BIUST Teaching, Research, & Innovation Symposium 2023 (TRDAIS 2023)
Botswana International University of Science and Technology
Palapye, Botswana, 18 - 19 September 2023
ISSN: 2521-2293

Study of two Turbulence Models in Predicting the Drag of a Bluff Body

M.D. Koontse and N. Subaschandar*
Department of Mathematics and Statistical Sciences
Botswana International University of Science and Technology,
Private Bag 16,Palapye, Botswana

Abstract

Drag reduction on a two dimensional D-shaped bluff body is numerically studied. In this research, mesh independence study is carried out at the same velocity value of $10m/s$. The effect of Reynolds number on the drag coefficient is also carried out. This was done by increasing the inlet velocity from $10m/s$ to $60m/s$ hence increasing the Reynolds number. The k-ϵ and SST turbulence models are used and also their suitability for predicting the drag is compared in this study. Preliminary results on drag forces are presented here. The preliminary results suggest that the drag values obtained for our two-dimensional bluff body is mesh independent. The results also show that the drag coefficient remains nearly constant with increase in Reynolds number. The study show that the SST turbulence model underpredicts the drag coefficient compared to the k-ϵ turbulence model.

Keywords: Turbulence model, drag, Bluff body.

1 Introduction

It is believed that the Navier-Stokes equations can be used to fully describe turbulent flows, but current limitations in computational resourses have made direct solution of the Navier-Stokes equations impractical for all but very simple flows at low Reynolds numbers. The quest for the ultimate turbulence model has been ongoing for nearly a century now. Early turbulence models were emperically formulated algebraic relations. As computer speed, software capability and efficient numerical schemes developed and numerical simulation evolved, differential equation based transport type turbulence models became the turbulence simulation methodology of choice. The use of transport type turbulence models has become standard practice for most engineering applications. Many current researchers are now solving the unsteady Navier-Stokes

*corresponding author: raos@biust.ac.bw

ISSN: 2521-2293

equations for large-scale or grid realized, turbulence and modeling the smaller or subgrid, turbulent scales that cannot be captured on the computational grid.[1]
The subject of drag reduction of a bluff body is an interesting problem with a wide range of applications. Because of the difficulties associated with theoretical analysis, the study of drag reduction has been almost entirely experimental.[2] The flow over a bluff body is a common occurance associated with fluid flowing over an obstacle or with the movement of a natural or an artificial body. Evident examples are the flows past an airoplane, a submarine and a road vehicle. At much lower Reynolds numbers, the flow over a bluff body is highly viscous and the force exerted on the body is mainly attributed to the skin friction. However, when the Reynolds number exceeds a critical value, vortex shedding occurs in the wake, resulting in a significant pressure drop on the rear surface of the body.[3]
The main purpose of this paper is to predict the drag of a bluff body using two turbulence models,namely, the k-ϵ turbulence model and the Shear Stress Transport(SST) turbulence model. This is done based on the effect of grid/mesh size and the effect of Reynolds number against the drag coefficient(C_d).Preliminary results are presented here.

2 Mathematical Formulation

The flow around a D-shaped 2-dimensional rectangular cylinder of length $l = 0.355$ and height $h = 0.25$ is considered in this work. The effect of mesh/grid independence and the effect of Reynolds number on the drag of a bluff body are studied.
The flow domain is two dimensional and the fluid is air. This flow is steady, fully turbulent,incompressible with constant viscosity and constant density.
Figure 1 below shows the two-dimensional sketch of the computational domain. L is the length of the domain and H is the height of the domain. Lower case h is the height of the D-shaped rectangular cylinder, l is the length of the cylinder and $r = \frac{h}{2}$ is the radius of the semi-cycle.

