Numerical Study of the Influence of Vortex Generator on Flow Structure and Heat Transfer Around a Cube Mounted in a Channel

Lecturer Dr.
Zainab Hasson Hassan*

Ass. Lecturer
Abdul jabbar Jwair shamikh**

* Machine and Equipment Engineering Dep., University of Technology, Baghdad-Iraq.
** Machine and Equipment Engineering Dep., University of Technology, Baghdad-Iraq.

Abstract

Numerical Investigation of the influence of vortex generator on the enhancement of heat transfer from a wall mounted cube which placed in the middle of a rectangular channel have been conducted using finite – volume method **.** The momentum and convective heat transfer equations were discretized and solve using large – eddy simulation (LES)**.** In order to study the influence of vortex generator on the flow structures and heat transfer coefficient**,** the flow which considered a fully developed turbulent flow and the convective heat transfer equations were solved around two cube configurations: a smooth cube and a cube with vortex generator attached to its surface **.** The vortex generator used in this paper is a simple rib attached to the top and the side walls of the cube**,** in the steam wise middle of the cube **.** The flow Reynolds number based on the bulk velocity and the height of the channel was 13000 **.** The LES results were compared with the experimental data of [6],a good agreement was obtained . Also the results showed that the flow in the boundary layer around the cube with vortex generator is more turbulent and unsteady than the flow around the smooth cube without the vortex generator. More turbulent structures are generated close to the surface of the cube resulting in a good mixing of heat and hence high heat transfer coefficient and this investigation is a efficient way to enhance the heat transfer from the cube with vortex generator **.**

Keywords: Large eddy simulation (LES) **,** Smagorisky sub – grid scale (SGS) model **,** finite volume method **,** flow past a cube in channel **,** heat transfer coefficient **.**

الخلاصة

هذا البحث يصف دراسة عددية لإيجاد تأثير وجود مولد الدوامة الموجودة على مكعب موضوع في وسط مجرى للهواء على تحسين عملية انتقال الحرارة في جدران المكعب باستخدام طريقة الحجوم المحددة **.** تم برمجة معادلات الزخم وانتقال الحرارة بالحمل باستخدام طريقة (LES) **.**
من اجل دراسة تأثير وجود مولد الدوامة على شكل الجريان ومعامل انتقال الحرارة ، تم حل معادلة الجريان الذي تم اعتباره كامل النمو واضطرابي وكذلك معادلة انتقال الحرارة **.** بالحمل حول مكعبين : الأول مكعب خالي

من مولد الدوامة والثاني مكعب مثبت علية مولد دوامة . مولد الدوامة المستخدم في هذه الدراسة عبارة عن عائقة بسيط موضوع بتلامس فوق وجانبي جدران المكعب ومثبت من منتصفه باتجاه الجريان .
أما عدد رينولدز المحسوب على أساس متوسط السرعة وارتفاع المجرى فكان 13000 . تم مقارنة النتائج المستخدمة من هذه الدراسة مع نتائج عملية فكان التوافق بينها جيد . كما أوضحت النتائج أن الجريان ضمن الطبقة المتاخمة حول المكعب بوجود مولد الدومة يكون أكثر اضطرابا وغير مستقر مقارنة مع الجريان حول المكعب الخالي من مولد الدوامة وهذا الاضطراب الكثير الحاصل في الجريان تم توليده بالقرب من سطح المكعب ناتج من الخلط الجيد للحرارة ولهذا يزداد معامل انتقال الحرارة ، وهو الغرض المطلوب من هذه الدراسة وهو تحسين انتقال الحرارة من سطح المكعب بوجود مولد للدوامة .

