Assessment of Turbulence Models for Flow around a Surface-Mounted Cube

Sercan Dogan, Sercan Yagmur, Ilker Goktepeli, and Muammer Ozgoren
Selcuk University, Konya, Turkey
Email: {sercandogan, syagmur, ilkergoktepeli}@selcuk.edu.tr

***Abstract*—Flow over a bluff body constitutes one of the most fundamental phenomena for engineering applications. Even though a cube is considered as a simple bluff body to examine with respect to the flow structure, there is more complicated flow physics around the cube than expected. Thus, a cube just like other bluff bodies is mostly used in the comparison process of numerical and experimental results to determine the more suitable turbulence model in case of undertaken problem. For this reason, in this paper, turbulence models have been employed to investigate the flow characteristics around a surface-mounted cube at *Re* = *3700* based on the edge length of the cube in terms of Computational Fluid Dynamics (CFD) and then compared with experimental results in the literature. Normalized and time-averaged results of velocity vector fields, streamwise and cross-stream velocity components, vorticity contours and streamline patterns have been numerically obtained by using k-ε Re-Normalization Group (RNG), k-ω Shear Stress Transport (SST) and Large Eddy Simulation (LES) turbulence models. LES turbulence model has presented the best prediction of hydrodynamic characteristics for the body among the investigated turbulence models in this study. Although k-ω SST turbulence model was the second successful one after LES method among the investigated turbulence models for the estimation of flow structure around the cube, k-ε RNG turbulence model was failed to capture the flow fluctuations in the wake region of the geometry.**

***Index Terms*—bluff body, CFD, cube, turbulence model, vorticiy, wake region**

I. Introduction

In various engineering applications, flow around a bluff body represents common example. For instance, a cube is taken into consideration as a simple bluff body for the investigation of the flow structure. However, more complex flow characteristics form around the cube than expected. With this manner, the cube as a bluff body is also utilized in the comparison of numerical and experimental studies for the determination of more appropriate turbulence model related with the case. Several experimental and numerical studies have been done about flow past a cube so far. Larousse *et al.* (1991) examined the flow past surface-mounted three-dimensional bodies in both air and water channels at *Re = 10000.* Although preliminary experiments were done in the water channel, most of experiments were conducted in the air channel. It was a deduction that three-dimensional obstacles caused a large scale unsteadiness leading to higher Reynolds shear stress [1]. Martinuzzi and Tropea (1993) investigated the flow structure around surface-mounted prismatic obstacles with square cross-sections by utilizing various visualization techniques in a water channel at channel height based Reynolds numbers of $8 \times 10^4 \leqslant Re \leqslant 1.2 \times 10^5$ [2]. Utnes and Ren (1995) presented a numerical model to compute the flow around a surface-mounted cube via standard k-ε turbulence model and compared with measurements [3]. Thomas and Williams (1999) numerically studied on flow over a surface-mounted cubical body by using LES method under the conditions of a typical full-scale urban wind environment at *Re = 10000*. This method was found sufficient for capturing the conical vortices over the roof of the body [4]. Krajnovic and Davidson (2000) used LES turbulence model to compute the flow characteristics of a sharp-edged surface-mounted cube at obstacle height based Reynolds number of *Re = 40000*. They have explained that LES method was used in the simulation of real-life experiments [5]. Yakhot *et al*. (2006) conducted a Direct Numerical Simulation (DNS) to examine the flow characteristics of a wall-mounted cube at *Re = 5610*. A horseshoe vortex in front of the cube and an arch-type vortex behind the cube have been obtained [6]. Lim *et al*. (2009) did a study about the flow past a surface-mounted cubical objects positioned in a turbulent boundary layer at *Re = 20000* based on the cube height. LES turbulence model has been used in the numerical simulations and experiments have been done in the wind tunnel. The results have shown that LES turbulence model was useful method for the studies related with channel flow [7]. Paik *et al*. (2009) utilized many types of Detached Eddy Simulation (DES) for the investigation of flow over dual wall-mounted cubes at *Re = 22000*. The time-averaged horseshoe vortex was attained as in the results taken from measurements but its position for numerical simulation is slightly closer to the leading cube when compared to the experiments [8]. Liakos and Malamataris (2014) considered the flow around a cube for low to moderate Reynolds numbers in the numerical study. Tornado-like vortex has formed through the side of the body due to the dihedral angle between the side of the geometry and the base of the channel [9]. Liao and Chen (2015) experimentally studied on flow around various obstacles by using Particle Image Velocimetry (PIV) system at *Re = 3700.* They have stated that position and size of vortex

