ISSN 2320-6802

INTERNATIONAL JOURNAL FOR ADVANCE RESEARCH IN ENGINEERING AND TECHNOLOGY

WINGS TO YOUR THOUGHTS…..

NUMERICAL SIMULATION OF BACKWARD FACING STEP FLOW

Stephane Ngako Emaga[1], Ruben Mouangue[2*], Francis Djanna[3], DenisTcheukam-Toko[4], Alexis Kuitche[5]

[1]Department of Electric, Energy and Automatism Engineering, ENSAI of Ngaoundéré,
P.O. Box 455 Ngaoundere, Cameroon
[2]Department of Energetic Engineering, University Institute of Technology, Ngaoundere
P.O. Box 455 Ngaoundere, Cameroon
[3]Department of Thermal Energetic Engineering, University Institute of Technology, Douala
P.O. Box 8698 Douala, Cameroon
*ruben.mouangue@univ-ndere.cm

Abstract: *This work is carried out to numerical simulation of incompressible flow over a backward-facing step. In the consider geometry, the flow present the recirculation zones that its length depends on the Reynolds number. This study proposes to use three turbulence models (k-ε realizable, k-ε linear production and Rij-ε) for predicting the reattachment length of the flow. The results obtained from these models are compared with experimental results of [1] and [2]. This comparison shows that the models used accurately (≤ 4.44%) predict the reattachment length for Reynolds numbers range of 100 ≤ Re ≤ 450. Beyond this limit, a significant difference (≥ 17 %) is found.*

Keywords: *Turbulence models, backward-facing step, incompressible flow, reattachment length.*

1. INTRODUCTION

The evaluation of parameters control of turbulent flow remains today a great interest to the scientific community. Indeed, the mastery of these parameters may help to understand certain phenomena such as sediment transport, erosion, combustion or even the design of hydraulic structures. The phenomena of flow separation and the swirls formation caused by sudden changes in flow section are often observed [3]. In many cases, the separation of fluid is undesirable and leads to a decrease in pressure and energy losses. However, there are cases where the flow separation is to encourage: for example stabilizing the burner flame or homogenizing a mixture in turbulence [4]. In addition, the recirculation zones involve a significant increase in drag. It is therefore important to understand the mechanisms of detachment and reattachment before designing any hydraulic structure. These mechanisms cannot be described with an analytical approach, experimental and numerical approaches are then the only way for this study.

This work focuses on the numerical prediction of detachment and reattachment of turbulent flows phenomenon that occur in most industrial applications based on standard statistical modeling. Among the most frequently geometries used for the study of separated flows, there are the backward-facing step geometries which is one of the most popular test cases used to evaluate a turbulence model's accuracy in fluid dynamics [3]. This is due to the relative simplicity of its geometry and the existence of experimental results. Several experimental studies [1]-[2] and theatricals [5]-[10] have been performed on this configuration but [7] resumed by [5] indicated that the most turbulence models can not accurately predict many important features of the backward-facing step flow, such as the reattachment length, the streamlines near the reattachment points and skin friction coefficient.

It will be the question for us to determine the position of reattachment point, velocity profiles at different sections and compare them with available experimental results to assess the accuracy of different turbulence models used.

Our manuscript begins with an introduction who reported on a few previous works undertaken on the prediction of the features of the backward-facing step flow. Thereafter turbulent models used are presented as well as the discretization method. The results obtained and discussions come from conclusion and perspectives to this work.

2. MATHEMATIC FORMULATIONS

2.1 Governing equations

The Reynolds-averaging principle is applied to the Navier-Stokes equations. After performing the averaging, the continuity and momentum equations can be shown as follows:

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho \bar{u}_i)}{\partial x_i} = 0 \quad \textbf{(1)}$$

$$\frac{\partial(\rho \bar{u}_i)}{\partial t} + \frac{\partial(\rho \bar{u}_i \bar{u}_j)}{\partial x_j} = -\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j}(\mu \frac{\partial \bar{u}_i}{\partial x_i}) + \frac{\partial R_{ij}}{\partial x_j} \quad \textbf{(2)}$$

Where $R_{ij} = \rho \overline{u'_i u'_j}$ the Reynolds is stresses.