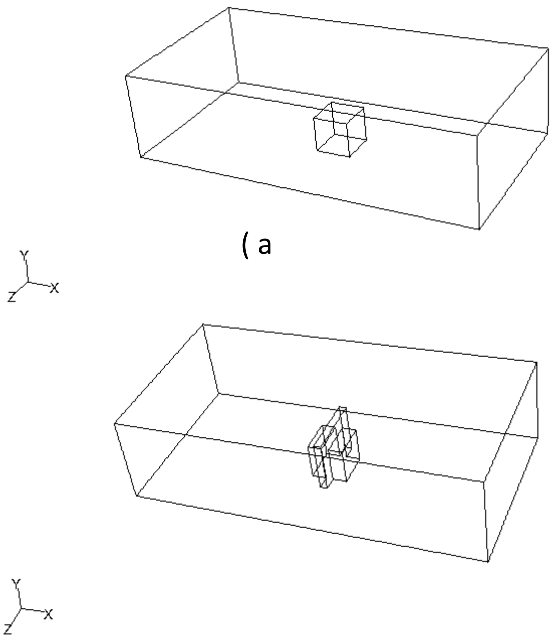
1 – Introduction

It is generally believed that local over heating of integrated circuit is the major cause for the technical failure of electronic equipment. Therefore, finding a method for an efficient heat removal from these components is necessary to ensure steady reliable long-term options. In general the heat removal from an integrated circuit depends very much on the flow structure around it. The electronic component is a bluff body that produces flow with separations. The flow around an integrated electronic circuit can be approximated to be similar to the flow round a cube mounted on a surface.
Previous investigations of the flow around a surface - mounted cube found that different kinds of flow instabilities give rise to different flow structures around the cube. Different numerical methods have been used to study the flow around a single cube mounted on a surface. Krajnovic and Davidson [1] and [2] used large-eddy simulation to investigate the flow structures around a surface mounted cube in a fully developed channel flow. They used different technique to visualize the flow. In their simulation, the Reynolds number was 13000 based on the incoming mean bulk velocity and cube height. They found that the flow separates from the surface of the cube on the lateral and the top side faces of the cube. Yakhot et al. [3] studied the same cube using Direct Numerical Simulation and they got similar flow structure.
Martinnzzi [4] experimentally investigated the complex flow structures which formed on the top – side face of the mounted cube on a surface. Many attempts have been done in the past to understand the physics of the flow around single cube but, very few attempts were performed to find a way to enhance the heat transfer from the cubes.
In this paper, the use of vortex generator mounted on the surface of the cube has been investigated to enhance the heat transfer. The vortex generator in the present work is a simple rib mounted on the top and the side faces of the cube. The Reynolds number of the flow is 13000, based on the incoming bulk velocity and height of the cube. The objective of the present work is to employ large - Eddy Simulations (LES) to investigate the influence of the vortex generators on the flow structures and local heat transfer coefficient.

2 – Physical Model and Boundary Conditions

The physical model was a single heated cube placed in the middle of rectangular channel, as shown in figure (1.a). The side long the of the cube 10mm. The dimension of the channel is (100×50×25) mm.
The (x) and (y) axes were taken in the streamwise and wall directions, respectively while the (z) direction denoted the spanwise direction.

Fig (1.b) shows the cube with vortex generator .The vortex generator (VG) are small devices primarily used to suppress the formation of the separation regions and to alter the aerodynamic coefficients by changing the flow structures around the bodies.
The influence of the flow separation is always negative for the aerodynamic coefficient and pressure drop. In case of heat transfer, flow separation might enhance or discard the heat transfer. The points where the flow separates and reattaches to the body are always associated with local increase of heat transfer coefficient due to the high turbulent intensity in the shear layers close to these points.
In order to enhance the heat transfer from the surface of the cubes, it is highly desired that the boundary layers on the surface of the cube are turbulent. The turbulent boundary layer implies high local mixing due to the small scale vortices and hence high convective heat transfer coefficient. On way to attain a turbulent boundary layer on the surface of the cube is to use vortex generators VG. The VG is very small devise attached to the surface of the cube in the places where boundary layer separation is expected. Another advantage of the VG is the formation of large scale vortices behind them.
The shape of the VG used in this work is a simple rib attached to the side and top faces of the cube at the middle of the cube. The thickness of the VG is 2mm and its height is 4mm.
The bulk velocity of the incoming flow was 3.8m /s yielding the value of the Reynolds number of 13000 based on the incoming bulk velocity and the height of the channel. The density of air was 1.16 kg/m^3 and its dynamic viscosity was 1.5×10^{-5} m^2/s. A constant mass flow rate of 0.5 kg / s was passed in the channel. The value of the Prandtl number of the air was 0.7. The rate of heat dissipated from the cube to the air is 50W. The bulk temperature for the incoming flow was 294K .The channel walls were insulated.



(a

(b

Figure (1) The physical model.(a) cube without VG, (b) cube with VG.

3 – Mathematical Model - Governing Equations

In this paper, the Large–Eddy Simulations (LES) was employed to solve for both the velocity and temperature fields. In LES, the large eddies are computed directly and the influence of the small-scale eddies on the large scale eddies are modeled.

The flow and temperature fields are governed by the continuity, nonsteady three dimensional Navier – Stokes and energy equations. The resulting filtered equations are:

$$\frac{\partial \overline{u_i}}{\partial xi} = 0 \tag{1}$$

$$\frac{\partial \overline{u_i}}{\partial t} + \frac{\partial}{\partial x_j}\left(\overline{u_i}\,\overline{u_j}\right) = -\frac{1}{\rho}\frac{\partial \overline{p}}{\partial x_i} + \vartheta \frac{j^2 \overline{u_i}}{\partial x_j\,\partial x_j} - \frac{\partial \tau_{ij}}{\partial x_j} \tag{2}$$

$$\frac{\partial \overline{T}}{\partial t} + \frac{\partial}{\partial x_j}\left(\overline{u_j}\,\overline{T_j}\right) = \frac{\vartheta}{pr}\frac{\partial^2 \overline{T}}{\partial x_j\,\partial x_j} - \frac{\partial h_j}{\partial x_j} \tag{3}$$

Where u_i, P and T are the resolved filtered velocity, pressure and temperature, respectively. Owing to the non–linear terms in the momentum and heat equations, the filtered- out small scale eddies feedback their effects on the large–scale motion through subgrid–scale stresses and heat fluxes. These influences appeared as extra terms in the filtered equation, i.e, τij and hj. The sub-grid scale (SGS) stresses: $\tau_{ij} = \overline{u_i\,u_j} - \overline{u_i}\,\overline{u_j}$ represent the influence of the unresolved scales, smaller than the filter size, on the resolved ones. The subgrid- heat fluxes : $hj = \overline{u_i T} - \overline{u_j T}$ represent the influence of the unresolved heat fluxes on the resolved ones. These subgrid stresses and heat fluxes are unknowns and must be modeled.

