OPEN
10.1515/tvsb-2016-0018

Transactions of the VŠB – Technical University of Ostrava

Civil Engineering Series, Vol. 16, No. 2, 2016

paper #18

Juraj KRÁLIK[1]

CFD SIMULATION OF AIR FLOW OVER AN OBJECT WITH GABLE ROOF, REVISED WITH Y+ APPROACH

Abstract

Aim of this contribution is to provide insight view into analysis focused on obtaining external pressure coefficients on isolated two storey low-rise building with 15° elevation gable roof using Computer Fluid Dynamics simulation and these are compared to values that offering Eurocodes. Final Volume Model consisting of polyhedral mesh will be used for analysis with two different turbulence models. Mesh was created with respect to y+ parameter, where desired value was below one which leads us to fine mesh type. Secondary aim of this contribution is to compare performance of selected turbulence models. For this purpose were chosen Detached Eddy Simulation and Large Eddy Simulation which are part of the Scale Resolving Simulation turbulence models.

Keywords

Ansys, fluent, airflow, turbulence, pressure coefficients, polyhedral mesh.

1 INTRODUCTION

As Computer Fluid Dynamic (CFD) software develops, problems of fluid dynamics becoming interesting for more engineers. CFD is a handy tool capable of reasonable predicting of air-flows. In this article will be used to predict external pressure coefficients on simple rectangular low-rise building with gable roof.

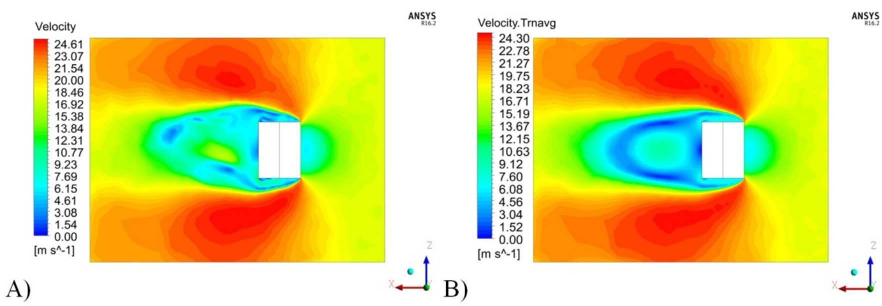


Fig. 1: Velocity in horizontal plane at +0.03 m elevation, A) RANS model, B) URANS/SRS model

There are three turbulent flow simulation methods Reynolds Averaged Navier-Stokes Simulations (RANS), Scale Resolving Simulations (SRS) and Direct Numerical Simulation (DNS) and several commercial and non-commercials software packages offering CFD simulations. For the purpose of this analysis was used commercial software package ANSYS Fluent R16.2. The main difference between RANS and SRS can be seen on velocity profile in horizontal plane in figure 1. One is providing steady state results and the other one time averaged state. From Detached Eddy

[1] Ing. Juraj Králik, Ph.D., Institute of Constraction in Architecture and Engineering Structures, Faculty of Architecture, Slovak University of Technology in Bratislava, Námestie Slobody 19, 812 45 Bratislava, Slovak Republic, phone: (+421) 903 951 403, e-mail: kralik@fa.stuba.sk.

Simulation turbulence model was selected Delayed Detached Eddy Simulation (DDES) and from Large Eddy Simulation it was Wall Modelled Large Eddy Simulation (WMLES).

2 Y^+ APPROACH

The y+ value is a non-dimensional distance (based on local cell fluid velocity) from the wall to the first mesh node, and is determining whether the influences in the wall adjacent cells are laminar or turbulent. In CFD often used to describe if mesh is fine or coarse. There are three subdivisions of the near wall region in turbulent boundary layer (see figure 2): viscous sub layer region with y+ < 5 (velocity profiles assumed to be laminar and dominate the wall shear); buffer region with 5 < y+ < 30 (dominates both viscous and turbulent shear); fully turbulent portion or log-law region with 30 < y+ < 300 (turbulent shear dominates). Values of y+ close to the lower bound y+ ≈ 30 are most desirable for wall function and y+ ≈ 1 for near wall modeling, [1]. So we can say that y+ is a suitable selection criterion for determining the appropriate mesh configuration and turbulence model.