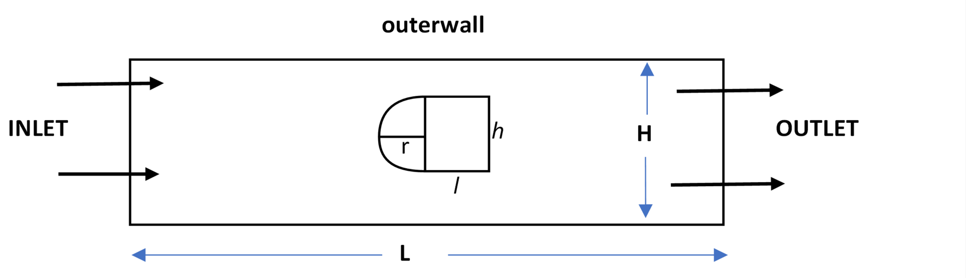

Figure 1: 2-D computational domain

The fundamental equations of fluid flow have been applied in this research. These equations results from the application of Newton laws to the moving fluid. The two dimensional form of

ISSN: 2521-2293

the transport equations for a general variable ϕ is given by

$$\frac{\partial(\rho\phi)}{\partial t} + \frac{\partial}{\partial x}(\rho u\phi) + \frac{\partial}{\partial z}(\rho w\phi) = \frac{\partial}{\partial x}\left(\Gamma\frac{\partial\phi}{\partial x}\right) + \frac{\partial}{\partial z}\left(\Gamma\frac{\partial\phi}{\partial z}\right) + S_\phi \tag{1}$$

where $\rho[kg/m^3]$ is the fluid density, $u(m/s)$ and $w(m/s)$ are the velocity components along the horizontal axis and along the vertical axis. Γ is the diffusion coefficient of ϕ and S_ϕ is the generation rate of ϕ per unit volume.
If ϕ in equation (1) was the velocity component we will now get the momentum conservation equations,The Navier Stokes Equations which for a two dimensional situation may be stated as

Horizontal component(x-direction):

$$\frac{\partial(\rho u)}{\partial t}+\frac{\partial}{\partial x}(\rho u^2)+\frac{\partial}{\partial z}(\rho wu) = \frac{\partial}{\partial x}\left[\Gamma\left(2\frac{\partial u}{\partial x} - \frac{2}{3}div\vec{V}\right)\right]+\frac{\partial}{\partial z}\left[\Gamma\left(\frac{\partial u}{\partial z} + \frac{\partial w}{\partial x}\right)\right]-\frac{\partial p}{\partial x}. \tag{2}$$

Vertical component(z-direction):

$$\frac{\partial(\rho w)}{\partial t}+\frac{\partial}{\partial x}(\rho uw)+\frac{\partial}{\partial z}(\rho w^2) = \frac{\partial}{\partial x}\left[\Gamma\left(2\frac{\partial w}{\partial z} - \frac{2}{3}div\vec{V}\right)\right]+\frac{\partial}{\partial x}\left[\Gamma\left(\frac{\partial u}{\partial z} + \frac{\partial w}{\partial x}\right)\right]-\frac{\partial p}{\partial z} \tag{3}$$

where $p[n/m^2]$ represents pressure and the diffusion coefficient in this case is given by

$$\Gamma = \mu + \mu_t$$

where $\mu(Ns/m^2)$ is the dynamic viscosity and μ_t is the turbulent viscosity.
The conservation of mass law, or continuity equation is also stated as:

$$\frac{\partial(\rho u)}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho w)}{\partial z} = 0 \tag{4}$$

Turbulence Modelling

This section presents the mathematical equations of two turbulence models studied in this research. Each model will be presented separately.

Shear Stress Transport Turbulence model(SST)

The SST turbulence model was originally used for aeronautical applications but has since made its way into most industrial, commercial and many research codes. The starting point for its development was the need for the accurate prediction of the aeronautical flows with strong adverse pressure gradients and separation. Over decades, the available models had consistently failed to compute these flows[4]. The SST turbulence model[6] represents a combination of the k-ϵ and the k-ω models. According to [4], the k-ω model is more accurate near the wall but presents a high sensitivity to the ω values in the free stream region, where k-ϵ model shows

