Study of Turbulence in a Establishedflow

SARI HASSOUN Zakaria, ALIANE Khaled, Mustapha Henaoui and BENAHMED Lamia

***Abstract*—The study of flow around obstacles is divided into three different positions: relative to the obstacle in front of the obstacle, the obstacle and downstream of the latter. The behavior of the fluid downstream of the obstacle is less known and physical and numerical modeling is being given the existence of recirculation zones with their complex behavior.**

The objective of this work is to propose a three-dimensional study unsteady with the computer code ANSYS CFX-13. The turbulence model SST-kW is used for solving the Navier-Stokes equations averaged in order to seek the flow control means around the obstacle.

***Keywords*— Turbulent flow; obstacle; finite volume; method Kω-SST; ANSYS -CFX.**

I. Introduction

The generation of the turbulence within a channel is usually caused by the presence of baffles and obstacles.

in the presence of obstacles fluid flow, are widely used in industry, and their application are extremely diverse. We can meet in the case of environmental problems related to the dispersion of pollutants through the cities and the effects of wind on buildings, ventilation road tunnel, the cooling fins of heat flares [1], the baffles in heat exchangers or solar collectors, urban pipelines, etc.

The study of the flow around obstacle may be following two approaches: digital approach where the flow is simulated based on a mathematical model; and the experimental approach where one must acquire measurements on the test bench.

- The digital approach is to discretize the governing equation of the flow by a digital method using different formulations [2, 3].
- The experimental approach can be treated in two aspects, the qualitative aspect where visualization techniques are used to analyze the different nascent vortices within the flow [4]. The second aspect is the aspect quantitative where recirculation zones downstream of the obstacle are quantized function of the dynamic parameters of the flow tells that the Reynolds number [5].

A literature search shows that many Studies were conducted to control behavior unfavorable linked to flows separated downstream of the barrier, examples include the work of DipankarChatterjee and BittagopalMondal [6], Roos and Kegelman [7] Kiya et al. [8], [9], Sigurdson [10] Chun Sung and [11], Sung et al. [12] S. Kang, Choi and H. [13].

SARI HASSOUN Zakaria, ALIANE Khaled, Mustapha Henaoui and BENAHMED Lamia, Mechanical Engineering, Tlemcen University, Algeria.

II. Problematic

In this work, we will study the vortex structures near the Lower and upper wall of the channel while it integrates the wall of law in a horizontal channel containing several obstacles in the formstairs.

The model of turbulence used in this study is the SST -Kω.

III. Equations Governing

A. Averaged equations

The averaged equations of conservation of mass, momentum and energy are respectively:

- The mass conservation equation

$$\frac{\partial \rho}{\partial t}+\frac{\partial(\rho U_i)}{\partial x_i}=0 \qquad (1)$$

Or

$$\frac{d\rho}{dt}+\rho\left(\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}\right)=0 \qquad (2)$$

- Conservation equations of amount of movement

$$\frac{\partial \vec{V}}{\partial t}+u\frac{\partial \vec{V}}{\partial x}+v\frac{\partial \vec{V}}{\partial y}+w\frac{\partial \vec{V}}{\partial z}=-\frac{1}{\rho}\overrightarrow{grad}P+\upsilon\nabla\vec{V} \qquad (3)$$

Or

$$\frac{\partial U_i}{\partial t}+U_j\frac{\partial U_i}{\partial x_j}=-\frac{1}{\rho}\frac{\partial P}{\partial x_i}+\upsilon\frac{\partial^2 U_i}{\partial x_j\partial x_j} \qquad (4)$$

- Energy Equations

$$\frac{\partial \overline{T}}{\partial t}+\overline{U}_j\frac{\partial \overline{T}}{\partial x_j}=\frac{\partial}{\partial x_j}\left[\left(\frac{\mu}{\Pr}+\frac{\mu_t}{\Pr_t}\right)\frac{\partial \overline{T}}{\partial x_j}\right]+\frac{1}{\rho C_P}\Phi \qquad (5)$$

B. Model SST-Kω

The model SST (Shear Stress Transport) is derived from the Standard k-ω model. This model combines the robustness and accuracy of the formulation of the k-ω model in the near-wall region with the k-ε model and all its types for the free flow away from the wall.

