The 17th Annual Conference of Mechanical Engineering Network
15-17 October 2003 Prajinburi, THAILAND

The Numerical Simulation of the Turbulent flow generated by an Intake Exposed to a Cross-Flow

A. JANYALERTADUN
Department of Mechanical Engineering, Faculaty of Engineering
Ubonratchathani University, Warinchumrap, Ubonratchathani, 34190
Tel. 0-45288376 ext. 3814, Fax. 0-45288278, E-mail: janyalertadun@hotmail.com

Abstract

An intake exposed to a cross-flow is important in many practical situations; one such example is ventilation extraction ducts where the inlet is suctioned off from the cross-flow. The focus of the current study is to model an experimental case using finite element discretisation of the Navier-Stokes equations. Three different turbulence models; the k-ε renormalization group (RNG), coarse direct numerical simulation (DNS) and large eddy simulation (LES) with the Smagorinsky subgrid scale (SGS) model are used in this study. The results show that the k-ε RNG turbulence model combined with the law of the wall and the coarse DNS model are unsuitable for simulating flows. With LES and applying the van Driest function in the wall regions, the results show qualitatively better agreement to the experimental results than the two-equation modelling.

1. Introduction

There are many situations that can be represented as an intake exposed to a cross flow. Examples are found inside engines, ventilation systems and processing plants. However, very little information is available on these flows [1]. An experimental study of such a flow was done by O'Brien [2]. At present, there is no reported investigations into numerical modelling of the interaction of an intake with a cross flow.

The numerical simulation of the flow generated by the interaction between an intake and a cross-flow, shown in figure 1, can have many applications. Examples are the cross-flow applied to laminar flow and very low velocity at the intake in the case of microperforations in an aircraft skin [3]; or the case study of intake flow from a fluid reservoir to fluid machinery, where the flow reservoir is turbulent moving parallel to the surface [4]. Therefore the investigated parameters and region of interest are different depending upon the flow application. The present work is the study of the numerical simulation of the turbulent flow generated by the interaction between an intake and a cross-flow based upon experimental work. The intake Reynolds number is 173,000, based on the experimental results. The region of interest is downstream of the intake and on the plane near the flat surface. Even though the geometry domain looks simple, the experimental results show that it is not simple flow, especially downstream of the intake and in the near wall region. Moreover the skew boundary layers, which can be found in some complex turbulence flows [5], can also be found in the experimental study of the present flow. Figure 2 is from the experimental study of an intake and a cross-flow and shows a flow visualisation on the surface with the regions of the flow clearly marked and the flow can be divided in four different regions; upstream flow, far-field flow, reversed flow and aft-flow regions. All name regions are indicated on the figure 2. The complicated flow (Strong skew) can be observed in region of the meeting of the far field and aft flow regions.

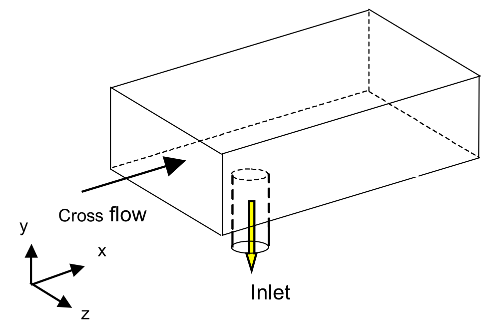


Figure 1 Schematic of a typical computational domain and boundary conditions.

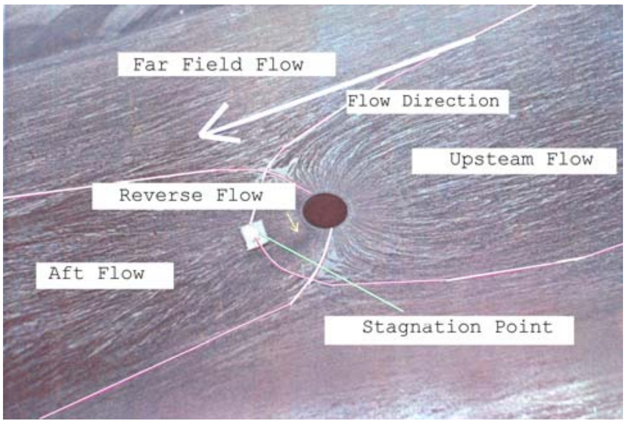


Figure 2 The experimental results of flow visualisation on the surface [1].

