MATEC Web of Conferences **72**, 01100 (2016)
HMTTSC-2016
DOI: 10.1051/matecconf/20167201100

NUMERICAL SIMULATION OF TURBULENT FLOW OVER OF SEMI-CYLINDRICAL PROFILE AT ZERO ANGLE OF ATTACK

Alexander Shebelev[1,*], and *Andrey* Gavrilov[1,2]

[1] Siberian Federal University, Krasnoyarsk, Russia
[2] Institute of Thermophysics, Novosibirsk, Russia

Abstract. Results of unsteady two-dimensional numerical simulations for the turbulent flow over the semi-cylindrical profile at Reynolds number 45000 are presented. Three different turbulence RANS eddy viscosity models are used for the simulation. The calculations show the formation of alternating large-scale vortices shed in the wake of the body. The calculated averaged pressure distribution on the contour demonstrate a good agreement with the experimental data. A series of calculations have been performed for the estimation of influence of mesh resolution in the wake of streamlined body on the unsteady solution accuracy. It was found that a mesh size ~0.04 of chord length is sufficient to resolve the vortex street.

1 Introduction

A practical interest to the simulation of detached high Reynolds number flows is associated with numerical resolution of unsteady vortex structures, which are strongly influence on the integral characteristics of such flows. This class of flows is difficult to predict by methods of computational fluid dynamics because of unfixed separation point and development of non-stationary flow pattern.

2 Numerical algorithm

Flow around semi-cylindrical profile at zero angle of attack was investigated numerically. For this flow, there are the experimental data for drag and lift forces and for pressure distribution along the contour of body [1]. The Reynolds number defined by the length of chord L, velocity of undisturbed flow U and the fluid viscosity and density is $\mathrm{Re} = \rho UL / \mu = 45000$. Calculations have been performed in two-dimensional formulation. The height of computational domain is 40L, the length in the streamline direction is 20L.

* Corresponding author: aleksandr-shebelev@mail.ru

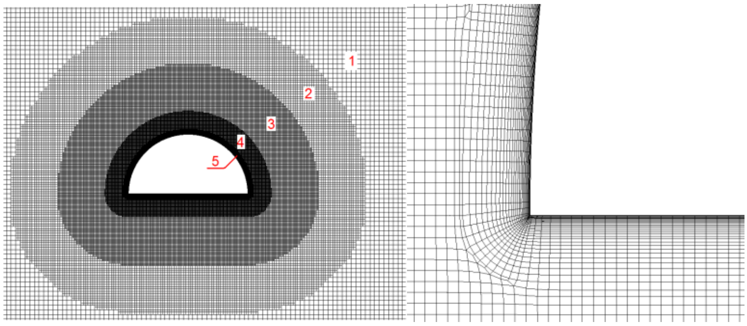

Fig. 1. Visual representation of the mesh.

The calculations were carried out on a hybrid hexahedral mesh composed by several uniform meshes and near wall body-fitted mesh (fig.1). To assess the influence of mesh resolution in the wake of body on the lift coefficient (*Cy*) and drag coefficient (*Cx*) four mesh with different mesh size in wake region were generated. Meshes parameters are specified in Table 1.

Table 1. Parameters computational meshes.

Mesh	mesh size in wake	number of cells, thousand
M1	0.02	250
M2	0.04	150
M3	0.08	126
M4	0.16	121

Non-reflecting outflow boundary conditions were set at the outlet. Symmetry conditions were set on the bottom and top faces of the computational domain. Non-slip boundary conditions were imposed at the wall. Mesh size near wall is enough for full resolution of near wall turbulent layer including viscous sublayer. The turbulence at the inlet boundary was defined by the length scale of turbulence $L_t=0.1\ L$ and turbulence intensity u/U=0.015.

Following Reynolds-averaged Navier-Stokes (RANS) models were used: the *k-ω SST* model with curvature correction [2], model with elliptic blending *k-ε-ζ-a* and model with elliptic relaxation *k-ε-ζ-f* [3].

The time step for unsteady simulation was *0.01 L/U*. The statistics was collected by averaging more than 100 of transit time. The calculations were performed with a finite volume in-house CFD code, σFlow [4].

3 Results and conclusions

The integral results of simulations performed with different turbulence models on mesh M1 and available experimental data are presented in the Table 2.

Table 2. Integral results.