ISSN: 2521-2293

a better behaviour. The SST model represents a blend of the two, through a weighting factor computed based on the nearest wall distance. The governing equations for the SST model are as follows;

$$\frac{\partial(\rho k)}{\partial t}+\frac{\partial(\rho uk)}{\partial x}+\frac{\partial(\rho wk)}{\partial z}=\bar{P}_k-\beta^*\rho\omega k+\frac{\partial}{\partial x}\left((\mu+\sigma_k\mu_t)\frac{\partial k}{\partial x}\right)+\frac{\partial}{\partial z}\left((\mu+\sigma_k\mu_t)\frac{\partial k}{\partial z}\right) \quad (5)$$

and

$$\begin{aligned}\frac{\partial(\rho\omega)}{\partial t}+\frac{\partial(\rho u\omega)}{\partial x}+\frac{\partial(\rho w\omega)}{\partial z}&=\frac{\partial}{\partial x}\left((\mu+\sigma_\omega\mu_t)\frac{\partial \omega}{\partial x}\right)+\frac{\partial}{\partial z}\left((\mu+\sigma_\omega\mu_t)\frac{\partial \omega}{\partial z}\right)+\frac{\alpha\bar{P}_k}{v_t}\\ &-\beta\rho\omega^2+2(1-F_1)\rho\sigma_{\omega 2}\frac{1}{\omega}\left(\frac{\partial k}{\partial x}\frac{\partial \omega}{\partial x}+\frac{\partial k}{\partial z}\frac{\partial \omega}{\partial z}\right)\end{aligned} \quad (6)$$

where ω is the frequency of dissipation of turbulent kinetic energy.
The production of turbulent kinetic energy is limited to prevent the build-up of turbulence in stagnant regions:

$$\bar{P}_k=min(P_k,10\beta^*\rho k\omega)$$

The weighting function F_1 is given by

$$F_1=tanh\left\{\left\{min\left[max\left(\frac{\sqrt{k}}{\beta^*\omega y};\frac{500v}{y^2\omega}\right);\frac{4\rho\sigma_{\omega 2}k}{CD_{k\omega}y^2}\right]\right\}^4\right\}$$

and

$$CD_{k\omega}=max\left(2\rho\sigma_{\omega 2}\frac{1}{\omega}\frac{\partial k}{\partial x_j}\frac{\partial \omega}{\partial x_j},10^{-10}\right)$$

where y represents the distance to the neighbour wall and v is the laminar viscosity. F_1 is zero away from the wall(k-ϵ model) and changes to unit inside the boundary layer(k-ω model), with a smooth transition based on y.
The turbulent viscosity is computed as;

$$v_t=\frac{a_1k}{max(a_1\omega;SF_2)}$$

where S is the invariant measure of the strain rate given by

$$S=\sqrt{S_{ij}S_{ij}} \quad ; \qquad S_{ij}=\frac{1}{2}\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)$$

and

$$F_2=tanh\left\{\left[max\left(\frac{2\sqrt{k}}{\beta^*\omega y};\frac{500v}{y^2\omega}\right)\right]^2\right\}.$$

ISSN: 2521-2293

The constants are computed as a blend of the k-ϵ and k-ω models, through the following generic equation;

$$\alpha = F_1\alpha_1 + (1 - F_1)\alpha_2.$$

The constants are

$\alpha_1 = \frac{5}{9}$ $\beta_1 = \frac{3}{40}$ $\sigma_{k1} = 0.85$ $\sigma_{\omega 1} = 0.5$

$\alpha_2 = 0.44$ $\beta_2 = 0.0828$ $\sigma_{k2} = 1$ $\sigma_{\omega 2} = 0.856$

$\beta^* = 0.09$

The Standard k-ϵ Turbulence model

A form of the standard k-ϵ turbulence model was first proposed by Harlow and Nakayana[5] and has since appeared in many reports. The model is often referred to as the standard k-ϵ turbulence model, where k stands for the turbulent kinetic energy and ϵ is the dissipation rate of the turbulent kinetic energy. The computation of the turbulent viscosity is made recurring to a turbulent model. The turbulent viscosity is given by the following equation

$$\mu_t = C_\mu \frac{\rho k^2}{\epsilon}.$$

The turbulence kinetic energy, k as well as its dissipation rate, $\epsilon(m^2/s^3)$ are computed using the following transport equations.