The standard Smagorinsky model was used in the present work to model the SGS stresses and the stresses and the heat fluxes. Smagrinsky [5] suggested that, since the smallest turbulent eddies are almost isotropic, we expect that Boussinesq hypothesis might provide a good description of the effects of the unresolved eddies on the resoled flow. In this model, the subgrid - scale stresses and heat fluxes are modeled as:

$$\tau_{ij} = -2\mu_{SGS}\,\overline{s_{ij}} + \frac{1}{3}\,\tau_{ij}\,\delta_{ij} = -\mu_{\,SGS}\left[\frac{\partial \overline{u_i}}{\partial x_i} + \frac{\partial \overline{u_j}}{\partial x_i}\right] + \frac{1}{3}\tau_{ij}\overline{\delta_{ij}} \tag{4}$$

and

$$h_{j = -\propto t \frac{\partial \overline{T}}{\partial x_j}} \tag{5}$$

The Sagorinsky - Lilly SGS model builds on Prandtls mixing length model and assumes that we can define a kinematic SGS viscosity, $\vartheta_{SGS} = \boldsymbol{\mu}_{SGS} / \rho$, which can be described in terms of one length scale and one velocity scale.

Since the size of the SGS eddies is determined by the details of the filtering function, the obvious choice for the length scale is the filter cutoff width Δ.The velocity scale is expressed as the product of the length scale and the average strain rate of the resolved flow, $\Delta x|\bar{S}|$ where $|\bar{S}| = \sqrt{2\overline{s_{ij}}\,\overline{s_{ij}}}$ and the local rate of strain of the resolved flow are $\overline{Sij} = \frac{1}{2}\left[\frac{\partial\overline{u_i}}{\partial x_i} + \frac{\partial\overline{u_j}}{\partial x_i}\right]$ (6)

The subgrid - scale viscosity, ϑ_{SGG}, is defined as:

$$\vartheta_{SGG} = (C_{SGS}\Delta)^2|\bar{S}| \tag{7}$$

Different values of C_{SGS} have been imposed so far [6] and [7]; however C_{SGS} =0.1 is used in the present work.
Similar to the subgrid - scale viscosity, the subgrid scale eddy diffusivity, $\propto t$, is defined as:

$$\propto t = \frac{1}{P_{rt}}\,(C_{SGS}\Delta)^2|\bar{S}| \tag{8}$$

Where P_{rt} is the subgrid - scale Prandtl number. The value of P_{rt} was taken as 0.6 in this [paper. The Smagorinsky model is widely used in the simulations of bluff body [8] and [9]. It used also in modeling of the unresolved fluxes in the simulations of heat transfer problem [7].

4 – Numerical Method

Simple structured grids are used in both cases: around smooth cube without VG and around the cube with VG. The shapes of these grids for the cube without and with the VG are shown in figure (2) .

The total number of nodes was 400,220 for the simulation over the cube without VG. Fifteen nodes, covered the height of the VG, were used to resolve the flow in the wake of the VG, while the computational grid for the case with VG increased to 445,800 nodes.
All the equations are discretized based on the finite volume method and a staggered Cartesian grid. In the present work, the code uses second - order accuracy.
Fully implicit second - order upwind schemes used in the discretization method guarantees the stability of the numerical method. For the time marching term in equations, the Adam - Bach forth second - order scheme was used.
The no - slip boundary conditions was used on all solid walls for all simulations. The free slip boundary condition was used for out flow, where, it was surmised that the length of the domain was long enough that the effect of the body are vanished and the fully developed condition was applicable. The symmetry boundary condition was used in the spanwise direction, which means the normal velocity is zero and the changes in other velocity components are zero.
The main body of the code was a method that should be used to solve coupled equations in the time step. The PISO algorithm was adopted in this work.
PISO may be seen as an extended SIMPIE algorithm with one extra corrector step. The PISO algorithm requires additional memory storage due to the second pressure correction equation. It also needs under - relaxation to stabilize the calculation process. Although this method results in significant computational effort in

comparison with SIMPLE algorithm it has been found that the method is fast and efficient.

