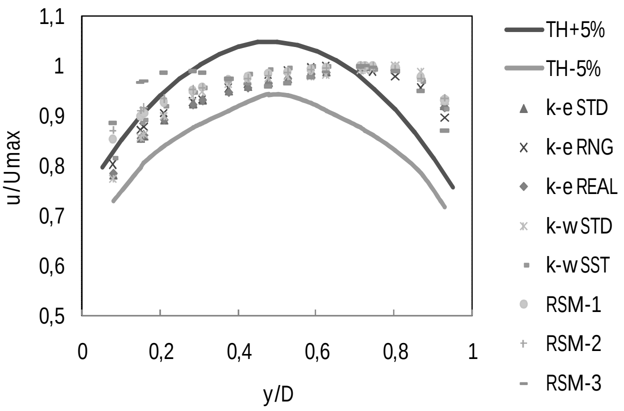

Figure 4. Velocity Profile at z/D=12

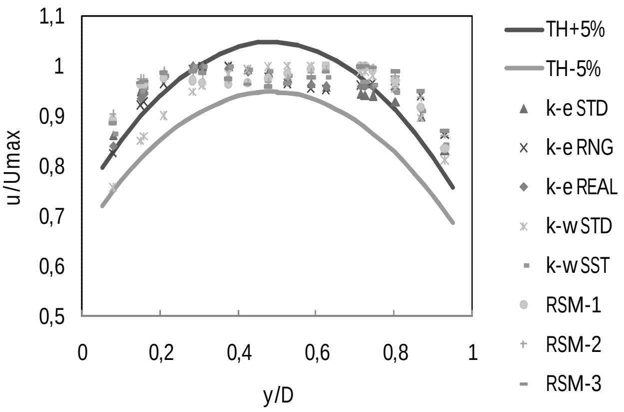

Figure 5. Velocity Profile at z/D=22

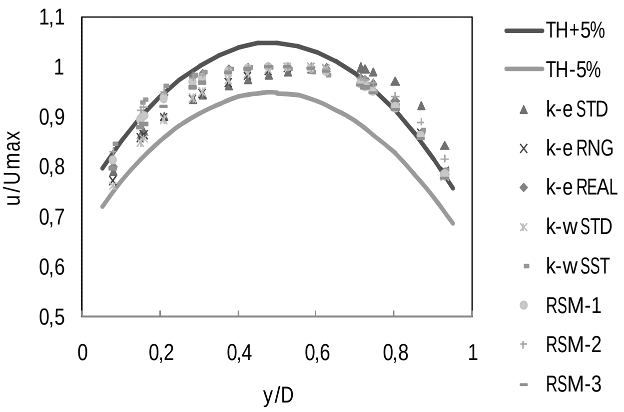

Figure 6. Velocity Profile at z/D=100

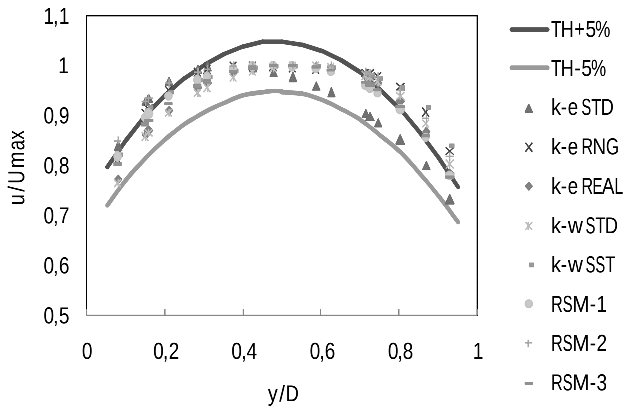

Figure 7. Velocity Profile at z/D=120

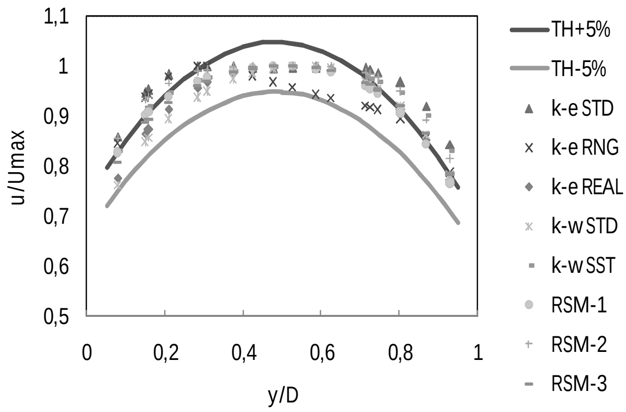

Figure 8. Velocity Profile at z/D=130

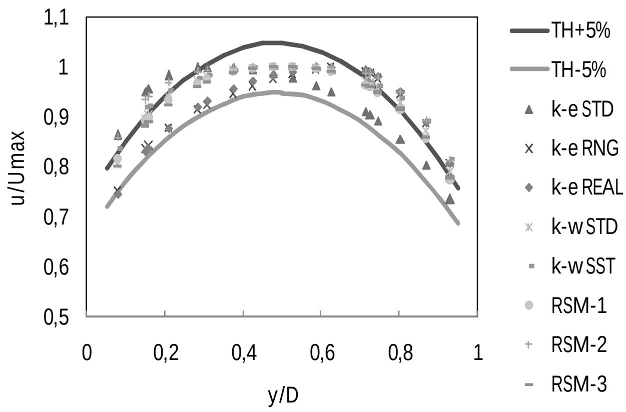

Figure 9. Velocity Profile at z/D=140

Turbulent intensity profile development: Similarly the development and establishment of the radial turbulent intensity profile is shown in figures 10 to 18. At various axial stations z/D downstream of double bend. These profiles are compared with Lawn experimental profile. As for the velocity profile, the eight turbulent models are examined.

In figure 10 at station z/D=0.8 downstream of the double bend, we see that the flow develops a high level of turbulence and asymmetric compared to Lawn