The definition of the turbulent viscosity is modified to take into account the transport of turbulent shear stresses.

https://doi.org/10.17758/URUAE16.AE08221028

The formulation of the two-equation model is:

$$\rho \frac{\partial k}{\partial t} + \rho \bar{U}_j \frac{\partial k}{\partial x_j} = \tilde{P}_k - \rho C_\mu \omega k + \frac{\partial}{\partial x_j}\left[\left(\mu + \mu_t/\sigma_k\right)\frac{\partial k}{\partial x_j}\right] \quad (6)$$

Specific dissipation rate:

$$\rho \frac{\partial \omega}{\partial t} + \rho \bar{U}_j \frac{\partial \omega}{\partial x_i} = 2\alpha\rho S_{ij} S_{ij} - \beta\rho\omega^2 + \frac{\partial}{\partial x_j}\left[\left(\mu_t + \sigma_\omega \mu_t\right)\frac{\partial \omega}{\partial x_j}\right] + 2\left(1 - F_1\right)\rho\sigma_{\omega 2}\frac{1}{\omega}\frac{\partial k}{\partial x_j}.\frac{\partial \omega}{\partial x_j} \quad (7)$$

The blend function F_1 is defined by:

$$F_1 = \tanh\left\{\left\{\min\left[\max\left(\frac{\sqrt{k}}{C_\mu \omega L}, \frac{500\upsilon}{L^2\omega}\right), \frac{4\rho\sigma_{\omega 2} k}{CD_{k\omega} L^2}\right]\right\}^4\right\} \quad (8)$$

Eddy viscosity is given by

$$\upsilon_t = \frac{\alpha_1 k}{\max\left(\alpha_1 \omega, \sqrt{2}\, S_{ij} F_2\right)} \quad (9)$$

The second blending function is defined by:

$$F_2 = \tanh\left[\left[\max\left(\frac{2\sqrt{k}}{C_\mu \omega L}, \frac{500\upsilon}{L\omega^2}\right)\right]^2\right] \quad (10)$$

To prevent the accumulation of turbulence stagnation regions, limited production is used:

$$\tilde{P}_k = \min\left(P_k, 10.C_\mu \rho k\omega\right) \quad (11)$$

$$P_k = \mu_t \frac{\partial U_i}{\partial x_j}\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right) \quad (12)$$

The model constants are calculated using the mixing function F_1 :

$$\phi = F_1\phi_1 + \left(1 - F_1\right)\phi_2 \quad (13)$$

The values of the model constants are:

C_μ =0.09, α_1 =5/9, α_2 =0.44, β_2 =0.0828, σ_{k1} =0.85, σ_{k2} =1.0, $\sigma_{\omega 1}$=0.5, $\sigma_{\omega 2}$ =0.856

C. Geometry and Boundary conditions

The geometry of the problem considered (Fig. 1) is three-dimensional, has three electronic components in the form of stairs, mounted in a horizontal channel length (L) and height (H).

The boundary conditions of the treaty problem are given as follows, as shown in Figure (1):

A constant speed is given to the channel input (U = 7 m / s), the velocity is zero the lower and upper walls of the channel and above the obstacle. The transverse velocity must be zero at the entrance and at the walls:

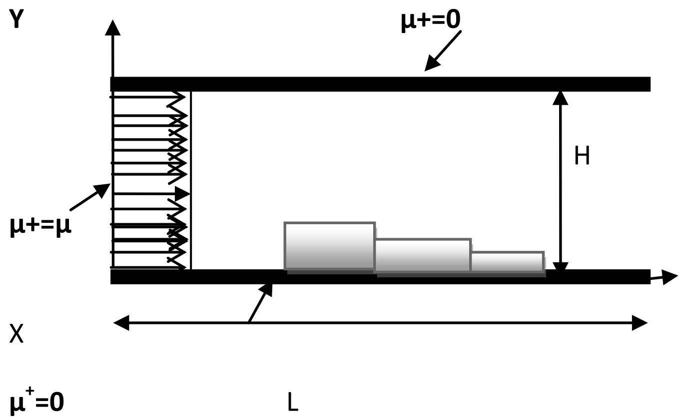

Fig. 1 - Geometry of computationaldomain

IV. RESULTS AND PERFORMANCES

A. Component of velocity

Figure 2 shows the contours of longitudinal speed in the direction of flow whereitisclearthat the flow isstronglyaccelerated in the zone above obstacles

It is obvious that the bridge by its presence in the channel reduces the flow area causing a strong acceleration of the flow.