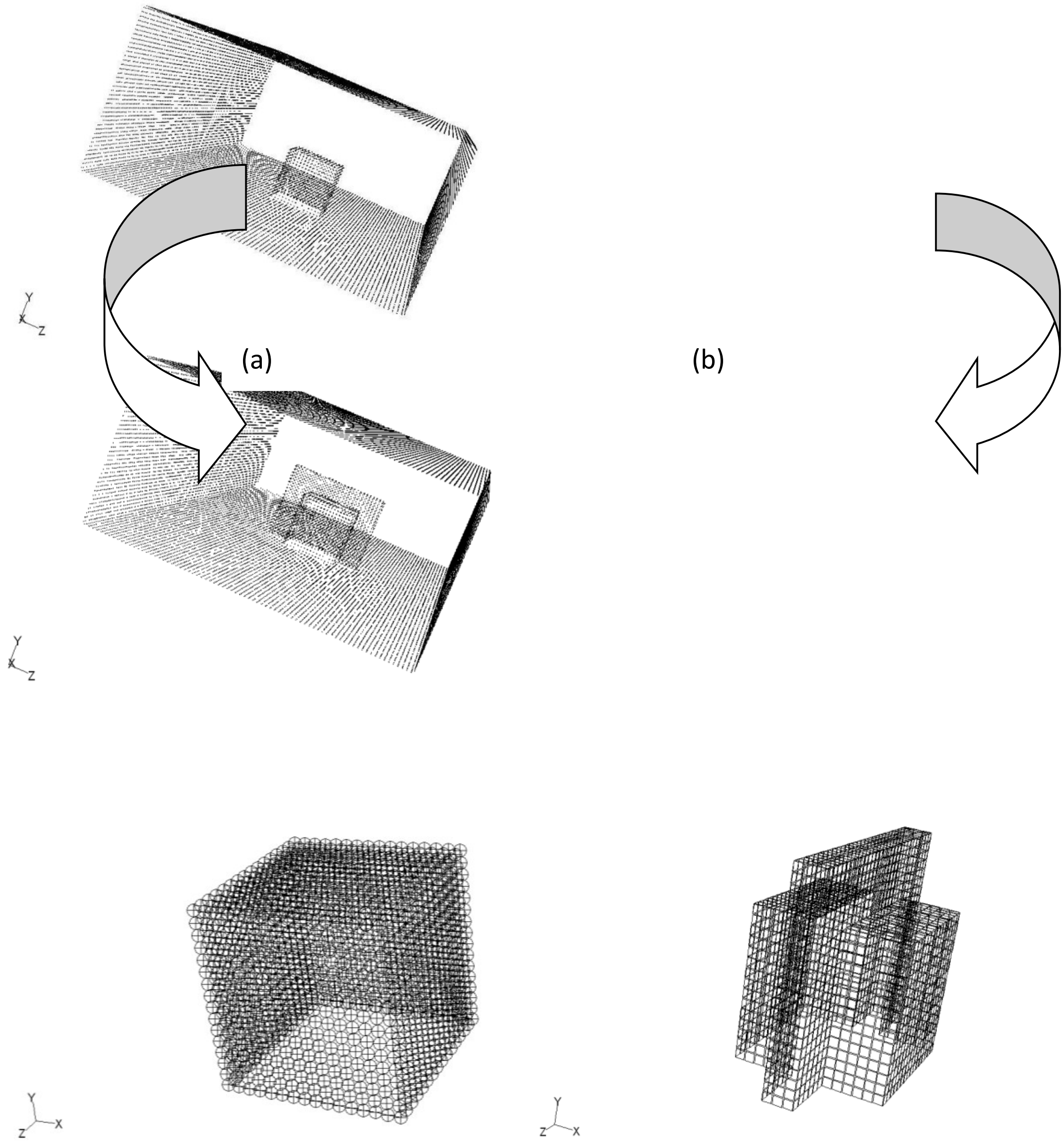


Figure (2) grid topology around cube.(a) without VG, (b)with VG.

5 – Results

In this work, the influence of the VG on the time – averaged flow**,** the instantaneous flow and the heat transfer coefficient is discussed **.** The simulation was performed using a constant time step of 0**.**001s. The simulation started with a uniform stream wise velocity equal to the bulk velocity. The other two components of the velocity are sat to zero value. The momentum equation was solved first while the energy equation is turned off. After the solution was converged and a fully developed flow was obtained, the energy equation was activated while the velocity field was frozen. When converged solution was obtained for the temperature obtained using 15,000 time steps. Figure (3) illustrated the streamwise velocity distribution on the vertical xy – plane at z=0.25m at four different locations in the stream wise direction at x=(0.2,0.4,0.6,0.8)m . Due to the differences of inflow conditions, the profile of the simulation results show a good agreement with the experimental data of [6] , meaning that the mesh resolution is fine enough to resolve the flow and hence the heat transfer .