Manuscript received January 21, 2017; revised April 23, 2017.

doi: 10.18178/ijmerr.6.3.237-241

formation depended on the obstacle shape. The length of recirculation zone affected flow recovery in the wake region [10]. Yagmur *et al.* (2015) prepared a study covering the experimental and numerical investigation of flow around various bluff bodies including circular, square and triangular cross-sectional cylinders at *Re = 5000* and *Re = 10000*. The wake length has been obtained as longer at *Re = 5000* by using PIV and LES methods for all cases. Furthermore, the longest wake length was yielded for triangular cylinder whereas the square cylinder has the shortest one in this classification [11]. The aim of the present study is to compare different turbulence models in terms of hydrodynamic flow characteristics in order to determine the most proper turbulence model among the investigated ones with respect to the experimental data from the literature.

II. Material and Method

Position For the numerical investigations, the turbulent viscosity is computed with various ways in the literature. In this study, three different turbulence methods of ANSYS-Fluent software (2013) [12] were utilized for the simulation of the flow field around the cube.

In this study, flow domain used by Liao and Chen (2015) [10] has been prepared to examine the flow characteristics of the surface-mounted cube at the edge length based Reynolds number of *Re = 3700* as shown in Fig. 1 with two-dimensional drawing.

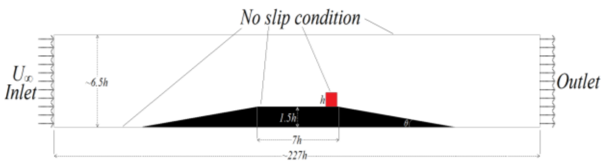
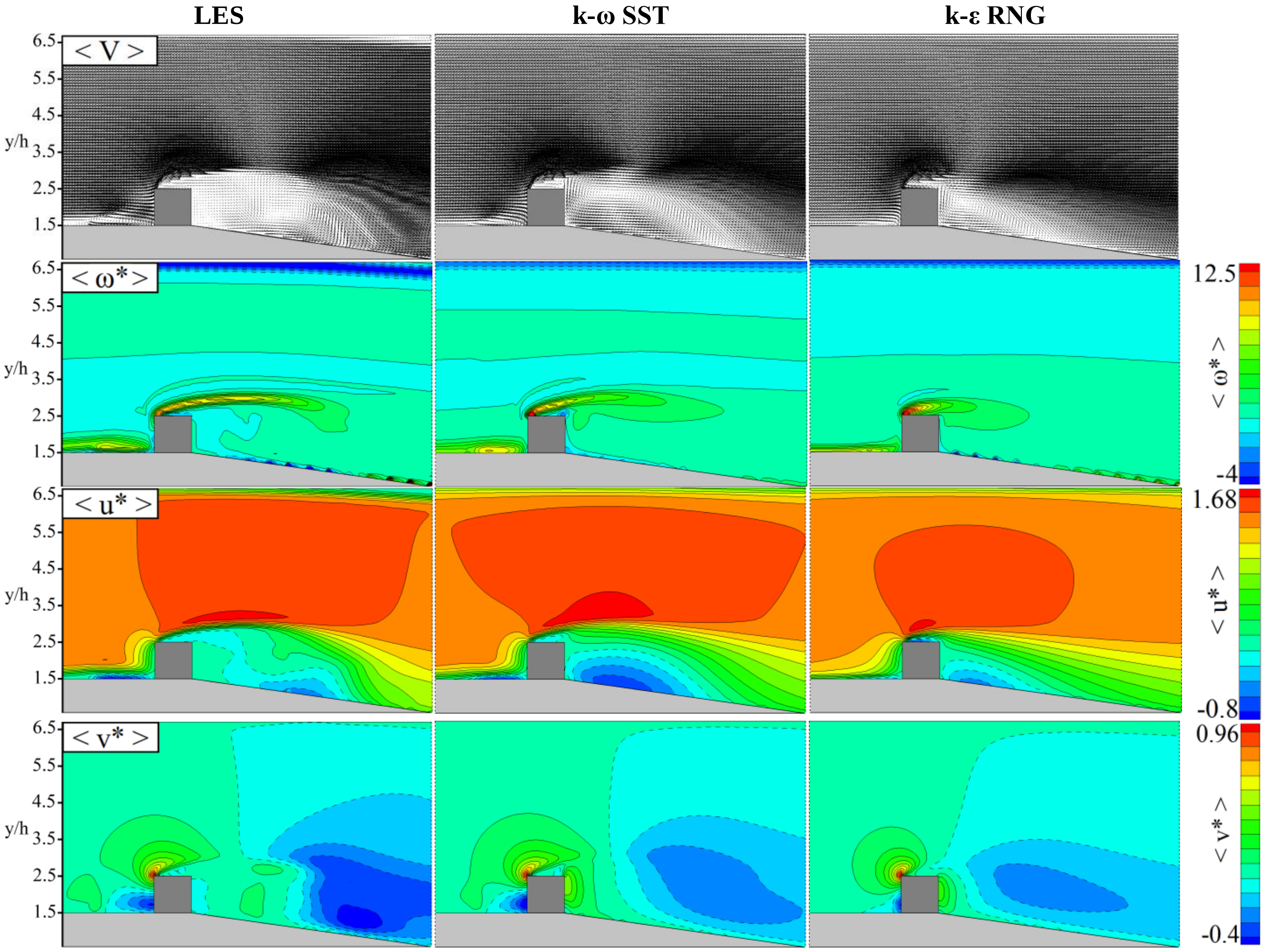