Several models are used to determine an approximate solution of these equations.

2.2 k-ε models

These models use the Boussinesq approximation (3) to model the Reynolds stress tensor using an eddy viscosity (μ_t) and to write the turbulent kinetic energy (k) and its dissipation rate (ε) equations:

According to this approximation:

$$R_{ij} = \rho \overline{u'_i u'_j} = \mu_t (\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i}) - \frac{2}{3} k \delta_{ij} \quad \textbf{(3)}$$

Where $\mu_t = \rho C_\mu \frac{k^2}{\varepsilon}$

2.3 k-ε realizable model of Fluent

The term "realizable" means that the model satisfies certain mathematical constraints on the Reynolds stresses, consistent with the physics of turbulent flows. The transport equations used by this model can be written [8]:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho k u_j)}{\partial x_i} = \frac{\partial}{\partial x_i}((\mu + \frac{\mu_t}{\sigma_k})\frac{\partial k}{\partial x_i}) + G_k + G_b - \rho\varepsilon - Y_M + S_k \quad \textbf{(4)}$$

$$\frac{\partial(\rho \varepsilon)}{\partial t} + \frac{\partial(\rho \varepsilon u_j)}{\partial x_i} = \frac{\partial}{\partial x_i}((\mu + \frac{\mu_t}{\sigma_\varepsilon})\frac{\partial \varepsilon}{\partial x_i}) + \rho C_1 S\varepsilon - \rho C_2 \frac{\varepsilon^2}{k + \sqrt{\nu\varepsilon}} + C_{1\varepsilon}\frac{\varepsilon}{k} C_{3\varepsilon} G_b + S_\varepsilon \quad \textbf{(5)}$$

Contrarily to other variants of this model C_μ is a function of the mean strain and rotation rates, the angular velocity of the system rotation, and the turbulence fields (k and ε)

The term G_k, representing the production of turbulence kinetic energy due to the mean velocity gradients; G_b is the generation of turbulence kinetic energy due to buoyancy; Y_M Represents the contribution of the fluctuating dilatation incompressible turbulence to the overall dissipation rate; S is the modulus of the mean rate-of-strain tensor; and C_2 are constants. σ_k And σ_ε are the turbulent Prandtl numbers for k and ε respectively. S_k And S_ε are user-defined source terms. In our case set to zero. The models constant are given by:

Table 1: Model constant

	C_1	C_2	$C_{1\varepsilon}$	σk	σε	η
Value	Max(0.43; η/(η+5)	1.9	1.44	1.0	1.2	S(k/ε)

The method used to solve the closed conservation equations is the finite volume [9]. It is to express the results of the study size (mass, energy, momentum) on a volume control and the Gauss's law is used to transform volume integrals into surface integrals. To locate the different variables, we use the concept of interlaced mesh where pressure, turbulent kinetic energy, dissipation, the Reynolds stresses are treated at the center of the control volumes, velocity are measured at the center of faces and the tensions shear are located at the corners. The advection terms are evaluated using the QUICK scheme [10]. The algorithm involves calculating a steady flow reducing unsteady terms of conservation equations. In this study, we use a SIMPLE procedure [11] which is to express the velocity at time (n+1) Δt function of $P^{(n+1)}$ from the conservation equations. We then obtain a Poisson equation to solve for pressure $P^{(n+1)}$ and combining with the conservation equation for momentum, we obtain a new velocity field at the time of (n+1) Δt satisfying the conservation equation for mass.