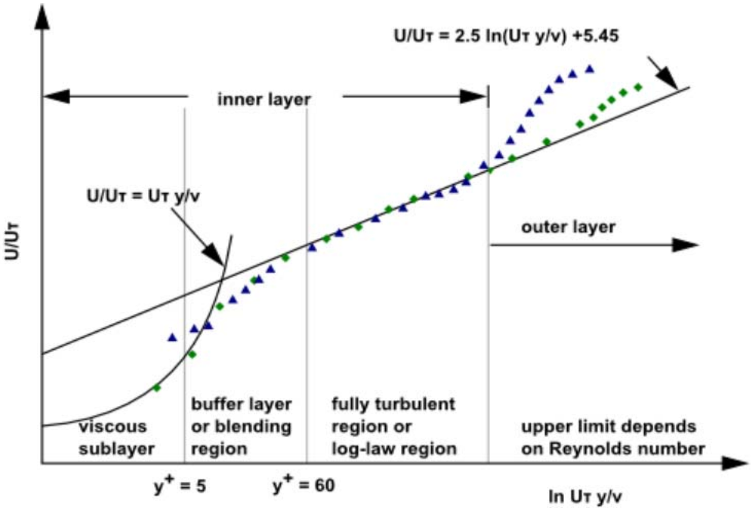


Fig. 2: Subdivisions of near-wall region, [1]

Low-rise building with ground floor dimensions 6 x 8 m with gable roof with 15° elevation was modeled in scale 1:100. So object model dimensions were scaled down to 0.06 x 0.08 m. To examine a fully turbulent environment Reynolds number needs to be greater than 10^5. For object with the characteristic length value of $L_{obj} = 0.06$ m and air density of $\rho_{air} = 1.225$ kg.m^{-3} and with turbulent viscosity of air with value $\mu_{air} = 1.7894 \cdot 10^{-5}$ kg.m^{-1} for the minimum value of Reynolds number $\mathrm{Re}_{min} = 10^5$ will reference air speed be:

$$v_{Re} = \frac{\mu_{air} \cdot \mathrm{Re}_{min}}{\rho_{air} \cdot L_{obj}} = \frac{1.7894 \cdot 10^{-5} \cdot 10^5}{1.225 \cdot 0.06} = 24.346\, m \cdot s^{-1} \quad (1)$$

Chosen reference air speed is $v_{air} = 25$ m.s^{-1} at reference height at the top of the roof (0.068 m). Respectively the Reynolds number will be:

$$\mathrm{Re}_{obj} = \frac{\rho_{air} \cdot v_{air} \cdot L_{obj}}{\mu_{air}} = \frac{1.225 \cdot 25 \cdot 0.06}{1.7894 \cdot 10^{-5}} = 1.027 \cdot 10^5 \quad (2)$$

For calculation of FLT is necessary to calculate skin friction coefficient, which is defined on plate as follows:

$$C_f = 0.058 \cdot \mathrm{Re}_{obj}^{-0.2} = 0.058 \cdot 1.027 \cdot 10^5 = 0.006 \quad (3)$$

The wall shear stress will be:

$$\tau_{air} = \frac{1}{2} \cdot C_f \cdot \rho_{air} \cdot v_{air}^2 = \frac{1}{2} \cdot 0.006 \cdot 1.225 \cdot 25^2 = 2.209\, Pa \tag{4}$$

Speed of fluid near boundary as frictional velocity:

$$u^* = \sqrt{\frac{\tau_{air}}{\rho_{air}}} = \sqrt{\frac{2.209}{1.225}} = 1.343\, m \cdot s^{-1} \tag{5}$$

Desired FLT for this analysis is y+ = 0.5 so the thickness of first cell should be:

$$y_{cell}^+ = \frac{y^+ \cdot \mu_{air}}{u^* \cdot \rho_{air}} = \frac{0.5 \cdot 1.7894 \cdot 10^{-5}}{1.343 \cdot 1.225} = 5.439 \cdot 10^{-6}\, m \tag{6}$$

Chosen thickness of FLT for this analysis is $1 \cdot 10^{-6}$ m so five times less than show calculation. This was done mainly because of bottom elements at object walls and roof ridge where the peak values of y^+ were found.

3 FINAL VOLUME MODEL

Computational domain size was set as 2B x 2B x 2H x 5B = 0.12 x 0.12 x 0.136 x 0.3 m (left, right, windward, above and leeward of object), the whole domain size was 0.42 x 0.32 x 0.136 m. This type of size of domain is the smallest one and is recommended for transient analysis only.