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho u k)}{\partial x} + \frac{\partial(\rho w k)}{\partial z} = \frac{\partial}{\partial x}[(\mu + \frac{\mu_t}{\sigma_k})\frac{\partial k}{\partial x}] + \frac{\partial}{\partial z}[(\mu + \frac{\mu_t}{\sigma_k})\frac{\partial k}{\partial z}] + P_k - \rho\epsilon \tag{7}$$

$$\frac{\partial(\rho \epsilon)}{\partial t} + \frac{\partial(\rho u \epsilon)}{\partial x} + \frac{\partial(\rho w \epsilon)}{\partial z} = \frac{\partial}{\partial x}[(\mu + \frac{\mu_t}{\sigma_k})\frac{\partial \epsilon}{\partial x}] + \frac{\partial}{\partial z}[(\mu + \frac{\mu_t}{\sigma_k})\frac{\partial \epsilon}{\partial z}] + \frac{\epsilon}{k}(C_1 P_k - C_2\rho\epsilon). \tag{8}$$

The term P_k represents the production rate of k as the results of the velocity gradients;

$$P_k = \mu_t[2(\frac{\partial u}{\partial x})^2 + 2(\frac{\partial w}{\partial z})^2 + (\frac{\partial u}{\partial z} + \frac{\partial w}{\partial x})^2] \tag{9}$$

The remaining model constants are given by

$C_\mu = 0.09$ $\sigma_k = 1.0$ $\sigma_\epsilon = 1.3$ $C_1 = 1.44$ $C_2 = 1.92$

ISSN: 2521-2293

Since the flow is considered steady, the $\frac{\partial}{\partial t}$ terms in all the above equations are considered to be zeros.

The above set of governing equations have been solved, along with appropriate boundary conditions for the computational domain using numerical methods. The boundary conditions are;

$u(0, z) = u_{\infty}(0m/s)$

$w(0, z) = u(x, 0) = u(x, +\infty) = w(x, 0) = w(x, +\infty) = 0$

At $x = L$, $\frac{\partial u}{\partial x} = \frac{\partial w}{\partial x} = \frac{\partial k}{\partial x} = \frac{\partial \epsilon}{\partial x} = 0$ for all z

At $x = 0$, $k = (\frac{3}{2} I \mu_{\infty}^{2})$

I= Turbulence Intensity

$\epsilon = \left(\frac{C_{\mu}^{0.75} k^{1.5}}{L} \right)$ for all values of z

H=Inlet height(m)

The near wall treatment of momentum and turbulence equations as implemented in EasyCFD[7] obeys the suggestions explained in [8]. The basic motion behind the automatic wall functions is to change from a low-Reynolds number scheme to a wall function build on the mesh nodes nearness to the wall. The first order upwind method was used to discretize the momentum equations, turbulent kinetic energy and turbulence dissipation rate equations.

At the inlet of the computational domain, a uniform velocity condition is given. The exit was considered as a conservative outlet where the streamwise and normal gradients are zero for all parameters. On the walls the zero mean velocity condition was imposed. The computational fluid dynamics software EasyCFD[7] was chosen to solve these equations because of its capability, simplicity and user friendliness. This software uses a finite volume based discretization method. The computations are converged when the normalised residues for continuity and momentum equations are less than 0.0001.

3 Results and Analysis

The preliminary results shows that the drag of a bluff body is grid independent for both turbulence models. Figure 2 reflects that between the two models, as grid size increase the drag coefficient(C_D) for the SST turbulence model maintained less value as compared to the k-ϵ turbulence model. The graph also shows that both models maintained almost consistent value hence their graphs looking almost parallel as reflected in the graph.