Figure 1. Flow domain on the XY plane

The surface-mounted cube has been located at the center point between two side walls. The last part of the platform at the downstream of the cube is a slope having an angle of $\theta = 8°$ just as in the first part. The inlet of the flow domain was started *117h* far away from the rear side of the cube and from the rear side of the cube to the outlet of the flow domain is about *110h*. By the way, *h = 30 mm*, an edge of the cube is the characteristic length for the cube. Third dimension is the width as approximately *3.5h* for the flow domain. Velocity inlet and pressure outlet have been defined at the inlet and the outlet, respectively. Wall boundary condition has been chosen as no slip boundary condition for the rest of the flow domain. The flow domain was created with a number of structured cut-cell grids were used to analyze flow field. To receive better results and view detailed vortex structures, the grid size of the flow domain is nearly 5.4×10^6 for the finer grid number in the flow domain, which provides enough mesh size for the simulate this case.



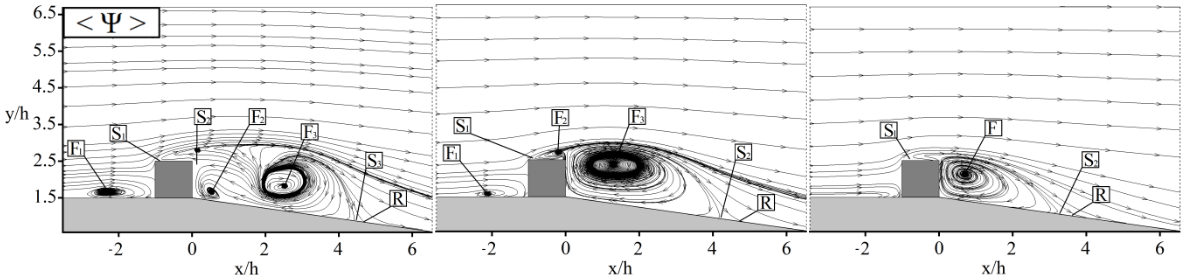


Figure 2. Comparison of normalized and time-averaged results of velocity vector fields <*V*>, vorticity contours <*ω**>, streamwise velocity components <*u**>, cross-stream velocity components <*v**> and streamline patterns <*ψ*> around the surface-mounted cube for XY plane at *Re* = *3700*