2. Numerical Simulation of Turbulent

Three turbulence modeling has been studied, the Reynolds average k-ε RNG, Coarse DNS and LES with Smagorinsky sub-grid scale model.

2.1 Renormalization group turbulence model

One of two-equation Reynolds averaging models, used in present simulation work, is the Renormalization Group k-ε (RNG) model [6]. The main approach of RNG model is to correct inadequacies of Reynolds stress model that tend to be ad hoc in nature and are based on Reynolds averages which can smooth out many of the important features of turbulence. The model systematically removes the smallest scales that are resolvable with available computer capacity. This model has a very similar form to the standard k-ε model but it employs an additional source/sink term in the dissipation equation and employs different values for the various model coefficients as shown in equation (1) and (2):

$$\rho\frac{\partial k}{\partial t}+\rho\overline{u}_i\frac{\partial k}{\partial x_j}=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+G-\rho\varepsilon \quad (1)$$

and

$$\rho\frac{\partial \varepsilon}{\partial t}+\rho u_j\frac{\partial \varepsilon}{\partial x_j}=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right]+C_1\frac{\varepsilon}{k}G-\frac{C_\mu\eta^3(1-\eta/\eta_o)\varepsilon}{1+\beta\eta^3}-C_2\rho\frac{\varepsilon^2}{k} \quad (2)$$

where

$$G=\mu_t\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)\frac{\partial u_i}{\partial x_j}$$

and $\eta=s\frac{k}{\varepsilon}$

with $S=\sqrt{2S_{ij}S_{ij}}=\sqrt{G/\mu_t}$

and $S_{ij}=\frac{1}{2}\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)$

Where i and j= 1, 2 and 3 correspond to x, y and z coordinates, respectively. The primary model coefficients of the RNG model for isothermal flows are c_μ, c_1, c_2, σ_k, σ_ε and the von Karman constant κ. The other two model coefficients η_o and β are directly obtained from the above primary model coefficients. The model coefficients are used in present study as follows; c_μ= 0.085, c_1 = 1.42, c_2 = 1.68, σ_k = 0.7179, σ_ε =0.7179 and κ = 0.3875 [7]. The two-equation viscosity modelling, k-ε RNG model was found to provide a balance between model sophistication and computation economy.

2.2 Coarse DNS or LES with no SGS model

DNS was applied to solved all scales of turbulence directly, but as the mesh resolution is not sufficient to solve the Kolmogorov scale of the turbulence, it is therefore called coarse DNS. The simulation used the option of a laminar solver for the high Reynolds number. The LES with no sub-grid scale model is sometimes called coarse Direct Numerical Simulation. The commercial code, called FIDAP, with the using no turbulent model was implemented for this approach.

2.3 LES with Smagorinsky sub-grid scale model

The partial differential governing equations of the large eddy simulation can be obtained by applying a spatial filter, indicated by the overbar, to the Navier-Stokes and continuity equations. The equations for an isothermal incompressible fluid using a constant filter width are:

$$\frac{\partial\overline{u}_i}{\partial x_i}=0 \quad (3)$$

$$\frac{\partial\overline{u}_i}{\partial t}+\frac{\partial\left(\overline{u}_i\overline{u}_j+\tau_{ij}\right)}{\partial x_j}=-\frac{1}{\rho}\frac{\partial\overline{P}}{\partial x_i}+\nu\frac{\partial^2\overline{u}_i}{\partial x_j\partial x_j} \quad (4)$$

where the residual stresses are given by

$$\tau_{ij}=\overline{u_iu_j}-\overline{u}_i\overline{u}_j$$

and u_i is the velocity component in i direction, P is the pressure, ν is the kinetic molecular viscosity and ρ is the density. The temporal and spatial co-ordinates correspond to t and x_j respectively. To close the filtered equations the residual stresses, called sub-grid stresses, are modelled using an eddy viscous model. The sub-grid stresses model can be expressed as:

$$\tau_{ij} = -2\mu_s S_{ij}$$

where μ_S is the eddy viscosity, calculated by the Smagorinsky model [8], given as:

$$\mu_s = (C_s \cdot \Delta)^2 \cdot \left(\frac{1}{2}\overline{S}_{ij}\overline{S}_{ij}\right)^{1/2} \qquad (5)$$