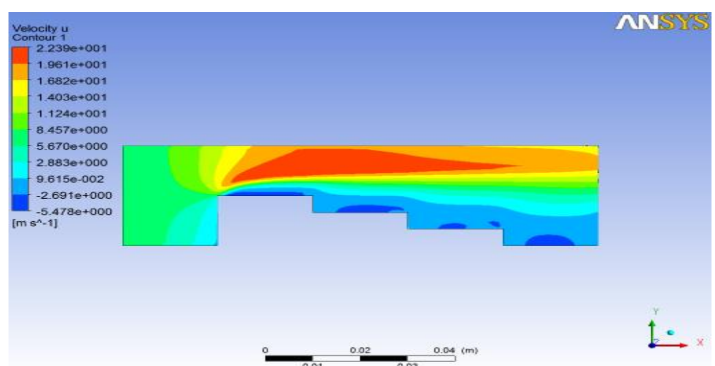

Fig.2 Contours of the longitudinal velocity,modelSST-Kω

Figure 3 shows the contours of the transverse velocities in the direction of flow as the speeds reached maximum values before the first obstacle. By against downstream of each staircase can be seen very low speeds.

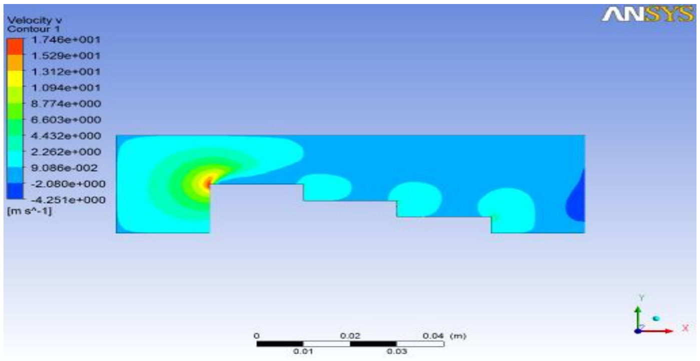

Fig.3 Contours of the transverse velocity , model SST-Kω

B. Turbulent kinetic energy

Figure 4 watches the contours of the kinetic energy of turbulence along the direction of flow, the high values are observed over stairs.

The rate of energy is relatively low downstream obstacles.

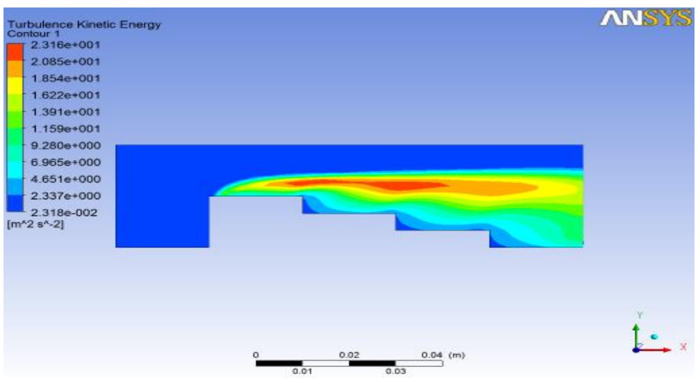

Fig.4 Turbulent kinetic energy, Model SST-kW

CONCLUSION

This study has allowed us through a 3D simulation to analyze and understand a number of important physical aspects. In this type of flow, especially we tried to emphasize the role of the presence of obstacles in the channel, the distribution of dynamic and thermal exchanges. The analysis of simulation results confirms that:

-The presence of an obstacle in the flow leads to an increase of the dynamic exchange and thus allows the improvement of heat transfer.

- Interaction between recirculation and main flow generates a high turbulence, it is marked in the intense velocity gradients and areas of high curvature trajectories upstream of the disturbance.

REFERENCES

[1] K.Aliane, O.Sebbane, A.Hadjoui, “Etude dynamique des modèles de refroidissement des aubes des turbomachines”, Proceedings du 11èmesjournée InternationalesdeThermique, Alger(Algerie),21-23Juin2003,(2003)315-320.

[2] A.Hadjoui, O.Sebbane, K.Aliane, A.Azzi,“Etude de l’apparition des zones tourbillonnaires au sein d’un écoulement confronté à des obstacles se trouvant à l’entrée d’un canal”, proceedings du 9ème Congrès de la Société Française de Géniedes Procédés,Saint-Nazaire,(France),9-11septembre2003,(2003)224-229.

