ENCIT 2018

17th Brazilian Congress of Thermal Sciences and Engineering
November 25th-28th, 2018, Águas de Lindóia, SP, Brazil

ENCIT-2018-0582
SCALE-ADAPTATIVE SIMULATION TURBULENCE MODELING FOR A CYLINDER IN CROSS FLOW

Taiane Pereira Pantoja
Universidade Federal do Pará, Rua Augusto Corrêa, 01, Guamá, Belém, Pará, Brasil, CEP 66075-110
taianepereira23@gmail.com

Erb Ferreira Lins
Universidade Federal do Pará, Rua Augusto Corrêa, 01, Guamá, Belém, Pará, Brasil, CEP 66075-110
erb@ufpa.br

Abstract. *The study of turbulence models is commonly performed by Computational Fluid Dynamics. This work applies the SAS (Scaled-Adaptive Simulation) model to describe the flow around a cylinder with a Reynolds number (Re) of* 3900 *in different mesh sizes. This work also verifies that the theoretical value of the Smagorinsky constant, even with turbulence limiter in the production, can generate good results. The drag, lift and Strouhal coefficients are compared with experimental data. The model is able to obtain good results, with fewer computational requirements than the LES models and with a good representation of the flow vorticity, which makes it more practical for fluid structure analysis or other computer intensive flow simulations.*

Keywords: *Large Eddy Simulation, Scale Adaptative Simulation, Turbulence.*

1. INTRODUCTION

Due to inherent complexity of turbulent flows, in the analysis of flow around aerodynamic structures it is usually necessary to rely on turbulence models in numerical approximation of the velocity and pressure fields. In the framework of Computational Fluid Dynamics (CFD), these models are classified according the level of representation of small scales of flow and the computational requirements to obtain a useful solution. Although no turbulence model is really needed, as one can compute the velocity and pressure fields in any complex flow using Direct Numerical Simulation (DNS), the computational requirements to uses this type of simulation are high. The most used turbulence model in industrial turbulent flows simulations are based on Reynolds-Averaged Navier-Stokes equations (RANS). All RANS models uses Reynolds decomposition, i.e., any primitive variable of flow are decomposed in a averaged part and a fluctuating part. The averaged part of Navier Stokes equations is then solved adding some model to compute the influence of fluctuating part on flow equations. Similarly, models based on Large Eddy Simulation (LES), also uses a scale decomposition, and the equations are used to solve only the large scales of flow while small scales are modeled. Since LES models require less computational power (Jiménez, 2003) than DNS and offer better solution than RANS models, it is often used in industrial applications of great complexity. Even better, is possible to achieve 'LES'-like accuracy using Scale Adaptative Simulation (SAS) (Egorov *et al.*, 2010; Menter and Egorov, 2010). In these methods, information about the turbulent scales are inserted in RANS equations to increase the modelling capacity of RANS-model. The SAS methods have similar results as Detached Eddy Simulation (Egorov *et al.*, 2010) and were developed based on a transport equation for turbulence. Initially these model were developed by Rotta data and later being revised by Egorov and Menter in order to correct some existing limitations, resulting in the inclusion of a term in the von-karman length scale, which increases when the equation is more unstable (Davidson, 2006), so this model is able to adjust the length scale, so that the results of the turbulent flow are described with more similarity to the experimental results.

In this work, we solve the flow around a cylinder using Re of 3900, to show the effectiveness of the SAS (Scale-Adaptive Simulation) model in the computation of the lift, drag and Strouhal coefficients and to describe spatial structure of turbulent structures.

2. TURBULENCE MODEL

The equations used in present work are the Navier Stokes equations, which describe the flow of Newtonian Fluids. For turbulent fluids, due to mathematical complexity, it is necessary to rely on numerical solutions to approximate the results.

Based on the physical principles of mass and linear momentum conservation. These equation can be written as follow

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 \tag{1}$$

$$\frac{\partial u}{\partial t}+\frac{\partial u^2}{\partial x}+\frac{\partial (uv)}{\partial y}+\frac{\partial (uw)}{\partial z}=-\frac{1}{\rho}\frac{\partial p}{\partial x}+\nu\left(\frac{\partial u^2}{\partial x^2}+\frac{\partial u^2}{\partial y^2}+\frac{\partial u^2}{\partial z^2}\right) \tag{2}$$

$$\frac{\partial v}{\partial t}+\frac{\partial (uv)}{\partial x}+\frac{\partial v^2}{\partial y}+\frac{\partial (vw)}{\partial z}=-\frac{1}{\rho}\frac{\partial p}{\partial y}+\nu\left(\frac{\partial v^2}{\partial x^2}+\frac{\partial v^2}{\partial y^2}+\frac{\partial v^2}{\partial z^2}\right) \tag{3}$$

$$\frac{\partial w}{\partial t}+\frac{\partial (uw)}{\partial x}+\frac{\partial (vw)}{\partial y}+\frac{\partial w^2}{\partial z}=-\frac{1}{\rho}\frac{\partial p}{\partial z}+\nu\left(\frac{\partial w^2}{\partial x^2}+\frac{\partial w^2}{\partial y^2}+\frac{\partial w^2}{\partial z^2}\right) \tag{4}$$

