Model build up for numerical study of the velocity field characteristics of jet flow in the work zone of small aerodynamic tunnel, by using LES method

Martin Ivanov, Sergey Mijorski

Изграждане на модел за числено изследване на характеристиките на скоростно поле на струйно течение в работния участък на малък аеродинамичен тунел, чрез CFD метода LES: *Публикацията представя числено изследване на разпределението на скоростното поле на струйно течение в работната зона на аеродинамичния тунел (АДТ) на Технически университет София. Тук е разгледано подробно изграждането на симулационния модел. Численото изследване се базира на методите на изчислителната динамика на флуидите (CFD), като използвания метод за решаване на системата частни диференциални уравнения на Навие - Стокс е „Метода на крупните вихри“ - LES (Large Eddie Simulation). При този подход, едромащабните турбулентни вихри на течението се изчисляват директно, а малките предварително филтрирани вихри се моделират чрез широко използвания модел на Смагорински. LES метода е добре приложим при моделиране на струйни течения, като предоставя значителна точност и детайлност на резултатите при сравнително ниска стойност на изчислителните итерации. В представената разработка е изследвано числено разпределението на скоростното поле на струйното течение в работния участък на АДТ на ТУ-София, като за гранични условия на проведената симулация са използвани реални данни от физически измервания. Получените резултати охарактеризират адекватно параметрите на струйното течение в работната област и дават възможност за разработване на виртуален модел на изследваното съоръжение.*

Key words: *Aerodynamic Tunnels (ADT, Wind Tunnels), Velocity Field Modeling, CFD, LES method*

INTRODUCTION

The presented work could be considered as case study, which is part of the logical continuation of the research, published in: *“Ivanov, M., Markov, D., “Experimental study of the velocity field characteristics of jet flow in the work zone of small aerodynamic tunnel”, Proceedings of “International Scientific Conference in Ruse – 2013*”, Vol.52, pp. 69-73, 2013*”*. As it is mentioned, there exists a general need for documentation, concerning the preparation of usage, support and the maintenance of the small air dynamic tunnel (ADT) at Technical University – Sofia (TU-Sofia), shown in Figure 1 [1, 2]. In the previously presented studies, the overall condition of the device was assessed, and accurate 3D model was built [1]. In addition, an experimental study of the velocity field in the work zone of the ADT is performed for various operating modes. Measurement and analyses of the air flow velocity is made in 176 points from the ADT work zone. The results show relatively high degree of homogeneity of the velocity field for the different regimes. In addition, velocity drop in all investigated cases is observed and assessed [1, 2].

The next step in the work, concerning the ADT of TU-Sofia database, is to perform numerical study of the velocity field distribution in the work zone of the device. The numerical study was based on the Computational Fluid Dynamics (CFD) methods, in particular – the Large Eddie Simulation method [3, 4, 5]. As boundary conditions were used the experimental results, achieved in the previously presented papers [1, 2]. The analyses of the numerical results will give significant input to the knowledge of work and maintenance, especially into the stage of design and performance of experimental studies at TU-Sofia’s ADT.

Also, the suggested LES method has the potential to represent the flow parameters distribution in the ADT work zone with significantly high accuracy. Such kind of flow visualization and assessment is missing up till now. From that point of view, this case study is innovative for the general application of ADT at TU-Sofia.

Figure 1. General View of the TU – Sofia's ADT

AIM OF THE PRESENTED NUMERICAL STUDY

The aim of the presented study is to investigate numerically the flow distribution parameters in the work zone of the small aerodynamic tunnel at Technical University – Sofia.

The numerical methods are based on CFD, in particular the LES methods. For boundary conditions are used the results from previously performed experimental measurements.

METHODS OF THE PRESENTED NUMERICAL STUDY

As it was mentioned, the main approach used in the presented numerical study was the Large Eddy Simulation method. LES is a mathematical model for turbulence, used in computational fluid dynamics, which was initially proposed by Smagorinsky [5]. The operating principal of LES is the so called low-pass filtering. This operation is applied to the Navier–Stokes equations in order to eliminate the small scale eddies from the solution. The governing equations are thus transformed, and the obtained solution is in the form of filtered velocity and pressure fields.