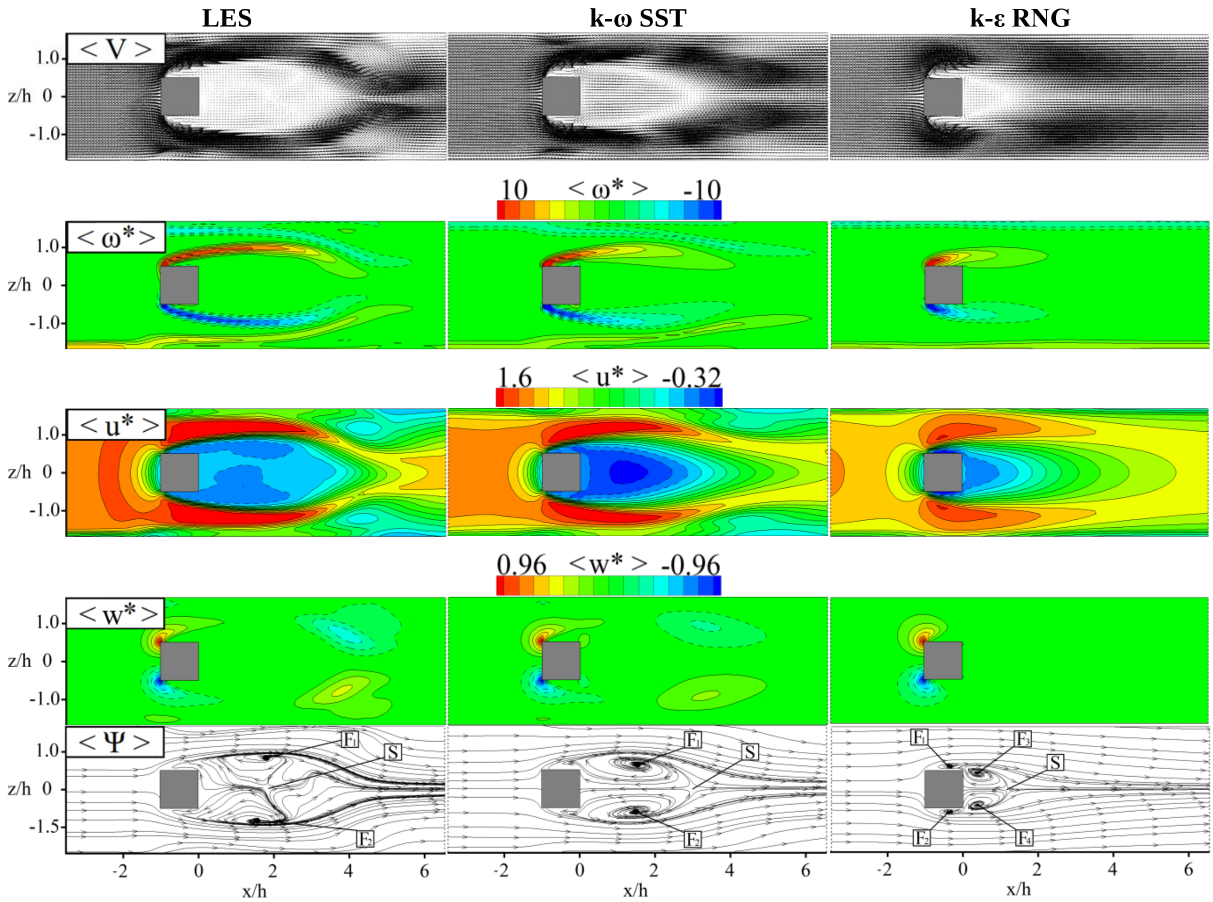


Figure 3. Comparison of normalized and time-averaged results of velocity vector fields <*V*>, vorticity contours <*ω**>, streamwise velocity components <*u**>, cross-stream velocity components <*w**> and streamline patterns <*ψ*> around the surface-mounted cube for XZ plane at *Re* = *3700*