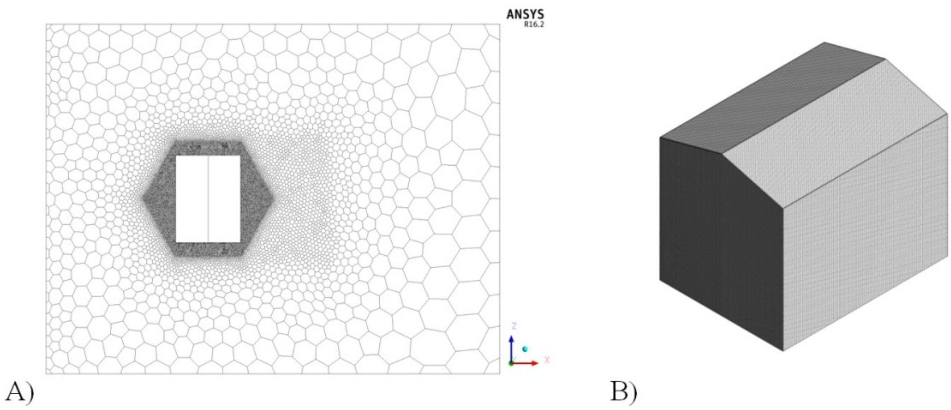


Fig. 3: A) bottom plane of computing domain, B) object with gable roof 15°

To improve solution was at bottom wall created small plane around object with dense mesh, mainly to secure low y+ at bottom corners of first layer of object with bottom wall. This can be seen on figure 3 A) and 4. Next to improve solution was mesh thickened around object and also behind it by solid box which was set to be body of influence with maximum element size set to 5x to object surface element size.

Object surface elements were set to be big as h/100 = 0.0006 m. Inflation was set on object walls with 15 layers, 1.2 grown rate and FLT was set to $1 \cdot 10^{-6}$. In sizing maximum element size and face were set to be h/2 = 0.03 and grown rate 1.2. Mesh was generate under ICEM and created were $4.442 \cdot 10^6$ tetrahedron elements. Polyhedral mesh as show figure 3 and figure 4 was generated under fluent solution module by converting whole domain into polyhedral mesh. This modification had direct influence on number of elements, which was smaller and of course CPU needs for the calculation, $1.513 \cdot 10^6$ polyhedral elements were created, what represents 2.93x reduction in number of elements.

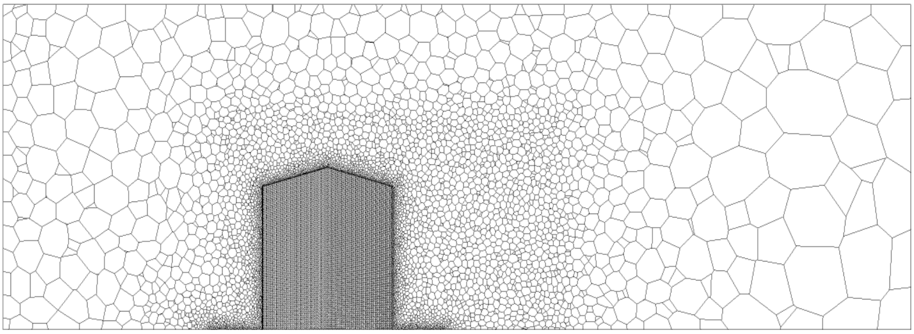


Fig. 4: Model mesh view in vertical axis plane

4 CFD SIMULATION

There are two fundamental approaches to design and analysis of engineering systems that involve fluid flow: experimentation and calculation. Modern engineers using both, where experimental in many cases are used to validate computational. CFD simulations can offer engineer good inside view all over computational domain and can quickly provide results of velocity magnitudes, pressures and many other turbulent parameters. CFD simulations input profiles were created using User Defined Functions (UDF).

4.1 Boundary conditions

Each surface had its "named section" to which were in solution module set boundary condition. Inlet was set as velocity inlet. Outlet as outflow as this boundary don't require additional information and data at exit plane are extrapolated from interior. Left, right and top faces of domain were defined as symmetry. Object faces and bottom plane were set as no slip walls without roughness.

Solution method was used SIMPLE pressure-velocity coupling scheme with second order spatial discretization, for transient formulation was used second order implicit method and was initialized with hybrid initialization with default settings for every simulation.