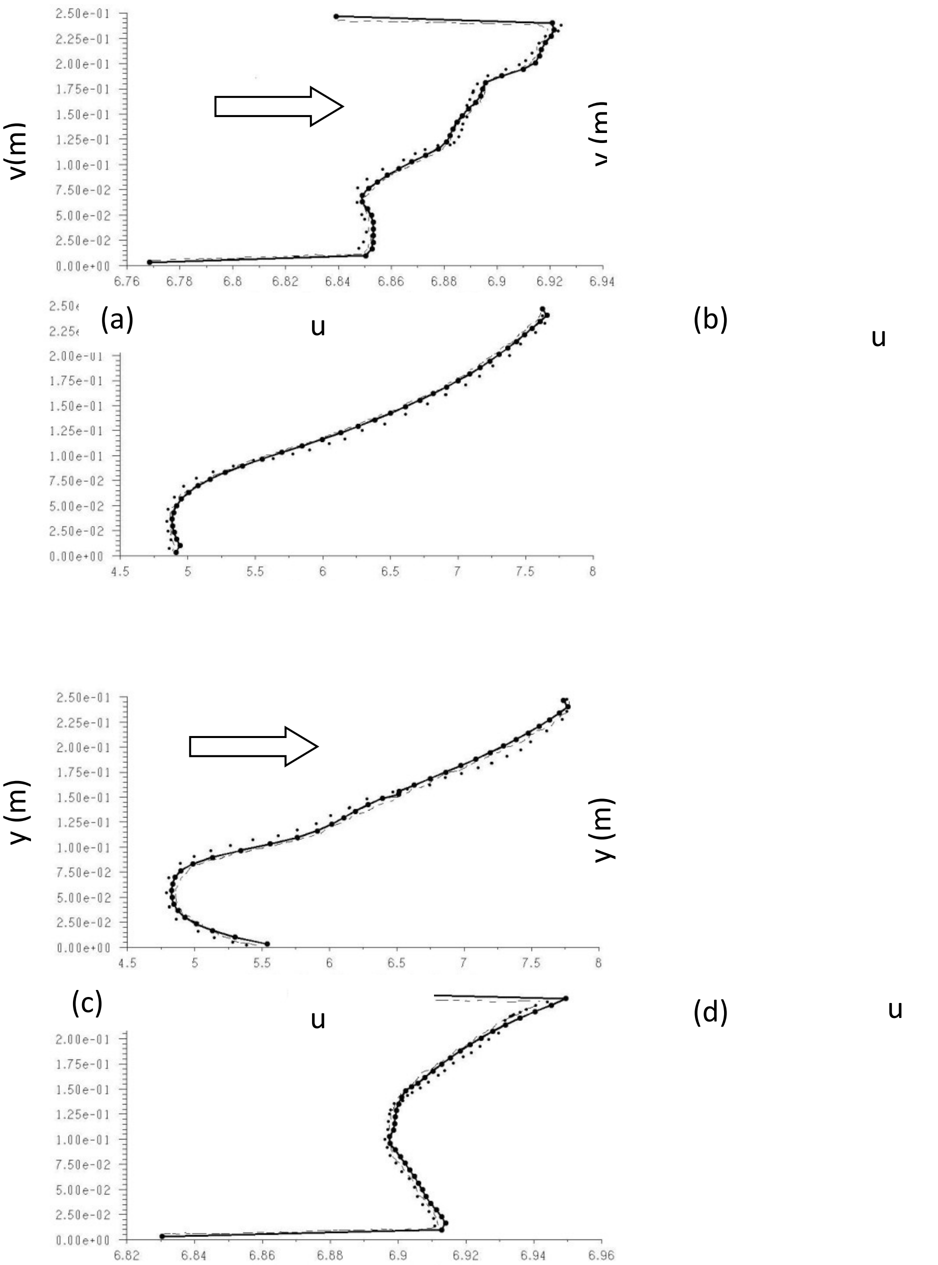
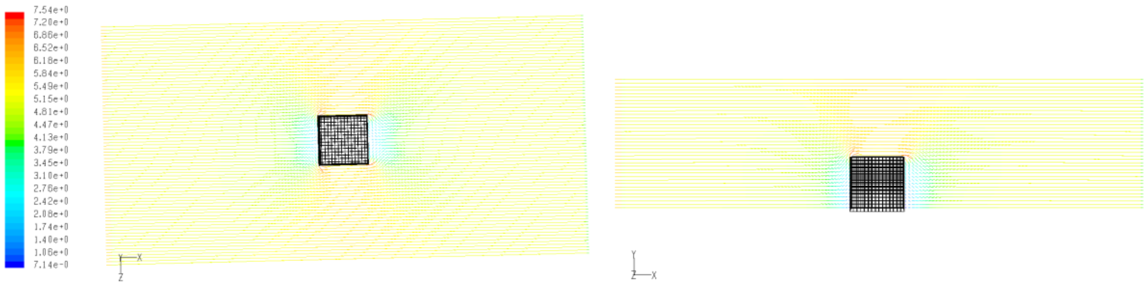


Figure (3) Streamwise velocity profile in vertical xy-plane at z=0.25m. (a)x=0.2m, (b)x=0.4m, (c)x=0.6m,(d)x=0.8m. Solid line (LES result around smooth cube); dashed(LES result around cube with VG); symbols: experimental data[6].

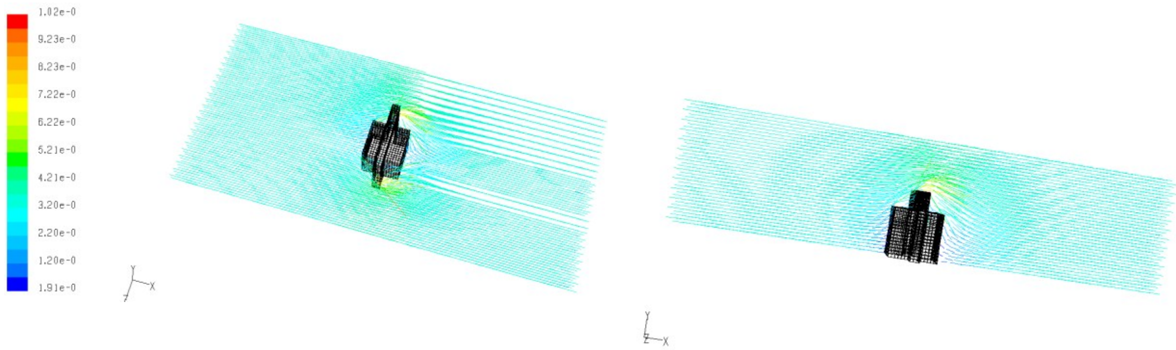
Figure (4) displays the time – a verged flow structures around the smooth cube by means of velocity vectors projected to xy - plane and xz - plane at z=0.25m and y=0.05m respectively. Figure (4.a) shows a circulation bubble in the wake next to the top face of the cube. Figure (4.b) shows two recirculation regions in the wake of the cube. The coming flow from the sides of the cube separates from the surface and circulates on those recirculation bubbles **.**