III. Results and Discussion

Flow around a surface-mounted cube have been examined by utilizing three turbulence models which are LES, k-ω SST and k-ε RNG methods. Comparison of different turbulence models at $Re = 3700$ has been shown in Fig. 2 and Fig. 3. Time-averaged and normalized results of velocity vector fields <*V*>, vorticity contours $<\omega^* = \omega L/U_\infty>$, streamwise velocity components $<u^* = u/U_\infty>$ and streamline patterns <*ψ*> around the geometry have been obtained and given for both XY (Fig. 2) and XZ (Fig. 3) planes of flow domain. However, cross-stream velocity components $<v^* = v/U_\infty>$ have only been presented in case of XY plane and third velocity components $<w^* = w/U_\infty>$ have only been shown for XZ plane. The dimensions of all images have been normalized with the edge length, *h*, of the surface-mounted as *x/h*, *y/h* and *z/h*. The maximum and minimum values are given with a legend bar for the contours in each figure and divided with 15 levels. The positive layers of flow patterns are displayed with continuous line contours as red background color, while the negative layers are indicated with dashed line contours as blue background color.

Velocity vector fields <*V*> of the surface-mounted cube have been given in Fig. 2 for XY plane of flow domain. According to Fig. 2, a reverse flow occurred in the wake region of the geometry due to flow separation from the sharp-tip corner. However, the wake length including reverse flow formed distinctly for each turbulence model investigated. For instance, the longest wake region has been obtained by LES method whereas k-ε RNG turbulence model remained limited for capturing reverse flow compared to other methods. Moreover, LES method was also found more successful in terms of detecting the small scale eddy just front of the cube near to the platform. In Fig. 2, time-averaged vorticity $<\omega^*>$ contours indicated that flow separation from sharp-tip corner for all turbulence models. Because of the shear layer, the flow motion from upper corner of front side to the wake region occurred in the clockwise. After the flow separation from the body, small scale eddies roll up to form in the wake. Vorticity contours have been given as $-4 \le <\omega^*> \le 12.5$ for all turbulence models. Maximum values of vorticity contours were observed over the cube near to the flow separation zone at the sharp-tip corner. The contours of the time-averaged and normalized streamwise velocity components $<u^*>$ have been presented in the range of $-0.8 \le <u^*> \le 1.68$ for flow past the cube. It was seen that flow accelerated while passing the sharp-tip corner and approached the maximum value. Flow acceleration was observed in the region near to the sharp-tip corner and this region was the largest in case of LES method whilst k-ε RNG turbulence model showed the narrowest one. Negative streamwise velocity values have also been attained as an indication of reverse flow. Negative minimum values of streamwise velocity components were more obviously for both LES and k-ω SST than k-ε RNG around $x/h = -2$ over the plate because of increment in static pressure. Furthermore, negative minimum values of velocity components were seen after the front upper corner on the cube and over the inclined surface with different sizes of clusters. In addition, more chaotic flow structure was seen in the wake when LES turbulence model was considered. Cross-stream velocity components $<v^*>$ are presented in Fig. 2 as $-0.4 \le <v^*> \le 0.96$ at $Re = 3700$. Positive maximum values of cross-stream velocity components formed at the sharp-tip corner while time-averaged negative maximum values were dominant for the front dihedral angle and far wake region of the model. Additionally, unsymmetrical flow structure occurred due to the interaction of the body and the platform. F shows the focus of the rotational flow field in a limit cycle; S is saddle point and accepted as an evident for intersections of streamlines that represents free stagnation point in the flow field and also R stands for the reattachment point of the flow for time-averaged streamline patterns $<\psi>$ in Fig. 2 for all cases. There are three foci points and saddle points and only one reattachment point acquired for LES. However, there are only one reattachment point, three foci points and two saddle points for k-ω SST method. Moreover, there are two saddle points, only one focus point and only one reattachment point obtained for k-ε RNG method. Locations of reattachment points are nearly at $x/h = 4.5$ for LES and k-ω SST methods, but it was at the approximate point of $x/h = 3.5$ in case of k-ε RNG turbulence model.