ISSN: 2521-2293

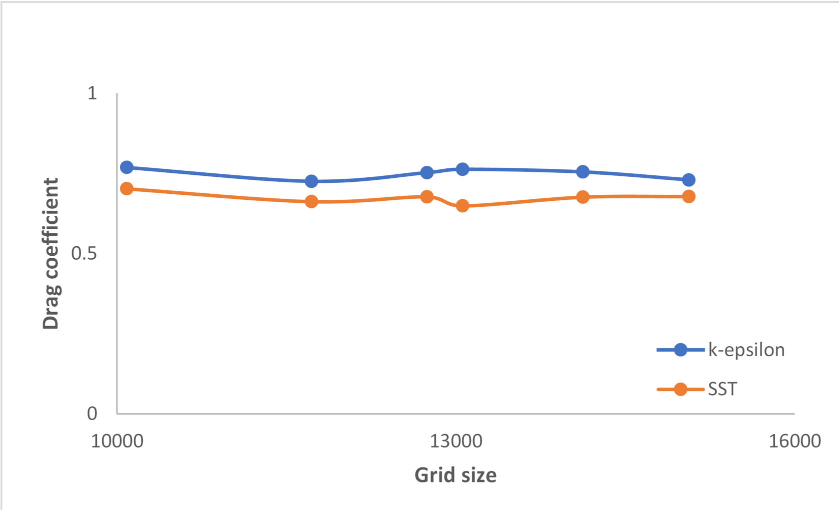

Figure 2: Effects of Mesh/Grid independence

The preliminary results also shows that as the inlet velocity increases(hence an increase in Reynolds number), the C_D maintained almost the same value as reflected in Figure 3. The SST turbulence model also maintained the least C_D value throughout as the Reynolds number increases.

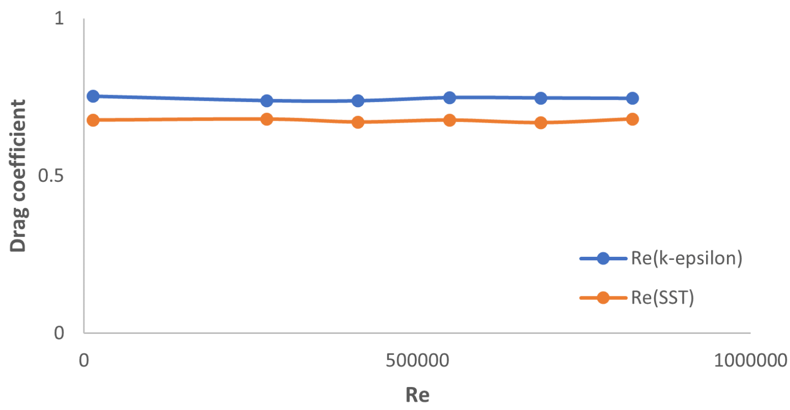

Figure 3: Effect of Reynolds number

4 Conclusion

The preliminary results suggest that between the two models for best results in predicting the drag of a bluff body, the SST turbulence model is the best model to use. The results also show that there was no any significant change in C_D for both models when increasing either the Reynolds number or grid size as reflected in figures 1 and 2.

References

[1] R.H Nicholas, *Turbulence models and their Applications to complex Flows*, University of Alabama at Birmingham,Revision 4.01.

[2] E. Rathakrishnam, *Effect of Splitter Plate on Bluff Body Drag*, AIAA Journal, Vol. 37, 1999.

[3] Choi,H. Jeon W and Kim J. *Control of Flow over a Bluff Body*,Annual Review of Fluid Mechanics, 40,2008, 113-139.

[4] F.R. Menter, M. Kuntz, R. Langtry, *Ten years of Industrial Experience with the SST Turbulence Model*, Turbulence Heat and Mass Transfer, 4(2003), 625-632. 10.4028, www.scientific.net/AMR.576.60.

ISSN: 2521-2293

[5] F.H. Harlow and P. Nakayama, *Transport of Turbulence Energy Decay rate*, Los Almos Science Lab, University of California Report LA-3854(1968).

[6] F.R. Menter, *Zonal two-equation k-ω Turbulence Model for Aerodynamic Flows*. AIAA Paper,1993-2906, 1993.

[7] A.G. Lopes, *EasyCFD: A Two-dimensional Computational Fluid Dynamics Software Manual.* Ver 4.4.4, www.easycfd.net, 2020.

[8] F.R. Menter, J.C. Ferreira and B. Konno, *The SST Turbulence Model with Improved Wall Treatment for Heat Transfer Predictions In Gas Turbines*, Proceedings of the International Gas Turbine Congress 2003, Tokyo, November 2-7,2003