(a) (b)

Figure (4)Time - a verged velocity vectors of the flow around the cube without VG.(a) the xz-plane at y=0.05m , (b) the xy-plane at z=0.25m.

Figure (5) shows the time –averaged stream lines projected to the vertical xy - plane at z=0**.**25m and xz - plane at y=0**.**05m in case of cube with VG **.**The only difference between Fig**.**4 and Fig**.**5 is the size of the separation bubble on the top and the lateral side faces.



(a) (b)

Figure (5) Time - a verged velocity vectors of the flow around the cube with VG.(a) the xz-plane at y=0.05m , (b)the xy-plane at z=0.25m.

Although there are separation bubbles in the time – averaged flow on the sides of the cubes with and without VG, the temporal evolution of these bubbles is completely different. In case of smooth cube without VG, the flow separates from the faces due to the adverse pressure gradient on the surface of the cube. On the other side , the bubbles that appear in the time – averaged flow on the top and side – faces of the cube with VG is mainly a result of the VG **.** In case of smooth cube**,** the separation regions are almost stationary **.**In case of recirculation region behind the VG **,** the boundary layer is characterized by relatively large scale vortices **.** The vortices are shed from the recirculation region behind the VG to the boundary layer of the cube.

Figure (6) shows the temperature boundary layer profiles around the cubes (left surface **,** top surface **,** right surface) at the vertical xy - plane at z=0**.**5m **.** High temperature is reported on the surface of the cube in the places where the flow separates or circulates. The time – averaged temperature distribution on the surfaces of the cube with VG is globally lower than that on the surface of the cube without VG.