Velocity vector fields $<V>$ for the cube have been presented in Fig. 3 with XZ plane of flow domain in the cross-section passing at the height of $y/h = 2$ with respect to the origin. Length of the wake was determined as longer for LES method while k-ε RNG turbulence model had the shortest. Time-averaged vorticity $<\omega^*>$ contours showed that flow separation occurred from sharp-tip corners for the range of $-10 \le <\omega^*> \le 10$ in Fig 3. Vortex shedding formed from the upper corner in the clockwise as positively and from the lower corner in the counter-clockwise as negatively. However, secondary vortices developed near to the wall under boundary layer effect and these were negative values for upper region and positive ones for lower region as clearly seen for both LES and k-ω SST models. The effect of flow separation was proceeded up to approximately $x/h = 5$ for LES and k-ω SST turbulence models while up to nearly $x/h = 2$ in terms of k-ε RNG method. Time-averaged and normalized streamwise velocity components $<u^*>$ have been given in the range of $-0.32 \le <u^*> \le 1.6$ for flow around the cube. It was determined that flow accelerated while passing the front sharp-tip corners and reached the maximum value. Nevertheless, flow decelerated in the vicinity of walls because of boundary layer effect just like in the stagnation point of upstream owing to higher static pressure. Flow acceleration was seen in the region near to the sharp-tip corners and this region was the most dominant in terms of LES method whereas k-ε RNG turbulence model indicated the narrowest zone for this case. Negative streamwise velocity values have also been calculated and negative minimum values were more effective up to $x/h = 4$ for LES and k-ω SST models in the wake and it was approximately at the point of $x/h = 2$ in case of k-ε RNG method. Time-averaged cross-stream velocity components $<w^*>$ in XZ plane have been shown in Fig. 3 for the range of $-0.96 \le <w^*> \le 0.96$ and these velocity components occurred symmetrically. Time-averaged cross-stream velocity clusters were also observed at the upstream as positively for upper sharp-tip corner and negatively for lower one. On the other hand, additional two clusters formed in the wake region different from upstream side. Time-averaged streamline patterns $<\psi>$ were shown in Fig. 3 to compare all turbulence models. There are two foci points and only one saddle point attained for both LES and k-ω SST methods. However, there are four foci points and only one saddle point obtained in case of k-ε RNG method.

As a conclusion, it has been stated that satisfactory results were not been provided by using k-ε RNG turbulence model. Even though k-ω SST turbulence model presented better results than k-ε RNG method, it remained behind LES turbulence model by a narrow margin in terms the flow structure around the surface-mounted cube at $Re = 3700$. Similar result about the comparison of these turbulence models was also stated by Ozgoren *et al*. (2015) [13].

IV. Conclusion

In the present study, flow characteristics of the surface-mounted cube depending on the Reynolds number of $Re = 3700$ were investigated with three different turbulence models in terms of CFD. It was explained that satisfactory results were not been acquired via k-ε RNG and k-ω SST methods and LES turbulence model yielded more successful results after time-dependent analyses. Flow separation from the front upper sharp-tip corner has been detected by using all three turbulence models. In that point where flow separation occurred, vorticity and cross-stream velocity components reached their maximum values. When vorticity contours of XZ plane were considered, influence of vortices detached from the surface of the cube on the channel walls was clearly seen. However, k-ε RNG was incapable of capturing this aforementioned effect. For this reason, LES and k-ω SST turbulence models can be preferred in channel flows used for industrial applications such as the determination of optimum design in terms of vortex flowmeters, design on building location, towers and so on.

ACKNOWLEDGMENT

The authors would like to acknowledge the funding of Scientific Research Project (BAP) Office of Selcuk University.

REFERENCES

[1] A. Larousse, R. Martinuzzi, and C. Tropea, "Flow around surface-mounted, three-dimensional obstacles," in *Proc. 8th Symposium on Turbulent Shear Flows*, Munich, 1991, pp. 127-139.