LES resolves the large scale eddies of the flow field directly, which allows better reliability of the solution, than alternative approaches such as Reynolds-averaged Navier–Stokes (RANS) methods. The other feature is that LES also models the smallest scales of the solution, based on widely used Smagorinsky model, rather than resolving them as direct numerical simulation (DNS) does. This avoidance of the small scale eddies calculation reduces significantly the computational cost of the simulation. That is why, this method is quite applicable for flow systems with complex geometry or complex flow configurations.

NUMERICAL MODEL

The numerical model was developed on the CFD package "*Open FOAM Version: 2.2*" and the simulation was performed on the "*pimpleFoam*" solver. The model geometry represents in very high accuracy the real ADT work zone, and it is presented on Figure 2.

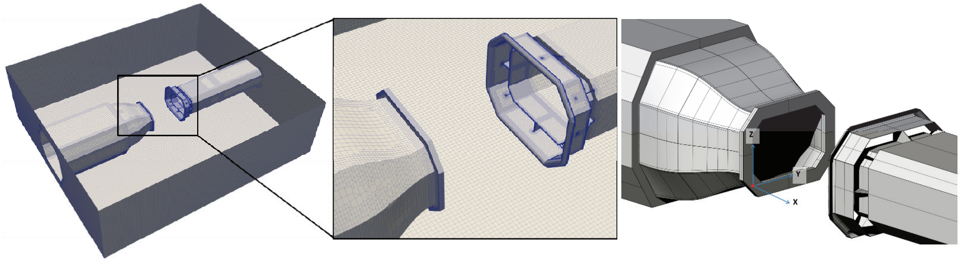

Figure 2. Numerical model and coordinate system, used in the study

The figure also shows the main coordinate system used in the study. The overall dimensions of the numerical domain were approximately 7m x 6m x 2m, including the entire work area, which was the main assessment object of the presented study. The dimensions were selected in order to reduce to minimum the effects of the domain boundaries over the numerical solution in the work area of the ADT. The ADT's work area is approximately 1m long and 0.85 m wide. The cross section of the air jet is roughly rectangular with a height of 0.5 m and width 0.75 m.

COMPUTATIONAL GRID

The overall number of cells in the computational grid was 7 136 910, which is relatively high, because the LES method requires significant refinement of the computational grid, in order to achieve numerically correct results. That is why the numerical grid was constructed with different polygonal shapes of the elements and with different number of cells per refinement level. These properties are described in Table 1 and 2.

Table 1. Types of cells in the computational grid

Type of cells:	Total number of cells:	Percentage from the grid:
Hexahedral	*6 297 969*	*88.2%*
Prisms	*161 061*	*2.26%*
Wedges	*8 081*	*0.113%*
Pyramids	*523*	*0.00733%*
Tetra wedges	*3 350*	*0.0469%*
Tetrahedral	*7 976*	*0.112%*
Polyhedral	*657 950*	*9.22%*

Table 2. Refinement level and cell spacing in the computational grid

Refinement level:	Total number of cells:	Cell spacing:
0	*1 181 363*	*40 mm*
1	*427 833*	*20 mm*
2	*915 702*	*10 mm*
3	*669 247*	*5 mm*
4	*444 812*	*2.5 mm*
5	*878 660*	*1.25 mm*
6	*2 619 293*	*0.625 mm*

General view of the computational grid is presented on Figure 3. Special volume and surface refinements were added around the areas of the nozzle and the collector. At these zones are expected high flow shear stresses and increased turbulence intensity.

Table 3. Prism layer surfaces refinements of the ADT walls

Surface areas:	Number of prism layers:	First cell height:	Scaling ration:
Nozzle - Front wall surfaces	*3*	*1.98 mm*	*1.25*
Nozzle - Flow align wall surfaces	*3*	*7.02 mm*	*1.25*
Collector - Front wall surfaces	*3*	*0.521 mm*	*1.25*
Collector - Flow align wall surfaces	*3*	*0.641 mm*	*1.25*
Collector - Back wall surfaces	*3*	*6.25 mm*	*1.25*