4.2 CFD input profiles

Setup for this model consisted of log law velocity profile in the form of UDF:

$$U(z) = \frac{u_{ref} \cdot \kappa}{\kappa \cdot \ln\left(\frac{z_{ref} + z_0}{z_0}\right)} \cdot \ln\left(\frac{z + z_0}{z_0}\right) \tag{7}$$

where:

u_{ref} - reference speed [m.s^{-1}],

z_{ref} - reference high [m],

z_0 - terrain roughness [m],

κ - Von Karman's constant [-],

$C\mu$ - model constant [-].

And specific dissipation rate ω was defined in UDF for the friction velocity as follows:

$$u^{*} = \frac{u_{ref} \cdot \kappa}{\ln\left(\frac{z_{ref}}{z_0}\right)} = \frac{25 \cdot 0.4}{\ln\left(\frac{0.06}{0.01}\right)} = 5.581\ m.s^{-1} \tag{8}$$

Turbulent dissipation rate profile:

$$\varepsilon(z) = \frac{u^{*3}}{\kappa \cdot (z + z_0)} \tag{9}$$

Specific turbulent dissipation rate profile:

$$\omega(z) = \frac{\varepsilon(z)}{k} \tag{1}$$

Where k is turbulent kinetic energy and was defined as follows:

$$k = \frac{u^{*2}}{\sqrt{C_\mu}} = \frac{5.581^2}{\sqrt{0.09}} = 18.603\, m^2.s^{-2} \tag{2}$$

4.3 DDES

Detached Eddy Simulation (DES) was introduced by Spalart and co-workers, to eliminate the main limitation of LES models by proposing a hybrid formulation that switches between RANS and LES based on the grid resolution provided. By this formulation, the wall boundary layers are entirely covered by the RANS model and the free shear flows away from walls are typically computed in LES mode. The formulation is mathematically relatively simple and can be built on top of any RANS turbulence model. DES has attained significant attention in the turbulence community as it was the first SRS model that allowed the inclusion of SRS capabilities into common engineering flow simulations. As the grid is refined below the limit the DES-limiter is activated and switches the model from RANS to LES mode.

The intention of the model is to run in RANS mode for attached flow regions, and to switch to LES mode in detached regions away from walls. This suggests that the original DES formulation, as well as its later versions, requires a grid and time step resolution to be of LES quality once they switch to the grid spacing as the defining length scale. DES limiter can already be activated by grid refinement inside attached boundary layers. In order to avoid this limitation, the DES concept has been extended to Delayed DES (DDES) by Spalart, [2,3]. Setup for this turbulence model simulation consists also from log law velocity profile (9) and specific turbulent dissipation rate profile (11).

4.4 WMLES

Wall Modeled LES (WMLES) is an alternative to classical LES and reduces the stringent and Re number-dependent grid resolution requirements of classical wall-resolved LES. The near-wall turbulence length scales increase linearly with the wall distance, resulting in smaller and smaller eddies as the wall is approached. This effect is limited by molecular viscosity, which damps out eddies inside the viscous sublayer (VS). As the Re number increases, smaller and smaller eddies appear, since the viscous sublayer becomes thinner. In order to avoid the resolution of these small near-wall scales, RANS and LES models are combined such that the RANS model covers the very near-wall layer, and then switches over to the LES formulation once the grid spacing becomes sufficient to resolve the local scales.

The advantage of WMLES is that the resolution requirements relative to the boundary layer thickness remain independent of the Reynolds number. For wall-normal resolution in WMLES, it is recommended to use grids with y+ ≈ 1 at the wall, [3].

Classical LES requires providing unsteady fluctuations at turbulent inlets/interfaces (RANS-LES interface) to the LES domain. This should make LES substantially more demanding than RANS, where profiles of the mean turbulence quantities (k and ε, or k and ω) are typically specified.

4.5 Study cases

Solution initialization consisted of hybrid initialization with standard setup. In both cases sampling of transient data started after 0.1 sec time of simulation, during this time was reached convergence criteria and the model was initiated for time sampling. The time step of 10^{-5} for LES simulation was chosen mainly because of convergence problems. A basic description about study cases can be found in table 1.

Tab. 1: Study cases with time sampling description.

Name	Spec.	Convergence	Steps / Iterations	Time step [s]	Time [s]
Case 1	DDES	10^{-4}	10 k / 153 093	0.0001	0.1 ≈ 1.1
Case 2	WMLES	10^{-4}	100 k / 1 183 459	0.00001	0.1 ≈ 1.1

5 EUROCODE

External pressure coefficients for wall and roof zones in table 2 are taken from Eurocodes, [4], for building with gable roof (duo pitched), a building with ground floor dimensions of d = 6m, b = 8m, roof elevation of 15° and h/d = 1. It needs to be noted that these coefficients are valid for wind direction from 0° up to 45° and analysis was done only for wind direction of 0°.