Where u, v and w are the fluid velocity components in x, y and z directions, respectively. ρ is fluid density, ν, the kinematic viscosity and p is pressure. When using LES turbulence model, one usually applies the spatial average operator

$$\phi(\mathbf{x})=\int_{\Omega}\phi(\mathbf{x}')G(\mathbf{x},\mathbf{x}';\Delta)d\mathbf{x}' \tag{5}$$

Where Ω is the integration domain, G and Δ is the filter function and width. LES models differs in the filter functions and width, so structures and correlations of the flow are captured in different manners. In Rotta (1968), a equation for a transport variable kL is done using

$$kL=\frac{3}{16}\int_{-\infty}^{\infty}R_{ii}(\mathbf{x},r_y)dr_y \tag{6}$$

$R_{ii}(\mathbf{x},r_y)$ is the sum of diagonal of correlation tensor $R(\mathbf{x},r)$. One can establish a two equation model using this variable as a scalar transported function and using it to obtain a length scale L. In practical terms it is often better use $\Phi=\sqrt{k}L$ as second variable. It should be observed that the correlation function acts as a spatial filter. Using a transport equation for the turbulent kinetic energy k and Φ as

$$\rho\frac{\partial k}{\partial t}+\rho\frac{\partial (U_jk)}{\partial x_j}=P_k-c_\mu^{-3/4}\rho\frac{k^2}{\Phi}+\frac{\partial}{\partial x_j}\left(\frac{\mu_t}{\sigma_k}\frac{\partial k}{\partial x_j}\right) \tag{7}$$

$$\rho\frac{\partial \Phi}{\partial t}+\rho\frac{\partial (U_j\Phi)}{\partial x_j}=\frac{\Phi}{k}P_k\left(\zeta_1-\zeta_2\left(\frac{L}{L_{\nu K}}\right)^2\right)-\zeta_3\rho k+\frac{\partial}{\partial x_j}\left(\frac{\mu_t}{\sigma_\Phi}\frac{\partial \Phi}{\partial x_j}\right) \tag{8}$$

Where $U=(u,v,w)$ and the eddy turbulent viscosity is given by

$$\mu_t=c_\mu^{-1/4}\rho\Phi \tag{9}$$

in special we use of lower limiter of the von Karman lenght scale as

$$L_{\nu K}=max\left(\kappa\left|U'/U''\right|,C_s\sqrt{\frac{\kappa\xi}{\beta/c_\mu-\alpha}}\Delta\right). \tag{10}$$

And the other terms are as follows:

$$U'=\sqrt{2S_{ij}S_{ij}} \tag{11}$$

$$S_{ij}=\frac{1}{2}\left(\frac{\partial U_j}{\partial x_i}+\frac{\partial U_i}{\partial x_j}\right) \tag{12}$$

$$U''=\sqrt{\frac{\partial^2 U_i}{\partial x_k^2}\frac{\partial^2 U_i}{\partial x_j^2}}\kappa \tag{13}$$

κ is the Von-Karman constant. Compared with other 2 equations models, the additional term $\frac{L}{L_{\nu K}}$ in Eq.(8) represents a production term with direct relation with the turbulent length scale.

2.1 Geometry and Mesh

The geometry used in this work represents a cylinder in crossflow (Norberg, 2003). This model was chosen due to simplicity of the geometric model, the high complexity of flow and the large amount of experimental data available in the literature for comparison.