2.4 Linear Production Model of Saturne EDF

The transport equations used by the k-ε Linear Production model of code Saturn EDF are given by:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho k u_j)}{\partial x_i} = \frac{\partial}{\partial x_i}((\mu + \frac{\mu_t}{\sigma_k})\frac{\partial k}{\partial x_i}) + P + G - \rho\varepsilon + \Gamma(k_i - k) \quad \textbf{(6)}$$

INTERNATIONAL JOURNAL FOR ADVANCE RESEARCH IN ENGINEERING AND TECHNOLOGY
WINGS TO YOUR THOUGHTS…..

$$\frac{\partial(\rho\varepsilon)}{\partial t}+\frac{\partial(\rho\varepsilon u_j)}{\partial x_i}=\frac{\partial}{\partial x_i}((\mu+\frac{\mu_t}{\sigma_\varepsilon})\frac{\partial\varepsilon}{\partial x_i})+$$

$$\rho C_{1\varepsilon}\frac{\varepsilon}{k}(P+(1-C_{3\varepsilon})G)-\rho C_{2\varepsilon}\frac{\varepsilon^2}{k}- \quad \textbf{(7)}$$

$$\varepsilon\frac{\partial(\rho u_j)}{\partial x_i}+\Gamma(\varepsilon_i-\varepsilon)$$

Where:

$P=-\rho R_{ij}\frac{\partial u_i}{\partial x_j}$: is the generation of turbulence due to shear; $G=-\frac{1}{\rho}\frac{\mu_t}{\sigma_t}\frac{\partial\rho}{\partial x_i}g_i$: is the generation of turbulence due to gravity; Γ is a possible source term from the conservation equation for mass.

The models constant are given by:

Table 2: Model constant

	C_μ	$C_{1\varepsilon}$	$C_{2\varepsilon}$	σ_k	σ_ε	$C_{3\varepsilon}$
Value	0.09	1.44	1.92	1.0	1.3	0 or1

2.5 Rij-ε model of Saturne EDF

The transport equations used by the Rij-ε model of code Saturn EDF are given by:

$$\frac{\partial(\rho R_{ij})}{\partial t}+\frac{\partial(\rho u_j R_{ij})}{\partial x_i}=\frac{\partial}{\partial x_i}(\mu\frac{\partial R_{ij}}{\partial x_i})+P_{ij}+G_{ij}+ \quad \textbf{(8)}$$

$$\Phi_{ij}+d_{ij}-\rho\varepsilon_{ij}+\Gamma R^i_{ij}+ST_{R_{ij}}$$

$$\frac{\partial(\rho\varepsilon)}{\partial t}+\frac{\partial(\rho\varepsilon u_j)}{\partial x_i}=\frac{\partial}{\partial x_i}(\mu\frac{\partial\varepsilon}{\partial x_j})+d+ \quad \textbf{(9)}$$

$$\rho C_{\varepsilon 1}\frac{\varepsilon}{k}(P+G_\varepsilon)-\rho C_{\varepsilon 2}\frac{\varepsilon^2}{k}+\Gamma\varepsilon^i-ST_\varepsilon$$

The production meanshear and gravity terms can be written by:

$$P_{ij}=-\rho(R_{ik}\frac{\partial u_j}{\partial x_k}+R_{jk}\frac{\partial u_i}{\partial x_k})G_{ij}$$

$$=-\frac{3}{2}\frac{C_\mu}{\sigma_t}\frac{k}{\varepsilon}(r_i g_i+r_j g_i)$$

$$k=\frac{1}{2}R_{ll};d=C_\varepsilon\frac{\partial}{\partial x}(\rho\frac{k}{\varepsilon}R_{km}\frac{\partial\varepsilon}{\partial x_m});$$

With: $G_\varepsilon=\max(0,\frac{1}{2}G_{kk});r_i=R_{ik}\frac{\partial\rho}{\partial x_k};P=\frac{1}{2}P_{kk};$

$$d_{ij}=C_s\frac{\partial}{\partial x_k}(\rho\frac{k}{\varepsilon}R_{km}\frac{\partial R_{ij}}{\partial x_m});$$

dij is the turbulent diffusion term and Φij the correlation pressure-strain term

The model constants are:

Table 3: Model constant

Constant	C_μ	$C_{\varepsilon 1}$	$C_{\varepsilon 2}$	C_S	C_ε
Value	0.09	1.44	1.92	0.22	0.18

The principle of the resolution used for the two previous models is the method of prediction-correction. At each time step n, the conservation equation for momentum is first solving by making explicit pressure. We deduce a first velocity value (called predicted velocity), $u^{n+1/2}$. The continuity equation is finally treated to modify the velocity field predicted to be non-zero divergence, correcting the pressure (Poisson equation).