Turbulent model	Period	*Cx*	*Cy*	*rms(Cx)*	*rms (Cy)*
k-ω SST cc	2.2	0.488	-0.89	0.046	0.314
k-ε-ζ-a	2.2	0.526	-1.014	0.064	0.252
k-ε-ζ-f	2.3	0.54	-0.944	0.064	0.208
Experiment	–	0.3	-1.1	–	–

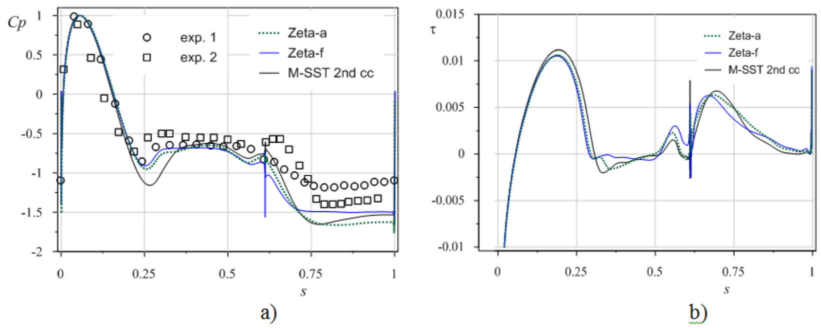

Fig. 2. Distributions of average pressure (a) and wall shear stress (b) on surface. Symbols correspond to the experimental data [1].

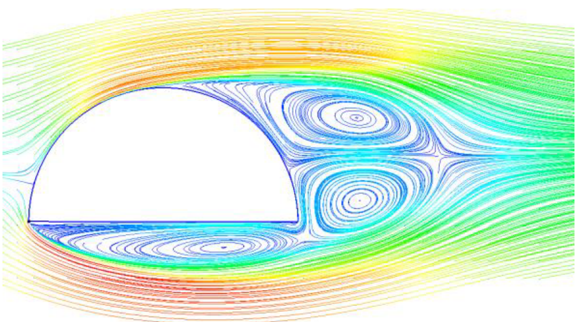

Fig. 3. Streamlines of average flow for model *k-ε-ζ-f*.

Models with elliptic relaxation better predict the position of separation point and correctly reproduce average value of pressure near the separation point (fig. 2a). On the bottom of semi-cylinder all turbulence models underestimate the value of pressure. The low level of pressure can be explained by the high intensity of vortex flow generating under the body (fig. 3). All turbulence models overestimate the drag coefficient *Cx*. Models with elliptic relaxation slightly better predict the lift coefficient *Cy* with divergence from experimental data about 10% (Table 2).

In general, the unsteady RANS modeling of detached flow shows good agreement with experimental data. The unsteady formulation of RANS models makes it possible to resolve the development of vortex street behind the body.

Table 3. Influence of mesh size in the wake on the integral force (*k-ε-ζ-f* model).

Mesh	Cx	Cy	rms(Cx)	rms(Cy)
M1	0.5912	-0.96	0.0532	0.318
M2	0.5914	-0.96	0.0532	0.318
M3	0.5914	-0.96	0.0534	0.318
M4	0.5916	-0.958	0.0532	0.318
Experiment	0.3	-1.1	–	–

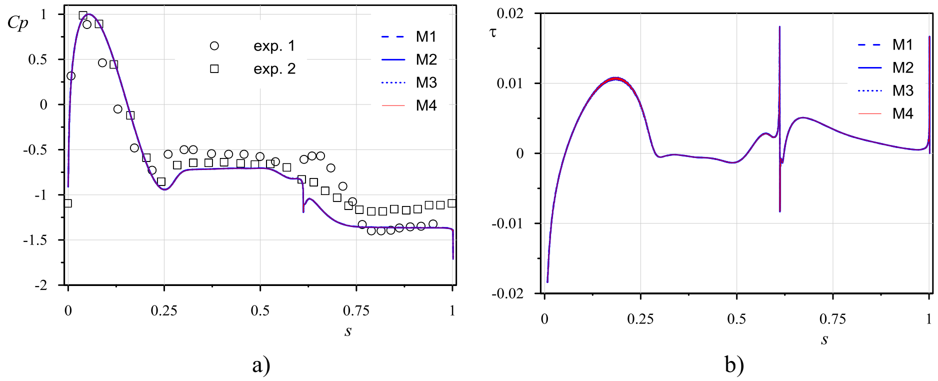

Fig. 4. Influence of mesh resolution in the wake behind the body for the turbulence model *k-ε-ζ-f*. Averaged pressure distribution (a) and wall shear stress (b) on the body surface.

Figure 4 and Table 3 demonstrate that the mesh resolution in the of wake behind the streamlined body at a distance more than 2 *L* from the body does not affect on the integral forces acting on the body. In addition, it was found that a mesh size ~0.04 of chord length is sufficient to resolve the vortex street.

References

1. Z.P. Sluchanovskaya, Sci. works Instit. Mech. **24 (**1973)
2. S.A. Isaev, P.A. Baranov, Yu.V Zhukova., A.E Usachov., V.B. Kharchenko, J. Eng. Phys. Thermophys. **87** (2014)
3. K. Hanjalić, M. Popovac, M. Hadžiabdić, Int. J. Heat Fluid Flow **25** (2004)
4. A.A. Gavrilov, A.V. Minakov, A.A. Dekterev, V.Ya. Rudyak , Siber. J. Industr. Math. **13** (2010)