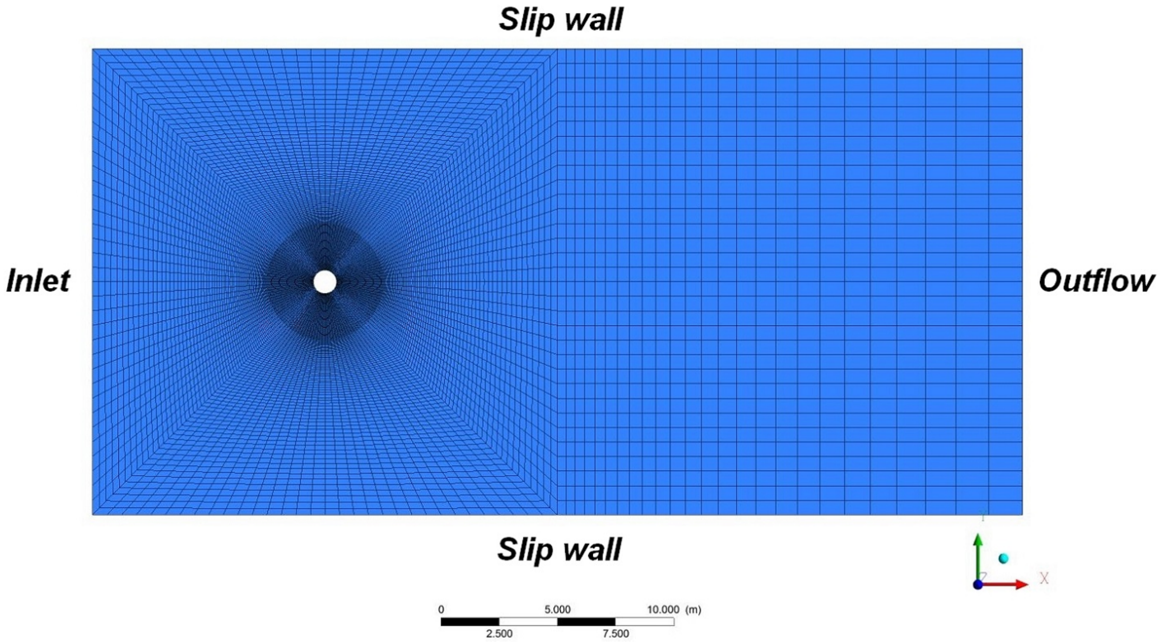

Figure 1. Computational mesh.

The domain consist of a prismatic box of dimensions $40D \times 20D \times 4D$, where D is the cylinder diameter. In figure (1), it is shown the finite volumes mesh along a cross section. The flow is aligned with positive x-direction and has magnitude $U = (U_\infty, 0, 0)$ at inlet boundary. Periodic boundary condition are set in transverse direction. The cylinder wall is set as non-slip boundary condition. Considering the cylinder diameter as caracteristic lenght the Reynolds number is given by Re of$\frac{U_\infty D}{\nu}$ of 3900 with ν as kinematic viscosity of fluid.

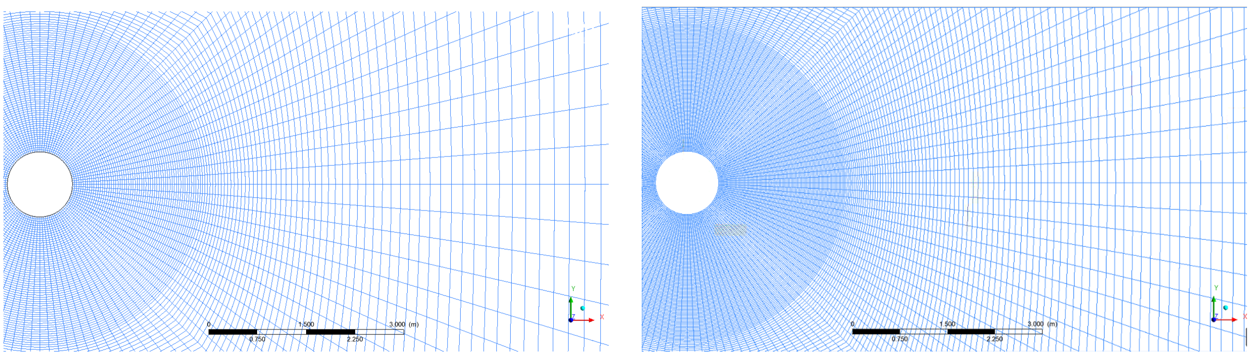

Figure 2. Mesh Details.

Table 1. Description of the mesh refinement from the number of nodes and cells.

Mesh	Nodes	cells
Mesh 1	433,425	412,416
Mesh 2	614,234	589,440
Mesh 3	856,812	824,832

The coarser mesh has 433,425 nodes and 412,416 cells and the finer mesh has 856,812 nodes and 824,832 cells. Figure (2) shows the mesh details around the cylinder, for these two meshes. In present work, the analysis of three meshes, which are described in the table (1). ANSYS Fluent 16.0 software was used in simulations (ANSYS, 2017), with scale-adaptive simulation model and production limiter. All equations are solved with a time step Δt of 0.01 seconds and, at each time step, the nonlinear iterations are performed until all equations residuals are below 10^{-4}. Finally it should be noted that developments in theory of LES models as seen in Meyers and Sagaut (2006) should be higher than the usual parameter set as $C_s \approx 0.11$, so in current work, we proposed $C_s = 0.17$. This value are also used was used in McMillan and Ferziger (1979) and Lilly (1967).