3. RESULTS

a. Computational domain and boundary conditions

The flow geometry and the coordinate system are shown in Figure 1. Computational domains Ω1 and Ω2 are such as Ω = [0, Lx] x [0, Ly]x[0, Lz] with Lx = 30h, Ly = 2h and Lz = 0 for Ω1; and Lx = 30h, Ly = 6h, and Lz = h for Ω2. An inlet section, Li = 10h is placed in x direction before the sudden expansion, whereas h is the step height. A uniform velocity profile imposed to the input domain (x = -Li) allow to obtain a Poiseuil profile perfectly established before the step. The Reynolds number (Reh) based on the step height and the rate expansion Er are the control parameters of the flow for this case.

$$R_{eh}=\frac{\rho Ue*h}{\mu};Er=\frac{Ly}{Ly-h}$$

Where Ue and h are respectively the inlet free-stream velocity and the characteristic length of the flow. With ρ and μ the dynamic viscosity and fluid density.

ISSN 2320-6802

INTERNATIONAL JOURNAL FOR ADVANCE RESEARCH IN ENGINEERING AND TECHNOLOGY

WINGS TO YOUR THOUGHTS…..

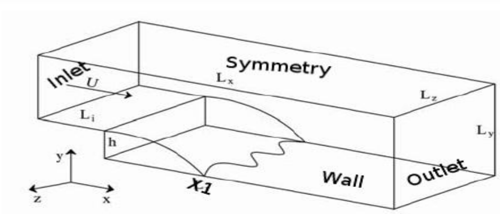

Figure 1: Computational domain and boundary conditions

The boundary conditions applied for the computations were as follows:

- Inlet: ux= Variable and uy=uz=0 ;
- Wall: ui=0;

Table 4: Table of parameters

experimental parameters [1]				
Reynolds number	Ux=Ue (m/s)	Number of cells	Convergence criterion	
			Pressure	Velocity
113,207	0,1579530	1200*40	10^{-6}	10^{-6}
169,811	0,2369441	1200*40	10^{-6}	10^{-6}
283,018	0,3948976	1200*40	10^{-6}	10^{-6}
396,226	0,5528651	1200*40	10^{-6}	10^{-6}
500	0,6976744	1200*40	10^{-6}	10^{-6}
750	1,0465116	1200*40	10^{-6}	10^{-6}
1000	1,3953488	1200*40	10^{-6}	10^{-6}
experimental parameters [2]				
5100	14,8225	750*60	10^{-4}	10^{-4}

b. Simulation parameters

We study here two configurations the one used by [1] with Er ≈ 2 and the one used by [2] with an expansion rate Er = 1.2. The other simulation parameters are taken to reproduce the best experimental conditions of the authors cited above.
The mesh of Ω1 and Ω2 corresponding respectively to experimental domain of [1] and that of [2] was made using the preprocessor Gambit [12] for the Fluent cases and with the preprocessor Gmsh [13] for the Saturn cases. A structured mesh uniformly spaced 80440 cells for the domain Ω1 and 48000 cells for the domain Ω2. This is to capture all the effects of turbulence in near-wall region.
The convergence criterion was set equal to 10^{-6} for the different parameters.

c. Comparison of numerical results and experimental data

In this section, we present the results of numerical simulation models using k-ε realizable model of fluent, k-ε linear production and Rij-ε models of code Saturn EDF. These results are simultaneously confronted with experimental data of [1] and [2] where Reynolds number based on the step height (h) and the average flow velocity is Reh = 5100.
Figure 2 below shows the dimensionless reattachment length (x/h) as function of Reynolds number. It may be noted that in purely laminar regime (Re ≤ 420) experimental data and numerical results of different models are fairly consistent, beyond this value, the disparities between the different results observed. These differences are explained by the fact that during the experiment the initial two-dimensional flow (2D) gradually becomes three-dimensional (3D) [1] while the various calculations are performed in 2D. Nevertheless, the results obtained with the Saturn code have the best agreement with experimental measurements while the result of TEACH code (parent of Fluent code) used by [1] get further away.