where S_{ij} is the resolved strain rate, C_s is the Smagorinsky constant whose value is defined to be 0.18 in the present calculations and Δ is the characteristic length scale which is calculated as:

$$\Delta = \left(\Delta_x \Delta_y \Delta_z\right)^{1/3} \qquad (6)$$

Where the length scale is directly calculated from the local cell size. However to account for the flow near the no slip walls becoming laminar, a damping function, called the van Driest damping function, is associated with near wall flows to calculate the Smagorinsky constant, C_s in the near wall region, defined as:

$$C_s = 0.18D \qquad (7)$$

where D is van Driest damping function:

$$D = 1 - \exp\left(-\left(y^+\right)/A\right) \qquad (8)$$

and A is the dimensionless damping constant, varying with flow conditions. For a smooth, impermeable wall, A is given as 26, and as such used in this present simulation. LES is a compromise between the two-equation turbulence models and DNS which solve the large scales of turbulence directly and models the small scales with the sub-grid scale.

3. Numerical models

3.1 Mesh modelling

The model geometry was created following an experimental study in which the areas downstream of the intake, called aft-stream and shown in figure 2, were interested. The upstream region is twice the intake diameter, D, between the inlet and the centre of the intake, whilst the downstream region is five times the intake diameter in the streamwise direction from the intake. The intake tube is 1.5D long and the fluid domain from the surface is 3D as shown in figure 3. The streamwise, spanwise and normal directions to the surface are formed on a right handed co-ordinate system, shown as x, y and z, respectively. The mesh structures are shown in figures 4.

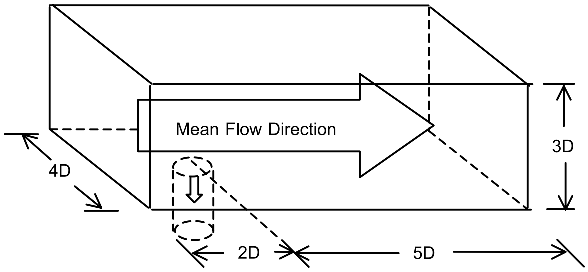


Figure 3 Geometry domain size, shows the mean flow direction, the size of domain in the streamwise, spanwise and normal to the flat surface directions.

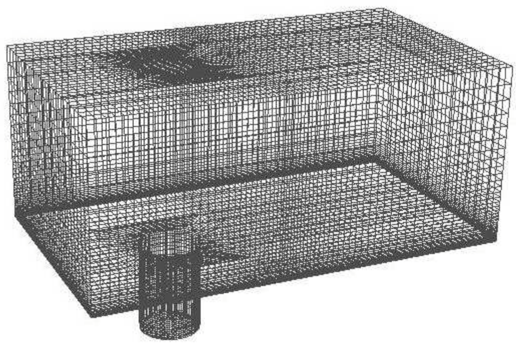


Figure 4 Mesh structure shows the densely mesh in the wall region and near the intake.

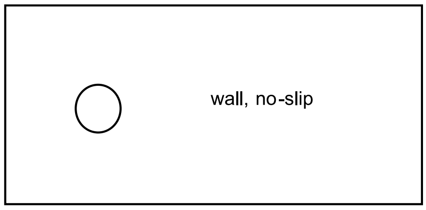
3.2 Boundary conditions

Surface condition

There are two groups of wall surfaces are specified in the problem; one is the surface of the flat plate, other is the surface of the intake tube. Both groups of surface are set as no slip condition which the velocity in all directions on the wall equal zero, shown in figure 5a.

Outlet flow

There are two-outlet flow boundaries in this simulation, called x-outlet and y-outlet, respectively. The x-outlet boundary condition is set as free surface, whereas the y-outlet boundary condition is defined as the uniform velocity constant of 7.4 m/s, shown in figure 5b.



a)

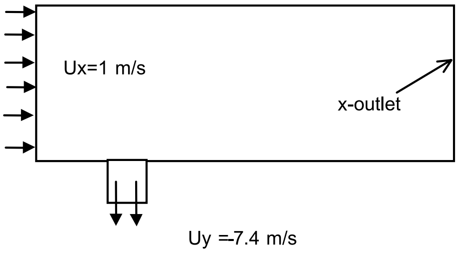


Figure 5 Boundary conditions setting a) z-x view, b) x-y view

4. Results

Flow visulisation

The flow visualisation experimental results are presented in figure (6) which shows the four different regions downstream of the intake at y/D = 0.05. The downstream flow figures are focused. Figures (7) visualise the results in the downstream flow using turbulence model coarse DNS, k-Ɛ RNG and LES. They show the streamlines for downstream starting from the line behind the intake.