Tab. 2: EPC defined by Eurocodes for gable roof with 15° elevation and h / d = 1.

Walls	D	A (Right)	B (Right)	A (Left)	B (Left)	E
$C_{pe,10}$	0.80	-1.20	-0.80	-1.20	-0.80	-0.50
$C_{pe,1}$	1.00	-1.40	-1.10	-1.40	-1.10	
Roof	**F (Right)**	**F (Left)**	**G**	**H**	**J**	**I**
$C_{pe,10}$	-0.9	-0.9	-0.8	-0.3	-1.0	-0.4
$C_{pe,1}$	-2.0	-2.0	-1.5		-1.50	

6 RESULTS

6.1 External pressure coefficients

CFD analysis is focused on obtaining external pressure coefficients (EPC) on simple shaped low-rise building and these are compared to EPC which provide Eurocodes and can be found in Table 2. In tables 3 and 4 are listed mean EPC on object wall and roof zones which were calculated under CFD-Post processor as follows:

$$C_p = \frac{p - p_{ref}}{0.5 \cdot \rho_{ref} \cdot v_{ref}^2}. \tag{1}$$

where:

p - pressure [Pa],

p_{ref} - reference pressure [Pa],

ρ_{ref} - reference density of air [kg·m^{-3}],

v_{ref} - reference air speed [m·s^{-1}].

Reference point from which were taken reference values was situated close to inlet in reference high at top of the roof. Distribution of pressure coefficients can be seen on figure 5 B).

Tab. 3: External pressure coefficients on object walls.

Case	Walls	D	A (Right)	B (Right)	A (Left)	B (Left)	E
CASE 1	Cp_{min}	-0.76	-1.02	-1.67	-1.06	-1.04	-0.61
	Cp_{mean}	0.53	-0.61	-0.62	-0.62	-0.62	-0.43
	Cp_{max}	0.84	-0.32	-0.41	-0.58	-0.42	-0.37
CASE 2	Cp_{min}	-0.65	-1.06	-1.13	-0.92	-0.89	-0.35
	Cp_{mean}	0.72	-0.72	-0.55	-0.73	-0.54	-0.23
	Cp_{max}	0.95	-0.39	-0.19	-0.62	-0.18	-0.14

Tab. 4: External pressure coefficients on object roof.

Case	Roof	F (Right)	F (Left)	G	H	J	I
CASE 1	Cp_{min}	-2.63	-2.57	-2.81	-3.20	-3.62	-1.06
	Cp_{mean}	-0.79	-0.79	-0.95	-0.68	-1.47	-0.66
	Cp_{max}	-0.42	-0.42	-0.55	-0.35	-0.88	-0.38
CASE 2	Cp_{min}	-2.26	-2.30	-1.60	-2.04	-2.26	-0.74
	Cp_{mean}	-1.16	-1.14	-1.05	-0.75	-0.80	-0.47
	Cp_{max}	-0.65	-0.67	-0.77	-0.40	-0.63	-0.26

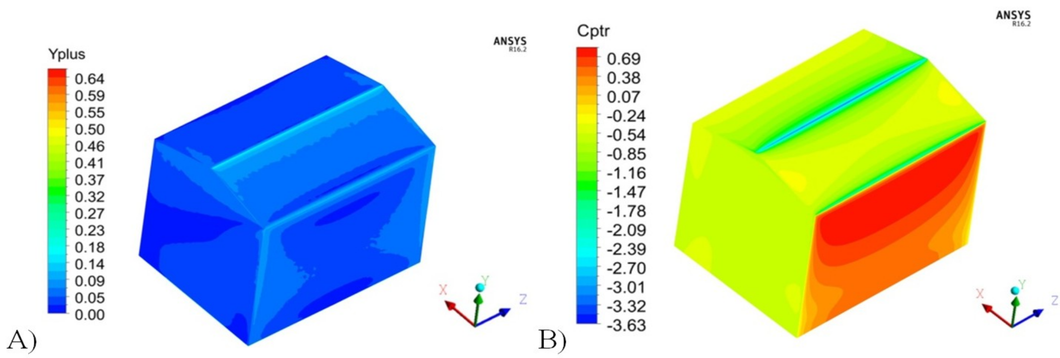


Fig. 5: Study Case 1: A) y+ on object, B) EPC on object

Case 1: From walls, zone D as windward face was predicted by CFD with mean value by 66% from EC value and with peak value up to 84% from EC. Zone A was represented by only 52% of EC value with 76% peak value. Zone B predicted with mean values of 78% of EC but with peak value of 152%. Leeward zone E was represented 86% of EC value with peak value of 122%.