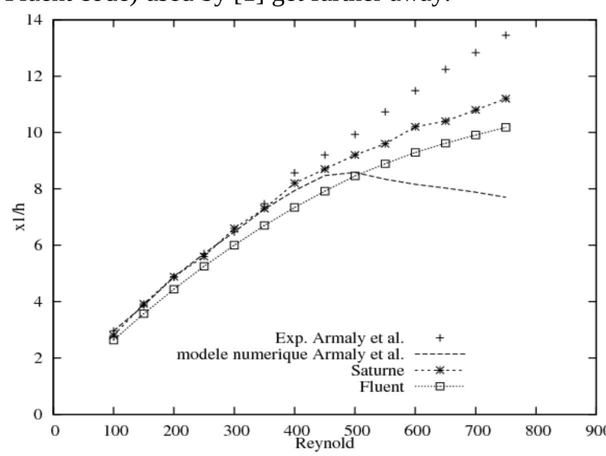

Figure 2: Comparison of experimental and numerical measurements for the reattachment length for Re <800

The mean stream wise velocity (normalized by the reference flow velocity (Ue) at the location x/h=4.0, 6.0, 10.0 and 19.0 are shown in Figure 3 below. In general we observe for each section that the different numerical models underestimate the near-wall velocity. In addition further away from the wall, the results provided by the different models are in agreement with experimental measurements of [2]. In particular, in the x/h = 4 section (just after the step) it is observed that the velocity profile obtained with the k-ε model linear production and experimental data show a distortion induced by the secondary current under the step. As for the k-ε realizable and Rij-ε models, they do not predict this effect in this region.

www.ijaret.org
Vol. 1, Issue X, Nov.2013
ISSN 2320-6802

INTERNATIONAL JOURNAL FOR ADVANCE RESEARCH IN ENGINEERING AND TECHNOLOGY

WINGS TO YOUR THOUGHTS.....

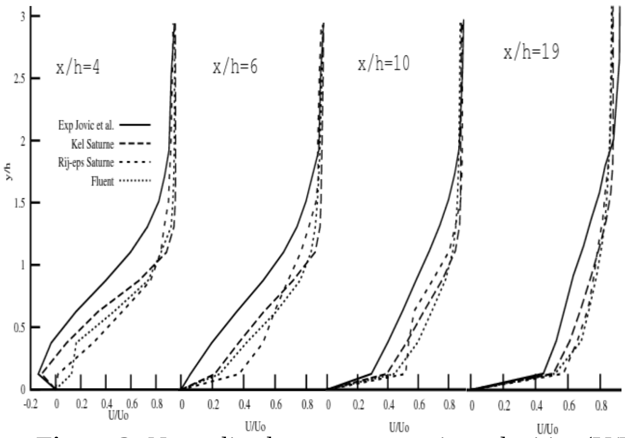

Figure 3: Normalized mean stream wise velocities (U/U0) at the locations x/h=4, 6, 10 and 19

4. CONCLUSION AND PERSPECTIVES

There has been talk for us to achieve a standard test with different models implemented in the code Saturn EDF and Fluent commercial code and compare the results with experimental data. The comparison shows that the k-ε linear Production model of Code Saturn EDF better predicted reattachment lengths (accuracy ≤ 4.44%) for a purely laminar flow regime. Near the walls, the models cannot predict the velocity of the flow. These discrepancies show that different turbulence models of Fluent and Saturn EDF codes used in this study should be strengthened to better predict the features of a backward-facing step flow. What will be the future investigations including the introduction of damping functions in the k-ε linear production model. Given that the application of this configuration in combustion which is our desired study field, will require even more numerical effort.