experimental profile. The double bend causes a nonstandard condition for a meter efficiency. On figures 11 to 18, we find that all models give the same result as turbulent intensity profile, developing well into Lawn profile.

It is remarkable in figures 17 and 18 to show the effectiveness of the RSM-3 model in predicting the profile set that is in perfect agreement with the profile of reference except near the walls where we do not have an experimentales data. The RSM-3 model is followed by k-ε standard model in figure 18 between y/D=0.4 and y/D=0.6. But this adjustment is lost for 0.6>y/D>0.4.

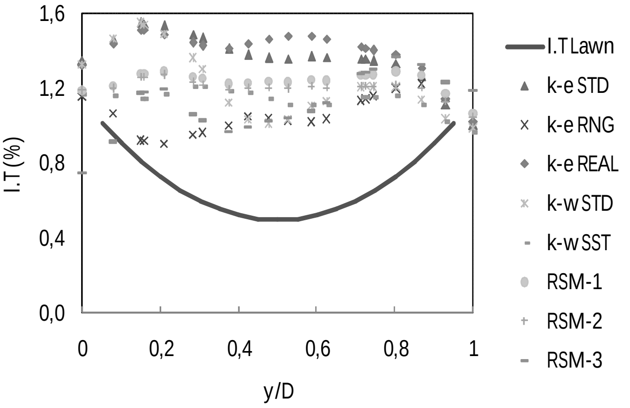

Figure 10. Turbulent intensity at z/D=0.8

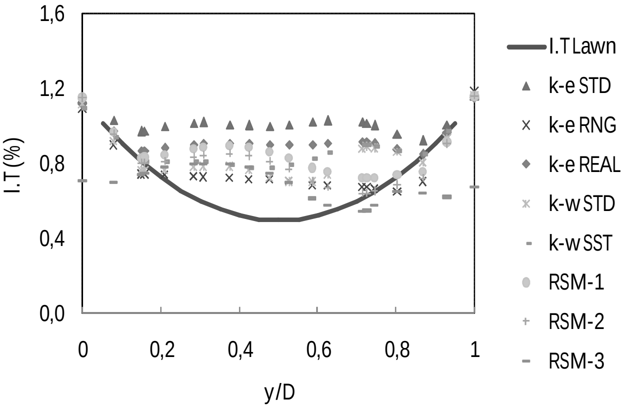

Figure 11. Turbulent intensity at z/D=6

From the analysis of velocity profiles and turbulent intensity profile we can see the effectiveness of the RSM-3 model to predict the development and establishment of flow in the presence of double bend.