From roof zones, zone F was predicted with mean value that represents only 88% from EC given value but with peak close to EC 132%. G zone represents 119% of EC value with 187% peak value. Zone H is represented with value that is 250% of EC but with 1067% peak value inside zone. Zone J was over predicted by twice the EC value 147% and almost 241% in peak value. Zone I was around 65% higher than EC with 265% high peak value.

Case 2: From walls, zone D was predicted with mean value by 90% from EC with peak value up to 95% from EC. Zone A was represented by only 61% of EC value with 66% peak value. Zone B predicted with mean value of 69% of EC with peak value of 103%. Leeward zone E was represented 46% of EC value with peak value of 70%.

From roof zones, zone F was predicted with mean value that represents only 128% from EC with peak very close to EC 113%. G zone represents 131% of EC value with 106% peak value. Zone H is represented with value that is 250% of EC but with 680% peak value. Zone J was predicted by 80% the EC value and almost 150% in peak value. Zone I was around 18% higher than EC with 185% high peak value.

6.2 DDES vs WMLES

Histogram in figure 6 is showing geometrical area count distribution of EPC in roof zone F (Left). From which we can see significant difference in predicted values from both cases and also lone peak values, which were observed at corners of object.

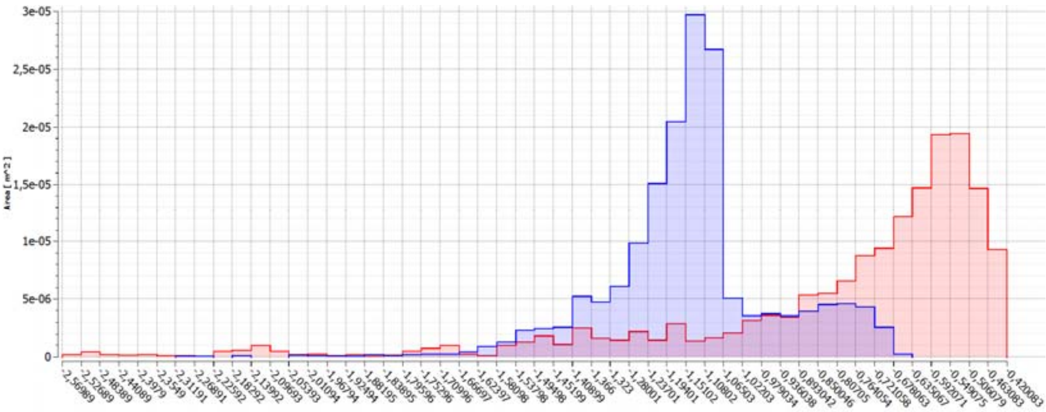


Fig. 6: Histogram of EPC in roof zone F (Left), geometrical count, Case1 is red and Case 2 is blue

In table 5 are listed values of mean values of EPC on roof per square meter. Here we can see that results in longitudinal symmetry are quite similar for each case.

Tab. 5: Mean external pressure coefficients on object roof per square meter.

Roof	X\Z	1	2	3	4	5	6	7	8
Case1	1	-0.664	-0.786	-0.855	-0.884	-0.887	-0.853	-0.785	-0.660
	2	-0.391	-0.458	-0.514	-0.540	-0.539	-0.513	-0.456	-0.388
	3	-0.760	-0.935	-0.999	-1.024	-1.023	-0.997	-0.931	-0.759
	4	-1.223	-1.369	-1.425	-1.443	-1.442	-1.421	-1.365	-1.217
	5	-0.747	-0.747	-0.766	-0.772	-0.770	-0.761	-0.740	-0.738
	6	-0.541	-0.517	-0.505	-0.495	-0.492	-0.497	-0.507	-0.530
Case2	1	-1.056	-1.121	-1.052	-1.030	-1.030	-1.059	-1.143	-1.080
	2	-0.525	-0.712	-0.850	-0.877	-0.888	-0.869	-0.718	-0.508
	3	-0.676	-0.710	-0.739	-0.753	-0.749	-0.729	-0.686	-0.663
	4	-0.799	-0.798	-0.754	-0.733	-0.728	-0.749	-0.795	-0.812
	5	-0.534	-0.548	-0.554	-0.549	-0.544	-0.536	-0.536	-0.545
	6	-0.345	-0.338	-0.354	-0.363	-0.361	-0.352	-0.339	-0.357

In figure 7 we can see distribution on EPC in area Z8-X1 from table 5, which is also a corner area of zone F (Left), extreme lone peak values were observed at the edges.