[3] G. Bitsuamlak, T.Stathopoulos,C.Bedard, “Effects of upstreamtwo-dimensional hills on design wind loads: a computational approach.”, Wind and Structures, 9 (1) (2006)37–58.
https://doi.org/10.12989/was.2006.9.1.037

[4] O.Sebbane, A.Hadjoui, K.Aliane, A. Azzi, “Nouvelle méthode de visualisation des écoulements à très grand nombre de Reynolds”, proceedings du 9èmeCongrès de la Société Française de Génie des Procédés,Saint-Nazaire, (France), 9-11 septembre 2003,(2003). 259-264.

[5] K.Aliane, O.Sebbane, A. Hadjoui, A.Azzi, R.Saim, “Contribution à l’étude de l’influence de la distance inter-obstacle sur la génération des zones tourbillonnaires au sein d’un écoulement”, Proceedings du 1erCongrès International deMécanique, Constantine, Algérie, 14-16 Décembre 2002, (2002),500-507.

[6] C. Dipankar, M. Bittagopal, Control of flow separation around bluff obstacles by superimposed thermal buoyancy. International Journal of Heat and Mass Transfer, Vol. 72, May 2014, pp 128-138, 2014.
https://doi.org/10.1016/j.ijheatmasstransfer.2014.01.013

[7] F. W. Roos J.TKegelman, Control of coherent structures in reattaching laminar and turbulent shear layers. AIAA vol. 24, pp. 1956-1963, 1986.

[8] M. Kiya., M. Shimizu, O. Mochizuki., Y. Ido, H. Tezuka, Active forcing of an axisymmetric leading-edge turbulent separation bubble. AIAA, pp. 32-45. vol. 93 1993.
https://doi.org/10.1108/eb059389

[9] M. Kiya., M. Shimizu, O. Mochizuki., Sinusoidal forcing of a turbulent separation bubble. J. Fluid Mech. Vol. 342, pp.119-139, 1997.
https://doi.org/10.1017/S0022112097005521

[10] L.W.Sigurdson, The structure and control of a turbulent reattaching flow. J. Fluid Mech. Vol. 298, pp. 139-165, 1995.
https://doi.org/10.1017/S0022112095003259

[11] K.B. Chun , H.J. Sung, Visualization of a locally-forced separated flow over a backward-facing step. Exp. Fluids vol. 25, pp. 133-142, 1998.

[12] H.J. Sung, Y.M. Chung, M.Kiya, Vortex simulation of leading edge separation bubble with local forcing. Fluid Dyn. Res. Vol. 18, pp. 99-115, 1996.
https://doi.org/10.1016/0169-5983(96)00006-8

[13] S. Kang, and H.Choi, Suboptimal feedback control ofturbulent flow over a backward-facing step. J. Fluid Mech.vol. 463, pp. 201–227, 2002.
https://doi.org/10.1017/S0022112002008728

Z. Sari-hassoun was born in 24-07-1978 in Tlemcen-Algeria; he received the M.S. degree from the University of Tlemcen in 2001, and the Magister degree in Mechanical Engineering from University of Tlemcen, Algeria in 2008.

He is an Assistant Professor of Mechanical Engineering at Ain Temouchent University, Algeria. His research focuses on the momentum and heat transfer mechanisms and fluid dynamic’s.

He is currently a Researcher with the “Mechanical Computational Laboratory: MECACOMP”, Tlemcen, Algeria.

K. Aliane was born on 20-04-1972 in Tlemcen-Algeria; he received (with honors) the M.S. degree from the University of Tlemcen in 1996. Then, he received his Magister and PhD degree in Mechanical Engineering from University of Tlemcen, Algeria in 2000 and 2008 respectively. In 2011, he received the ability degree to direct researches from the Department of Mechanical Engineering of Tlemcen.

Currently, he is professor of Mechanical Engineering at Tlemcen University, Algeria.

His research focuses on the momentum and heat transfer mechanisms in complex geometry applied to electronics cooling components, heat exchanger and solar collectors.

He is currently a Researcher with the ‘‘Mechanical Computationnel Laboratory, Tlemcen, Algeria) were he has developing a technique’s for Passive control of the turbulent flow over obstacles