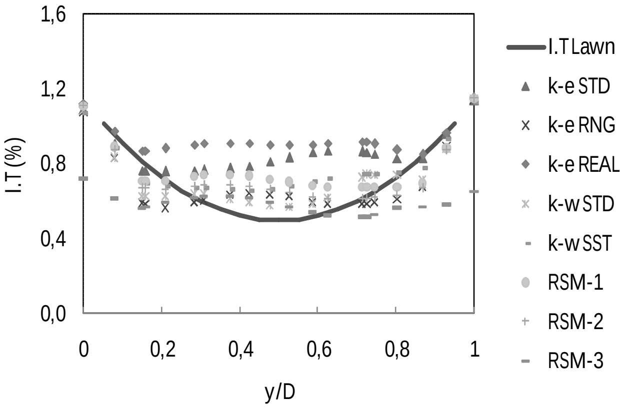

Figure 12. Turbulent intensity at z/D=9

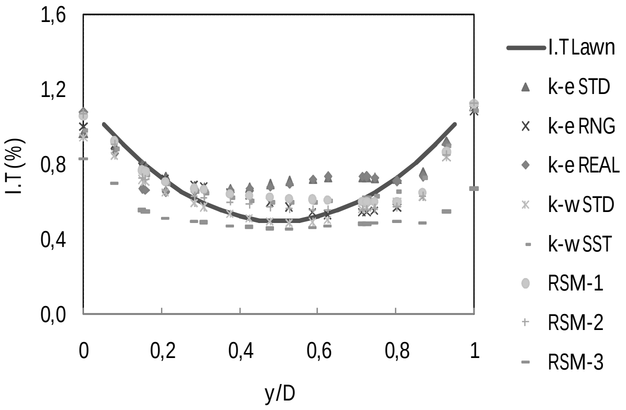

Figure 13. Turbulent intensity at z/D=12

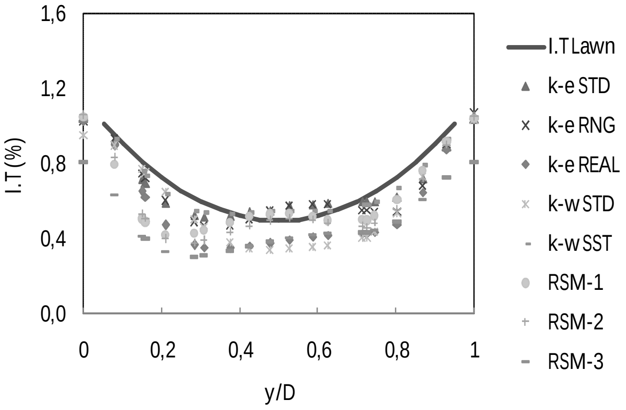

Figure 14. Turbulent intensity at z/D=22

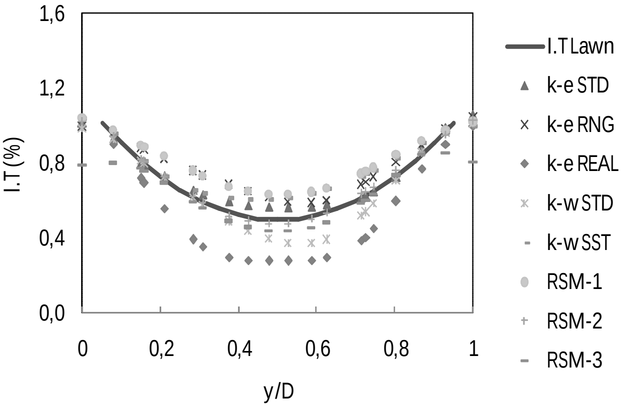

Figure 15. Turbulent intensity at z/D=100

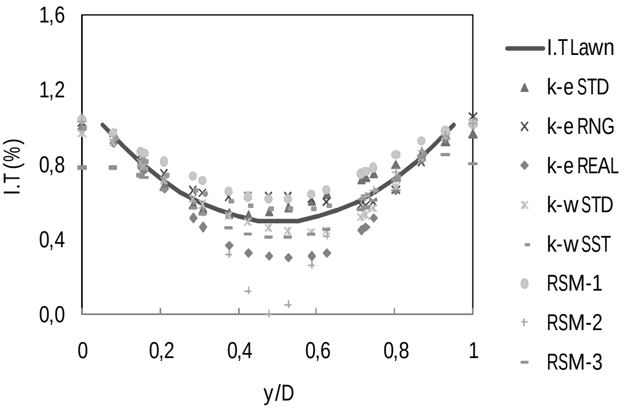

Figure 16. Turbulent intensity at z/D=120

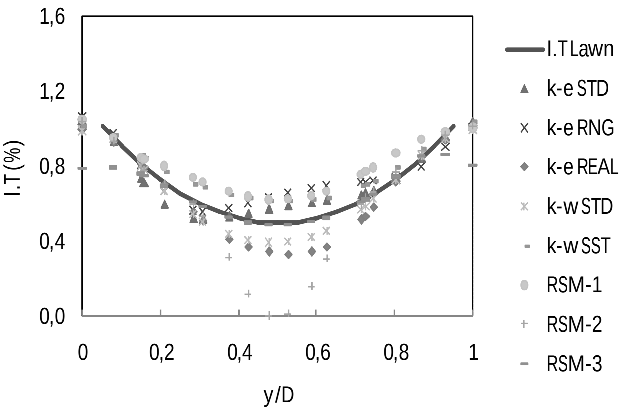

Figure 17. Turbulent intensity at z/D=130

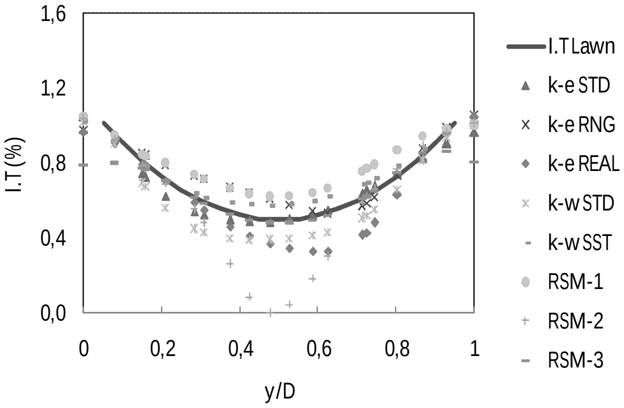

Figure 18. Turbulent intensity at z/D=140

CONCLUSION

The purpose of this work is to study the effectiveness of models of turbulence to whether formulate the development and establishment of turbulent flows in three dimensions in the presence of a disturbance like a 90° double bend out of plane. The simulation code is Fluent v6.3. The simulation is done with an air flow with a Reynolds number of 10^5. In light of the results in figures 8 and 9 for the velocity profile and figures 17 and 18 for the profile of turbulence intensity were the flow is fully developed, at station z/D=130 and 140, we can conclude that among the eight models used, the RSM-3 model is the most effective model to well simulate the development and the establishment of three dimensions turbulent flow.

Indeed be used for simulation of flows in the presence of flow conditioners. This will be a second publication in paper number FEDSM-ICNMM 2010-31291.

ACKNOWLEDGEMENTS

This work is part of the research project no. J0303920080007 approved by the Ministry of Higher Education and Scientific Research. 2009. Contribution to conceive a new flow conditioner.

REFERENCES