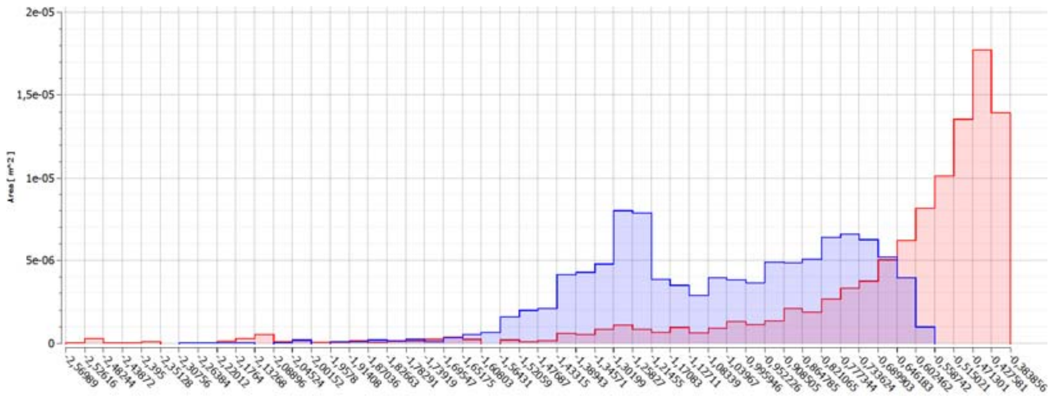


Fig. 7: Histogram of EPC in roof zone Z8-X1, geometrical count, Case1 is red and Case 2 is blue

Next comparison between both cases can be seen on figures 8 to 11, starting with velocity profiles in longitudinal axis plane. Both cases used the same UDF file with inputs. While in Case 1 were as turbulence inputs turbulent kinetic energy k and specific turbulent dissipation rate ω, in Case 2 was turbulence modeled also by using spectral synthesizer, which had influence on velocity profiles, this can be seen on figure 8 B).

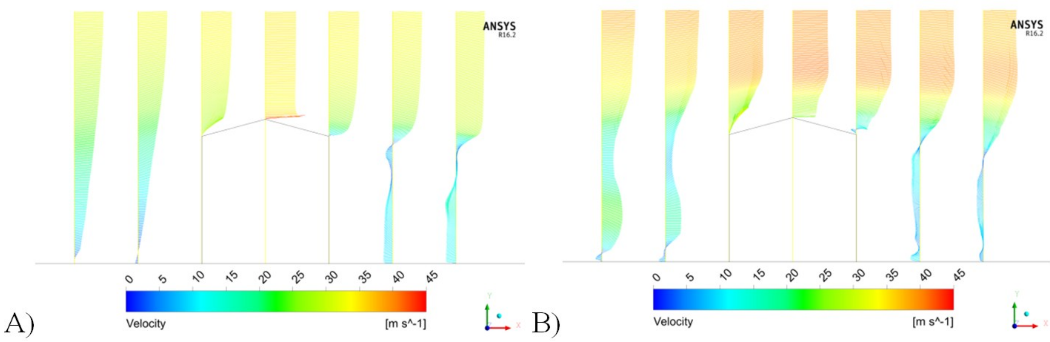


Fig. 8: Velocity profiles in middle vertical plane: A) Case 1, B) Case 2

In figure 9 are shown resolved contours of mean velocity profiles in vertical axis plane for both cases. Here can be seen predicted point of stagnation on windward face in both cases approximately in the same high, which is around 2/3 of building high. Significant differences were found in windward and mainly leeward recirculation, see also figure 10.