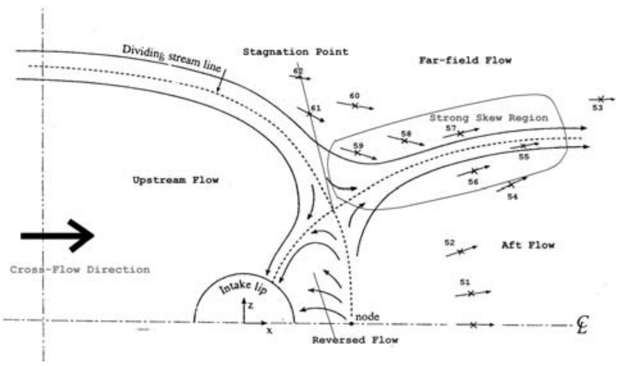
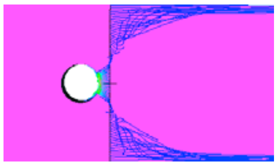
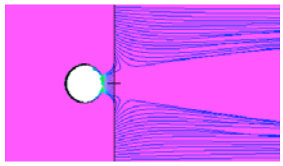


Figure 6 The sketch of flow visualization on y/D=0.05 Plane, in experimental work [1]

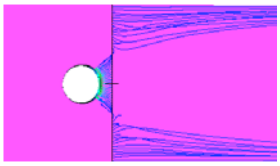
The streamline results for downstream for all the turbulence models represent the different flow regions; figure 7 shows three regions, the upstream flow, far-field flow, aft flow regions and the region of conversed flow, far-field flow and aft-flow regions. They are in agreement, in the direction of local flow, with the experimental results. However the distinguishing feature of the results of the streamlines downstream can be observed that on the y/D =0.05 plane the streamline flow pattern from the coarse DNS and LES are similar and show the complex flow in the region of meeting between different boundary layers, whilst the results from k-Ɛ RNG model can not show clearly. On the y/D = 0.1 plane, there are not many differences in the results of the streamlines for downstream for all of the turbulence models. This implies that the effect of turbulence modelling is dominant in the near wall regions.



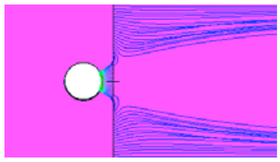
a) Coarse DNS (y/D = 0.05 plane)



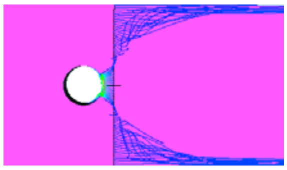
d) Coarse DNS (y/D = 0.1 plane)



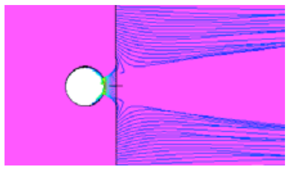
b) k-Ɛ RNG (y/D = 0.05 plane)



e) k-Ɛ RNG (y/D = 0.1 plane)



c) LES (y/D = 0.05 plane)



f) LES (y/D = 0.1 plane)

Figure 7 The streamlines downstream for different turbulence models from the line at the rear of the intake. y is distant from the flat surface in the normal to flat surface direction.

Velocity Profile

The velocity profile was also studied for the evaluation of different turbulence models, compared to the experiment, the data can be divided in two regions, one is the region of aft-flow, nearby the symmetric axis, which is far from the region of the meeting of the different boundary layers (points 51 and 52); and the other is the region near the meeting of the different flow boundary layers (points 53 to 60), as shown in figure 6. The results show that the k-Ɛ RNG does not give a good agreement with the experimental work in all the investigation points or both regions, whilst the coarse DNS and the LES model give a good agreement with the experimental in the aft flow region. However, in the region of the meeting of the different boundary layers the results from coarse DNS and LES do not give a good agreement with experimental data, especially in near wall regions as shown in figure 8. Moreover, the shape of velocity profile in the different regions show that in the region of the aft-flow where far from the meeting of different boundary layers, the velocity profiles are less steep than in the region of the meeting between the different boundary layers. This implies that the boundary layers of the flow in aft flow region are dominated by viscosity. Therefore the flow in this region can be assumed to be a new boundary layer or a laminar region.