From the numerical specifications of the simulation, the results were obtained with a total time of $250s$ and which results in $25,000$ iterations.

3. RESULTS AND DISCUSSIONS

The results obtained for drag and lift coefficients are shown in fig.(3). As can be seen, this flow has a vortex shedding process, which is usual in this flow regime. Despite this, the flow is not periodic, as RANS simulations usually predicts in this case. The SAS model, instead, can predict accurately, with correct dynamic correlation length, the turbulent structures of flow. In table (2), the aerodynamic parameters obtained in SAS model and experimental data are compared, when analyzing the results we obtain a greater convergence from the mesh refinement, especially when analyzing the Strouhal coefficient, it is observed that the higher the refinement, the better the description of the oscillating flow. The model formulation can correctly generate a turbulent spectrum (as a LES model), in the regions were strong instability are present, still acting like a RANS model in stable regions of flow. In figure (3). it is show the isosurfaces of $Q-$criterion, colored by velocity magnitude for the mesh 3. As could be noted, the model can predict turbulent structures in the main flow with richer detail.

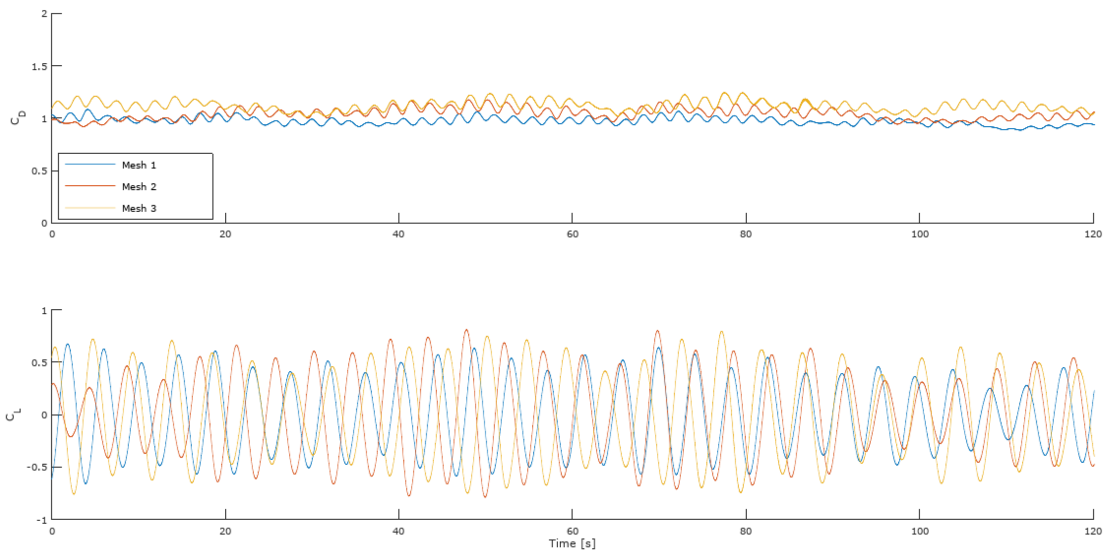

Figure 3. Time history of drag coefficient C_d and lift coefficient C_l.

Table 2. Mean values of flow aerodynamic parameters.

Parameter	mesh 1	mesh 2	mesh 3	Norberg (2003, 1993)
C_d	0.97	1.06	1.12	1.02
C_l	3×10^{-3}	2×10^{-3}	-6×10^{-4}	0
C_{l_rms}	0.35	0.39	0.42	0.30
St	0.24	0.22	0.219	0.21