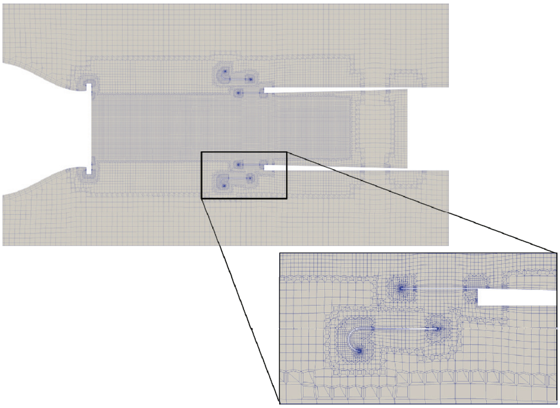

Figure 3. Computational grid for the presented study

In order to capture the effect of the detailed nozzle and collector geometry shapes, three prism layers were added to all ADT surfaces. Detailed information for the prism layer addition is presented in Table 3. Also, the grid inside the work zone, where the fully developed core region of the jet flow is expected, was constructed with rectilinear grid, based on 10 mm wide grid elements. This was done in order to assure flow distribution results with sufficiently high accuracy.

INITIAL AND BOUNDARY CONDITIONS

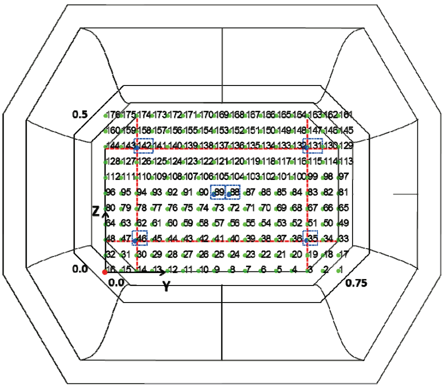

Figure 4. Positions of the velocity measurement locations at the ADT nozzle outlet

The initial conditions were initialized with calculated potential flow, according to the preliminary obtained flow measurement results at nozzle outlet [1, 2]. Figure 4 shows the position of the 176 points, at which measurement results for the velocity filed are obtained. In the numerical simulations, these points were used as non-uniform velocity inlet at the nozzle outlet.

The simulations were performed under isothermal conditions. In fact, the governing energy equation and the temperature equation was not calculated. The numerical scheme, for connecting the velocity and pressure fields, was PIMPLE. This scheme is combination between PISO and SIMPLE algorithm. In the SIMPLE algorithm, a pressure correction term is used while the velocity corrections are neglected, because they are unknown. This results in rather slow convergence. The PISO algorithm also neglects the velocity correction in the first step, but then performs one in a later stage, which leads to additional corrections for the pressure.

CONCLUSION

● Numerical model of the work zone of the small air dynamic tunnel at Technical University – Sofia was proposed and constructed. The model was built within the "*Open FOAM Version: 2.2*" software package.

● Complex computational grid was also constructed, in order to assure flow results with significant accuracy. The numerical model's initial and boundary conditions were set according to previously performed experimental measurements of the ADT flow parameters.

● Results from the performed numerical study are presented and submitted for publishing in: *"Mijorski, S., Ivanov, M., "Results from numerical study of the velocity field characteristics of jet flow in the work zone of small aerodynamic tunnel, by using LES method", Submitted for publishing in: Proceedings of "International Scientific Conference in Ruse – 2014", October, 2014"*

REFERENCES

[1] Ivanov M., Markov D., Zanev D., „Experimental study of the velocity field of jet flow in the work zone of aerodynamic tunnel in department "Hydroaerodynamics and hydraulic machines", at Technical university Sofia", Proceedings of: „XVIII Scientific conference - FPEPM 2013, 15-18.09.2013", Vol II, pp. 32-39, 2013;

[2] Ivanov, M., Markov, D., "Experimental study of the velocity field characteristics of jet flow in the work zone of small aerodynamic tunnel", Proceedings of "International Scientific Conference in Ruse – 2013", Vol.52, pp. 69-73, 2013;

[3] Meneveau C., "Turbulence: Subgrid-Scale Modeling". Scholarpedia 5 (1): 9489, 2010;

[4] Meneveau C., Katz J., "Scale-Invariance and Turbulence Models for Large-Eddy Simulation". Annu. Rev. Fluid Mech. 32 (1): pp.1–32, 2000;

[5] Smagorinsky J., "General Circulation Experiments with the Primitive Equations", Monthly Weather Review 91 (3): pp.99–164, 1963.

For contacts:

Senior Assist. Prof. Martin Ivanov, PhD, Technical University - Sofia, FPEPM, department: 'Hydroaerodynamics and Hydraulic machines', 029653305, martin_ronita@yahoo.com

Dr. Sergey Mijorski, PhD, BMT Fluid Mechanics Ltd., Teddington, London, sergey_mijorski@yahoo.com

This paper has been reviewed.