[2] R. Martinuzzi and C. Tropea, "The flow around surface-mounted,prismatic obstacles placed in a fully developed channel flow," *Journal of Fluids Engineering*, vol. 115, pp. 85–92, Mar. 1993.

[3] T. Utnes and G. Ren, "Numerical prediction of turbulent flow around a three-dimensional surface-mounted obstacle," *Applied Mathematical Modelling*, vol. 19, pp. 7–12, Jan. 1995.

[4] T. G. Thomas and J. J. R. Williams, "Simulation of skewed turbulent flow past a surface mounted cube," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 81, pp. 347–360, May 1999.

[5] S. Krajnovic and L. Davidson, "Flow around a three-dimensional bluff body," in *Proc. 9th Millenium International Symposium on Flow Visualization*, Edinburgh, 2000, pp. 1–10.

[6] A. Yakhot, H. Liu, and N. Nikitin, "Turbulent flow around a wall-mounted cube: a direct numerical simulation," *International Journal of Heat and Fluid Flow*, vol. 27, pp. 994-1009, Dec. 2006.

[7] H. C. Lim, T. G. Thomas, and I. P. Castro, "Flow around a cube in a turbulent boundary layer: LES and experiment," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 97, pp. 96–109, Feb. 2009.

[8] J. Paik, F. Sotiropoulos, and F. Porte-Agel, "Detached eddy simulation of flow around two wall-mounted cubes in tandem," *International Journal of Heat and Fluid Flow*, vol. 30, pp. 286–305, Apr. 2009.

[9] A. Liakos and N. A. Malamataris, "Direct numerical simulation of steady state, three dimensional, laminar flow around a wall mounted cube," *Physics of Fluids*, vol. 26, May 2014.

[10] B. Liao and S. Chen, "Experimental study of flow past obstacles by PIV," *Procedia Engineering*, vol. 126, pp. 537-541, 2015.

[11] S. Yagmur, S. Dogan, M. H. Aksoy, E. Canli, and M. Ozgoren, "Experimental and numerical investigation of flow structures around cylindrical bluff bodies," in *Proc*. *EPJ Web of Conferences*, May 2015, pp. 1-7.

[12] Fluent 14.0 User Guide, Fluent Inc., Canonsburg, PA, 2013.

[13] M. Ozgoren, S. Yagmur, S. Dogan, M. H. Aksoy, E. Canli, and I. Goktepeli, "Flow characteristics around a vortex flow meter via CFD and PIV methods," in *Proc. 10th Pacific Symposium on Flow Visualization and Image Processing*, Naples, 2015, pp. 1-16.

Sercan Dogan was born in 1988, is currently studying on his Ph.D. thesis in Selcuk University. He gets his Bachelor's Degree at Selcuk University in 2010 and MSc degree at Selcuk University in 2013. He is working as research assistant at Mechanical Engineering Department of Selcuk University, Konya, Turkey. His main research interests are PIV, CFD, heat transfer, renewable energy and fluid mechanics.

Sercan Yagmur was born in 1988, is currently studying on his Master thesis in Selcuk University. He gets his Bachelor's Degree at Selcuk University in 2012 and MSc degree at Selcuk University in 2016. He is working as research assistant for 2 years at Mechanical Engineering Department of Selcuk University, Konya, Turkey. His main research interests are PIV, CFD, fluid mechanics, and aerodynamics.

Ilker Goktepeli graduated from Eskisehir Osmangazi University (Turkey) in 2014 and is doing master degree in Selcuk University (Turkey). He has been working as a research assistant at the Department of Mechanical Engineering of Selcuk University (Turkey) since 2015. Research assistant Goktepeli, studies on experimental and numerical fluid mechanics applications, renewable and conventional energy systems.

Muammer Ozgoren graduated from Cukurova University in 1993 and then completed his MSc and PhD studies at the same University in sequence 1996 and 2000. He stayed at Lehigh University in USA between 1998 and 2000 to perform his Ph.D studies. Ozgoren works on experimental and numerical fluid mechanics applications, renewable and conventional energy conversion systems, and efficient use of energy and refrigeration technologies. He published national and international papers more than 100 papers for aforementioned subjects.