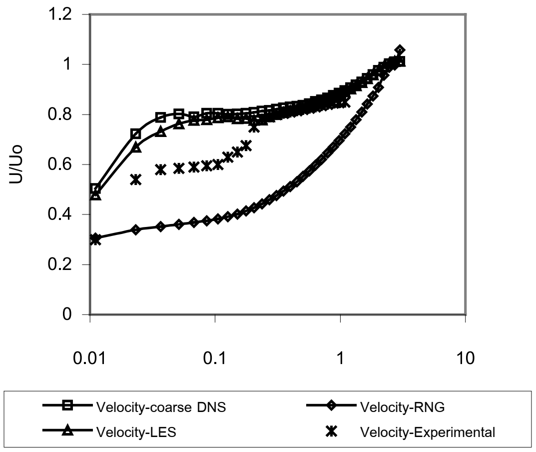


Figure 8 Velocity profile in the area of meeting of different flow boundary layers and different turbulence models

Skew Boundary layer

In the turbulence model investigation, although at point 17 and 28, where it is assumed to be in aft-flow region, even though the velocity profiles of the coarse DNS and LES agree with experimental results, the results of the skew boundary layer in this region does not agree with experimental results. Furthermore in the region of the meeting of different flow boundary layers, the results of all modelling do not agree with experimental results. The skew boundary layer profiles are shown in figures 9. However the simulation results agrees with the experimental results in a point that the simulation results within the region of the meeting of the different flow boundary layers have a stronger skew boundary layer than the region away from the meeting region. This is in agreement with experimental results. As same as the results of velocity profiles, the different between the LES and coarse DNS results can be shown in the region near the meeting between different boundary layers. The disagreement of skew boundary layer profiles of the numerical to experimental results implied that it is more complicate to predict the skew boundary layer profiles than the velocity profiles.

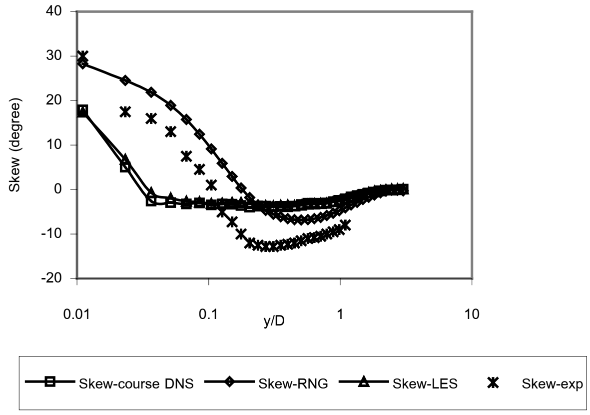


Figure 9 Skew angle profile in the area of meeting of different flow boundary layers and different turbulence models

5. Discussion

One of the main objectives of this study is to establish effective computational procedures for investigating the flow generated by an intake in a cross-flow. The investigation of three different turbulent models, the Reynolds average k-ε RNG, coarse DNS and LES with the Smagorinsky sub-grid scale model, are carried at with a mesh resolution of approximately 100,000 nodes in the FEM code, FIDAP. The flows are turbulent shear flow with the pressure gradients in all directions. The k-ε RNG, which used to simulate turbulent, shear flows in the past and available in the present code was initially tested. DNS was applied, using the laminar approach which does not model any scales of turbulence, but solves it directly; however the mesh resolution is too coarse to solve the Kolmogorov scale, it is therefore called coarse DNS. LES is used as a compromise between the Reynolds average and DNS models, in which the large scales of turbulence are solved directly whilst the small scales are modelled. The modelling of the small scale in LES is called the “subgrid scale model” and in the present study the Smagorinsky subgrid scale was used. The Smagorinsky model [8] calculates the small scales of turbulence, based on the eddy viscosity model in which the viscosity term depends on the Smagorinsky constant, the length scale (the grid size) and strain rate of local flow.