Table 5: Nomenclature

Small letter

X Longitudinal coordinate (m)

y Vertical coordinate (m)

z Transversal coordinate (m)

h Step height (m)

Capital letter

P Generation of turbulence due to shear

G Generation of turbulence due to gravity

U Mean velocity in streamwise direction $(m.s^{-1})$

U_i Vitesse de l'écoulement suivant l'axe i $(m.s^{-1})$

X_1 Mean reattachment length (m)

Greek letters

ν Kinematic viscosity of air $(m^2.s^{-1})$

μ Dynamic viscosity of air (Pa.s)

ε Turbulence dissipation rate

k Turbulence kinetic energy $(J.kg^{-1})$

ρ Air density $(kg.m^{-3})$

Dimensionless number

Re Step height Reynolds number

σ_k Turbulent Prandtl number for k

σ_ε Turbulent Prandtl number for ε

Special caracter

Ω_i Computational domain

S_{ij} Tenseur moyen du taux de contrainte

μ_t Turbulent viscosity

$\emptyset_{ij}$ Correlation pressure-strain term

δ_{ij} Kronecker symbol

d_{ij} Turbulent diffusion term

R_{ij} Reynolds shear stress

Lx Period in stream wise direction

Ly Period in normal direction

Lz Period in span wise direction

REFERENCES

[1] B. F. Armaly, F. Durst, J. C. Pereira and B. Schonung (1983). Experimental and theoretical investigation of backward facing step flow. J. Fluid Mech. vol.127; pp.473-496

[2] S. Jovic, D. Driver. (1994). Bacward-facing step measurements at low Reynolds number, Reh=5100, NASA Ames Research Center.

[3] Elham Rajabi and Mohammad R. Kavianpour (2012): Intelligent prediction of turbulent flow over backward-facing step using direct numerical simulation data Engineering Application of Computational Fluid Mechanics Vol. 6, No. 4, pp. 490-503

[4] M. K. Denham and M. A. Patrick. Laminar flow over a downstream-facing

[5] step in a two-dimensional flow channel. Trans. Inst. Chem.Engrs. vol.52 ; pp.361-367(1974)

INTERNATIONAL JOURNAL FOR ADVANCE RESEARCH IN ENGINEERING AND TECHNOLOGY
WINGS TO YOUR THOUGHTS…..

[6] Tae Seon Park and Hyun Gjin Sung (1995). A nonllinear low-Reynolds-number k-ε model for turbulent separated and reattaching flows Ill Flow field computations. Int. J. Heat Mass Transfer. Vol. 38, No. 14, pp. 2657-2666.

[7] K. Khuwaranyu and S. Putivisutisak : Combined low-Reynolds-number k–ω model with length scale correction term for recirculating flows ;Journal of Engineering and Technology Research Vol.1 (8), pp.171-180, November, 2009

[8] Ko S (1999): A near-wall Reynold stress model for backward-facing step flows. Journal of Mechanical Science and Technology 13(2), pp. 200-210

[9] T.-H. Shih, W. W. Liou, A. Shabbir, Z. Yang, and J. Zhu.A New k-ε Eddy-Viscosity Model for High Reynolds Number Turbulent Flows-Model Development and Validation. Computers Fluids, 24(3) : 227–238, 1995.

[10] S. V. Patankar (1980). Numerical heat transfer and fluid flow. Hemisphere. Washington, D, C. pp 115-120.

[11] B. P. Leonard and S. Mokhtari. ULTRA-SHARP Nonoscillatory Convection Schemes for High-Speed Steady Multidimensional flow. NASA TM 1-2568 (ICOMP-90-12), NASA Lewis Research Center, 1990

[12] C. Geuzaine et J. F. Remacle: Gmsh Reference Manual. 2012 http://www.gnu.org

[13] Gambit 2.2 Documentation: User's Guide. Fluent. Inc. (2006)

[14] EDF R&D. Code Saturne 2.0.4 tutorial, user's guide theory and programmer's. http://www.code-saturne.org, 2008

[15] Fluent 6.3.26 Documentation: User's Guide. Fluent. Inc. (2006).

[16] H. Naji, A. Rechia, G. Mompean. "Simulation numerique d'un ecoulement turbulent tridimensionnel" 18eme Congres francais de Mécanique, Grenoble 27-31 aout 2007.