[1] ISO 5167, *Measurement of fluid flow by means of orifice plates nozzles and ventury tubes inserted in circular cross section conduits running full, 1991.*

[2] Yeh, T.T and Mattingly, G.E, Flow meter installation effects due to a generic header, *NIST Technical note 1419,1996.*

[3] Gallagher J. ,Flow conditioning, Why, What, When and where, *Siminarion Aavanzado de Medition de Flujo de fluidos, Aguascalientes, Ags, Mexico, September 2009.*

[4] Morrow, T.B., Park, J.T. and Mckee, R. J., Determination of Installation Effects for a 100 mm Meter using a Sliding Vane Technique. *Flow Measurement and Instrumentation Vol .2, N°1, pp 14-20, 1991.*

[5] Rans, R. & al , Flow Conditioning and Effects on Accuracy for Fluid measurement, *7th East Asia Hydrocarbon Flow Measurement Workshop 5th – 7th, Malisia, March 2008,*

[6] Laribi B., Wauters P. and Aichouni M., Experimental Study of the Decay of Swirling Turbulent Pipe Flow and its Effect on Orifice Flow Meter Performance, *ASME Fluids Engineering Division Summer Meeting, FEDSM'01, May 28. 2001*

[7] Laribi B., Wauters P. and Aichouni M.,' Comparative study of aerodynamic behaviour of three flow conditioners, *European Journal of Mechanical and Environmental Engineering, Vol. 48/1, pp. 21-30, March 2003*.

[8] Merzkirch, W. and Kalkühler, K. Velocity and turbulence measurements in the flow downstream of flow conditioners, *Advances in Fluid Mechanics and Turbo machinery, eds. H.J. Rath, C. Egbers, pp. 125-135, Springer-Verlag, Berlin, 1998.*

[9] Darin L. & Bowles E. B., Conditioning On Gas Measurement, *Pipeline & Gas Journal, PP 59-62 February 2008.*

[10] Fluent v6.3, Fluent incorporated, Journal Centerra source park, USA, 2006