The k-ε RNG model with the law of the wall in the near wall region shows disagreement in terms of velocity profiles in the near field region (near the surface) compared to the experimental data. The coarse DNS and LES with Smagorinsky sub-grid scale model do agree. The probable reasons for the unsuitability of the k-ε RNG model to predict the present flow are:

i) The flow is anisotropic turbulence flow: the k-ε RNG turbulence model is based on the eddy viscosity turbulence model which calculates all scales of turbulent as isotropic, whilst the large scale of the shear flow near the surface are anisotropic.

ii) There are different boundary layers in the domain which implies different flow characteristics. The k-ε RNG model attempts to model all the flow with a single model with a set of non-universal empirical constants.

iii) Transient behaviour: The RNG k-ε equations are time averaged model where the detailed information about fluctuations is lost, whilst the turbulence flow is a time dependent phenomena.

Furthermore, the tests show that the results of coarse DNS and LES give a better agreement with the experimental results compared to k-ε RNG in all regions. The LES give better agreement with experimental results in the region of strong skew boundary layers.

6. Conclusion

In conclusion of the results of the evaluation of turbulence models to the experimental results, the simulation results from the turbulence models, coarse DNS, k-ε RNG and LES were divided into three parts; the flow visualisation, the velocity profile and the skew boundary layer studies. The flow visualisation studies show the results from all turbulence modelling can represent the four different flow boundary layers on the plane parallel and near the surface and agree with the experimental results. However, in the region of meeting between the different flow regions the coarse DNS and LES show that the flow is complex, whilst the k-ε RNG did not show the complex flow in the region of meeting between the different flow regions very clearly. The degree of complex flows in the meeting of different boundary layers also can be represented in the skew boundary layers study by the stronger skew in the boundary layers. In the velocity profile studies, the results from coarse DNS and LES show very good agreement with experimental data especially where the skew in the boundary layers are not strong. In the region of meeting between the different boundary layers where represented the strong skew boundary layers, the velocity profile from the LES were closer to experimental results than coarse DNS. On the other hand, the k-ε RNG model fails to predict the velocity profiles, compared to the experimental data in whole domain.

Regarding to the results in this test and refer to the disadvantages of DNS and the advantages of LES. The LES model is the most appropriate of the flow generated by an intake exposed to a cross-flow. It can be implied that the choosing of the modelling in turbulence flow have to be special cared to correct the flow characteristic, especially the complex turbulence flow. The upstream boundary condition also is the topic for more investigations [9].

Acknowledgement

Thanks for Fluid Mechanic Research Group, University of Hertfordshire, UK, especially Profeesor A.E. Holdo and Dr. G. De-With for the supports in this work.

Reference

[1] Hold∅ A.E., O'Brien and Calay R.K., 1997, Flow generated by the interaction of an inlet and a cross flow, ASME fluids engineering division summer meeting, FEDSM97-3289.

[2] O'Brien M., 1996, Aircraft engine intake interaction with a cross-flow, PhD. thesis, University of Hertfordshire, UK.

[3] Macmanus D.G. and Eaton J. A., 1996, Predictions and observations of the flow filed induced by lamina flow control microperforations, Experimental thermal and fluid Science, vol. 13, pp. 395-407

[4] O'Brien M. and Hold∅ A.E., 1996, Visualisation of the flow pattern generated by the interaction of a circular cross-section inlet and a cross-flow, Journal of wind engineering and industrial aerodynamics, vol. 59, pp. 95-102.

[5] Eaton J.K., 1995, Effects of mean flow three dimensionality on turbulent boundary layer structure, AIAA journal, vol. 33, no. 11, pp. 2020-2025.

[6] Yakhot V., Orszag A., Thangam S., Gatski T.B. and Speziale C.G., 1992, Development of turbulence models for shear flows by a double expansion technique, Physics of Fluids A, vol. 4, no. 7, pp. 1510-1520.

[7] FIDAP Theory Manuals, 1999, FLUENT Incorporated

[8] Smagorinsky J., 1963, General circulation experiments with the primitive equations: 1. The basic experimental, monthly weather review, vol. 91, pp. 99-164.

[9] Janyalert-adun A., Hold∅, A.E. and Wakes S.J., 2000, Evaluation of the effect of Upstream Boundary Condition on the Modelling of an Intake Exposed to a Cross-Flow, Proceedings of the third international symposium on turbulence, heat and mass transfer, Nagoya, Japan, April 2-6, 2000, pp. 981-988.