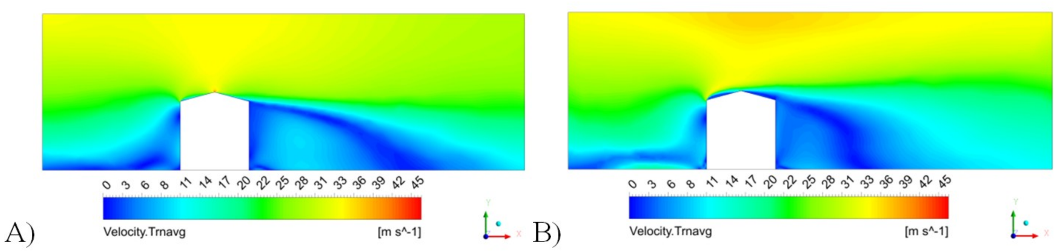
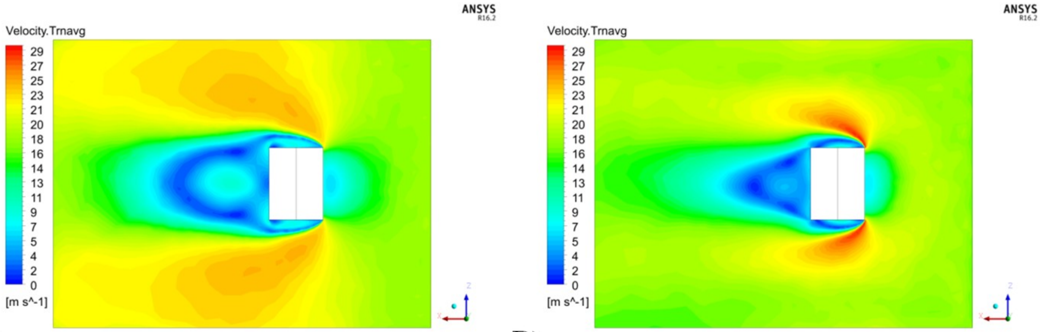


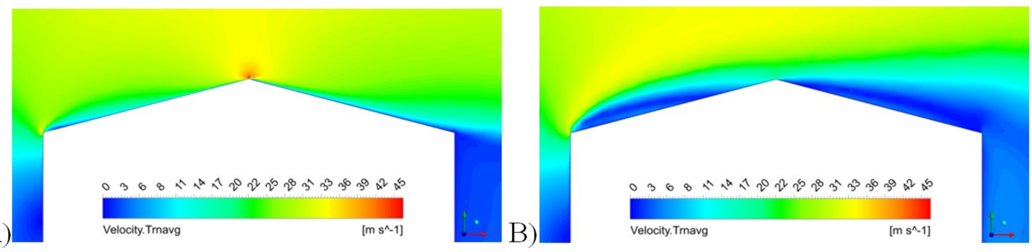
Fig. 9: Contours of mean velocity in middle vertical plane: A) Case 1, B) Case 2

Figure 11 is showing detailed view at roof ridge, where the separation zones on roof were under predicted in Case 1 (windward roof and leeward roof).



A) B)

Fig. 10: Contours of mean velocity in horizontal plane at +0.03 m: A) Case 1, B) Case 2



A) B)

Fig. 11: Contours of mean velocity in middle vertical plane, ridge detail: A) Case 1, B) Case 2

At the end, the biggest difference between both cases was in consumed time per simulation, while in Case 1 simulation took 3 days, in Case 2 it was 30 days to complete by using desktop computer.

7 CONCLUSIONS

From presented results it's hard to talk about any accuracy, both models performed very well (no problems with convergence). Some differences were found in predicted EPC between used turbulence models. Reached time averaged mean values of EPC were higher compared to those that are provided by Eurocode. Generally, walls were better predicted by CFD than roof zones.

ACKNOWLEDGMENT

This contribution is the result of the researches supported by Slovak Grant Agency VEGA. Registration numbers of the projects are 1/0256/16 and 1/0951/16.

LITERATURE

[1] ANSYS Fluent 14.0 Theory Guide

[2] Spalart, P. & S. Deck & M. Shur & K. Squires & M. Strelets & A. Travin. A New Version of Detached Eddy Simulation, Resistant to Ambiguous Grid Densities*, Journal of Theoretical and Computational Fluid Dynamics 20*, 2006, pp. 181–195. DOI 1007/s00162-006-0015-0

[3] Menter F.R. Best Practice: Scale-Resolving Simulations in ANSYS CFD. Version 2.0. http://www.ansys.com/staticassets/ANSYS/staticassets/resourcelibrary/techbrief/tb-best-practices-scale-resolving-models.pdf, 2015.

[4] Eurocode 1: Actions on structures - General actions - Part 1-4: Wind actions.