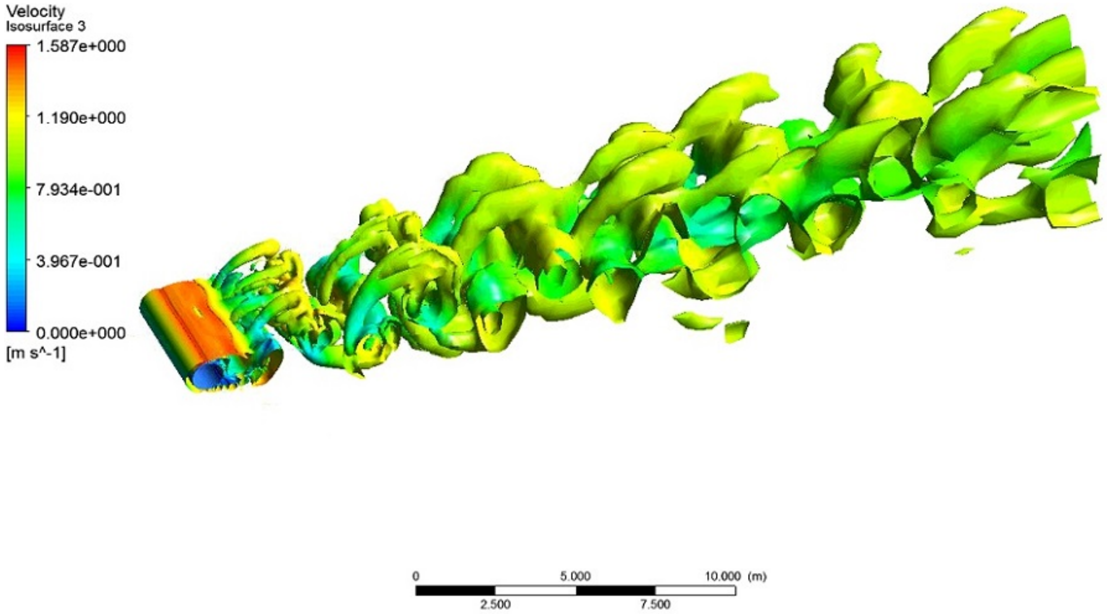

Figure 4. Q-criterion isosurfaces colored by velocity magnitude.

4. CONCLUSIONS

In this work, the flow over a fixed cylinder for Re of 3900 in a turbulent sub-critical regime was done using the SAS model. The model was proven adequate to describe with good accuracy the turbulent region, presenting details of turbulent structures at the flow. Both the calculation of the average drag and lift parameters from the mean of C_d and C_l shows the model could produce relevant results for modelling more complex aerodynamic systems and rough bodies. it is important to emphasize that from the results obtained for the Strouhal coefficient, this model can be applied in fluid-structure analyses, where the vortex shedding frequency has a determinant role. The model presented here also uses a higher value for C_S parameter, differently from many analyses current seen in literature. This value has a direct impact on turbulent viscosity calculation and points out that structure of SAS model are, as predicted by (Menter and Egorov, 2010), very similar to the structure of Smagorinsky Model in region with strong instabilities.

5. REFERENCES

ANSYS, 2017. *ANSYS Fluent Theory Guide, Release 16, ANSYS, Inc.*

Davidson, L., 2006. “Evaluation of the sst-sas model: Channel flow, asymmetric diffuser and axi-symmetric hill”. In *ECCOMAS CFD 2006: Proceedings of the European Conference on Computational Fluid Dynamics*.

Egorov, Y., Menter, F.R. and Lechner, R., 2010. “The scale-adaptive simulation method for unsteady turbulent flow predictions. part 2: Application to complex flows”. *Flow, Turbulence and Combustion*, Vol. 85, p. 01390165.

Jiménez, J., 2003. “Computing high-reynolds-number turbulence: will simulations ever replace experiments?” *Journal of Turbulence*, Vol. 4, p. N22. doi:10.1088/1468-5248/4/1/022. URL `https://www.tandfonline.com/doi/abs/10.1088/1468-5248/4/1/022`.

Menter, F.R. and Egorov, Y., 2010. “The scale-adaptive simulation method for unsteady turbulent flow predictions. part 1: Theory and model description”. *Flow, Turbulence and Combustion*, Vol. 85, p. 01130138.

Meyers, J. and Sagaut, P., 2006. “On the model coefficients for the standard and the variational multi-scale smagorinsky model”. *J. Fluid Mech*, Vol. 569, pp. 287–319.

Norberg, C., 1993. “Flow around rectangular cylinders: Pressure forces and wake frequencies”. *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 49, No. 1, pp. 187 – 196. ISSN 0167-6105.

Norberg, C., 2003. “Flow around a circular cylinder: Aspects of fluctuating lift”. *Journal of Fluids and Structures*, Vol. 15, No. 3, pp. 459 – 469. ISSN 0889-9746.

Rotta, J.C., 1968. *Uber eine Methode zur Berechnung turbulenter Scherstromungen*. Ph.D. thesis, Aerodynamische Versuchsanstalt Gottingen, Gottingen, Germany.

6. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.