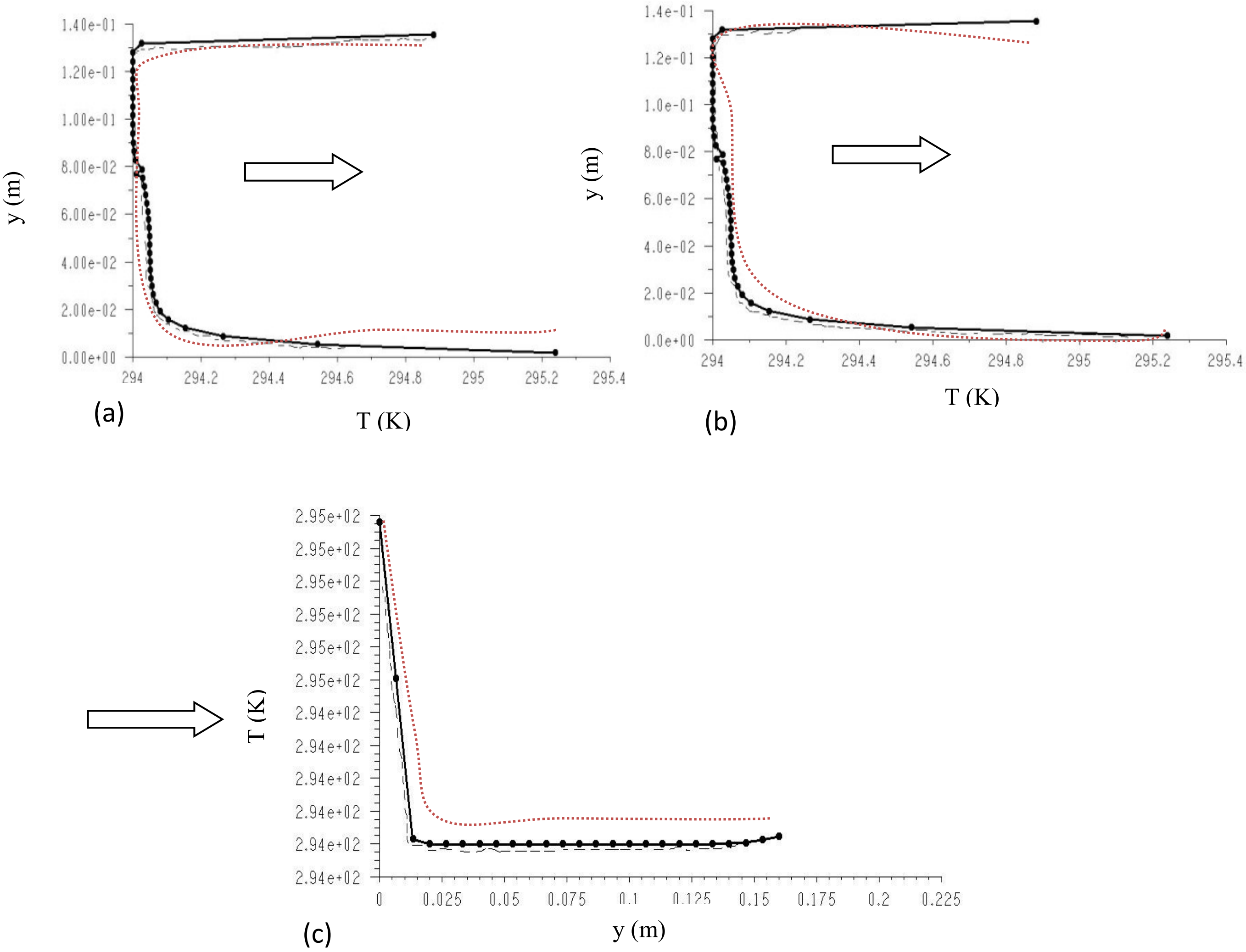


Figure (6) Temperature boundary layer profiles around the cubes at the vertical xy - plane at z=0**.**5m **.** (a) left surface **,**(b) right surface **,**(c) top surface. solid line with symbols (LES over smooth cube);dashed line (LES over cube with VG);dot line (experimental data [6]).

Figures (7) shows the contours of time averaged temperature distribution at xy - plane and xz- plane at z=0.25m and y=0.05m respectively with VG, while figure (8) show the same temperature counters but around cube without VG. The shedding of vortices from the VG alters the wake structures behind the cube and hence the temperature distribution, where a sharper temperature gradient is noticed in the wake of the smooth cube than that of the cube with VG.

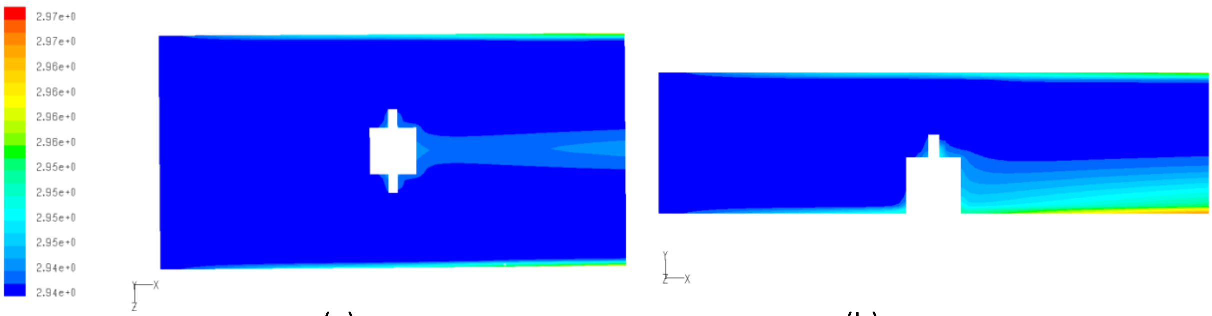


Figure (7) Contours of temperature distribution around the cube with VG.(a) the xz-plane at y=0.05m, (b) the xy-plane at z=0.25m.

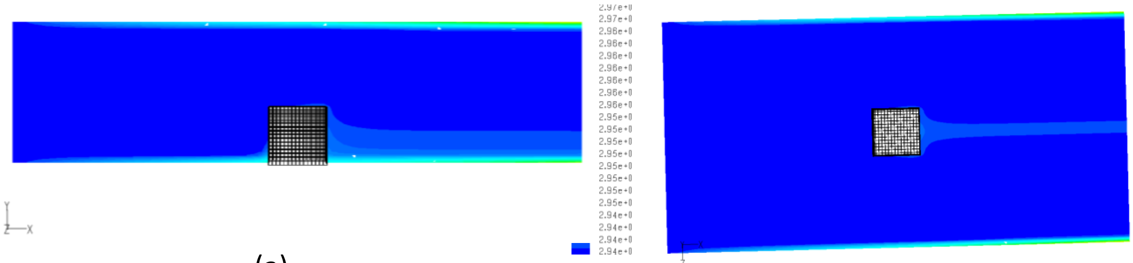


Figure (8) Contours of temperature distribution around the cube without VG.(a) the xz-plane at y=0.05m, (b) the xy-plane at z=0.25m.

The temperature of the surface of the cube and the inlet bulk temperature are used to calculate the local heat transfer coefficient (h) as follows:

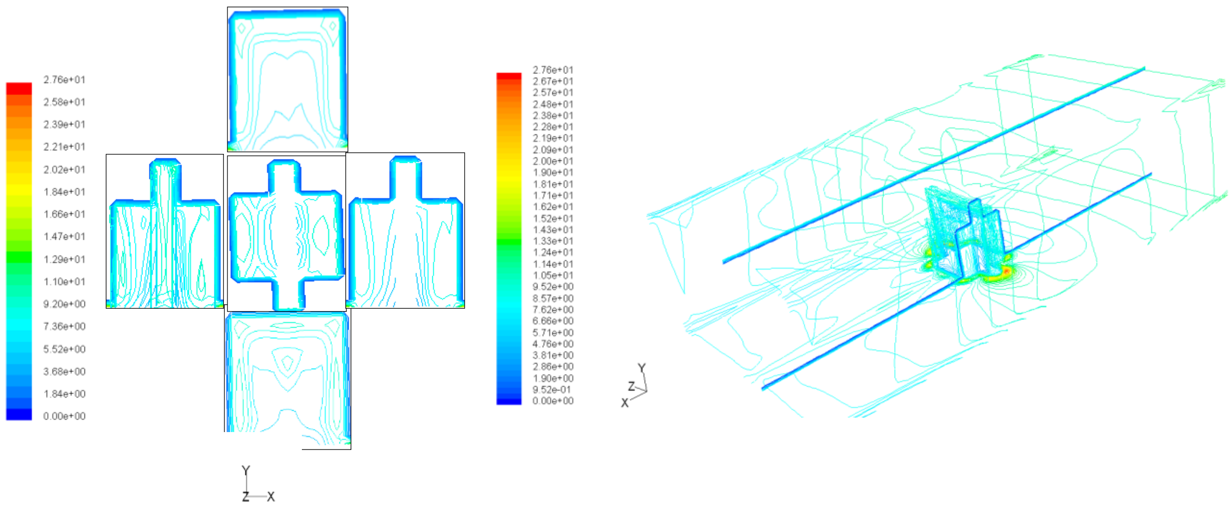
$$h = \frac{q^{\circ}}{(T_s - T_b)} \tag{9}$$

Here q° is the heat flux , T_s is the surface temperature and T_b is the inlet bulk temperature **.** This definition makes the heat transfer coefficient a direct measure for the surface temperature which is suitable for judging the performance of the VG since the main goal is the cooling of the surface.

Figure (9) shows the influence of the flow structures on the heat transfer on the surface of the cube with VG with the corresponding local heat transfer coefficient (h). Similarly **,** Figure (10) shows the flow pattern on the surface of smooth cube and the corresponding local heat transfer coefficient.

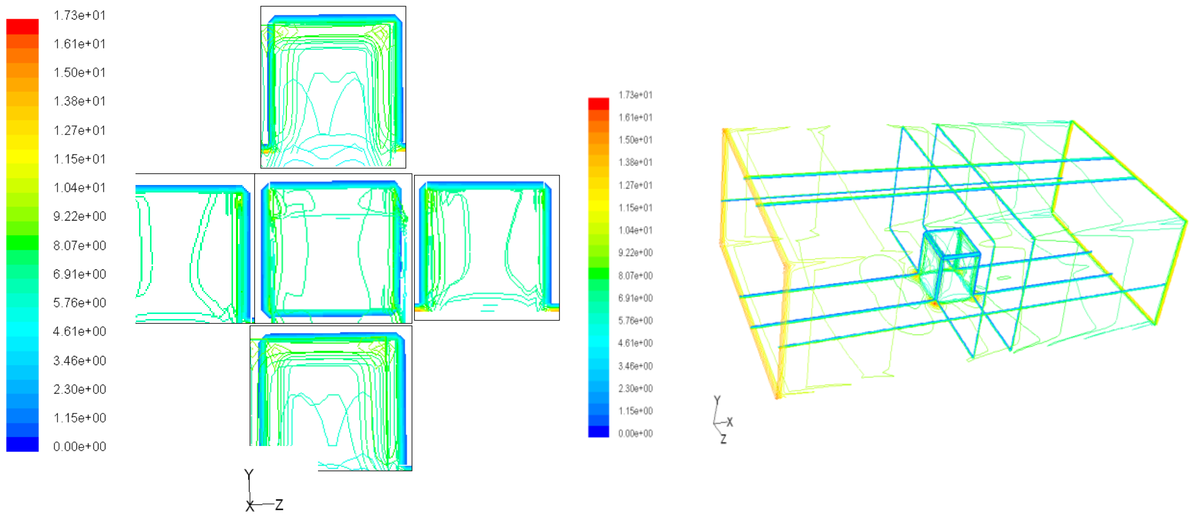
As expected, the temperature of the surface is high in the places associated with a flow separation. This is shown as low heat transfer coefficient on the side and top - side faces close to the leading edge and on the lee- side faces . On the streamwise face **,** the down wash flow due to the stagnation region on the stream wise face carries the heat from the surface resulting in a high heat transfer coefficient **.**

As a result the global mean heat transfer coefficient is higher than the mean heat transfer coefficient of the cube without VG on all the faces of the cube.



(a) (b)

Figure (9) Contours of heat transfer coefficient (h) with VG. (a) for the five faces of the cube,(b) for all faces of the cube.



(b)

(a)

Figure (10) Contours of heat transfer coefficient (h) without VG.(a) for the five faces of the cube,(b) for all faces of the cube.

Table (1) shows that front side can have large enhancement on the mean heat transfer coefficient . There is an increase of the mean heat transfer coefficient of 32% on front face while its found 20% on the lateral-side faces.
The minimum enhancement of the heat transfer coefficient is found on the top-side face , the increases is 17% . Globally there is an enhancement of the heat transfer coefficient of about 23.3% when VG is used.

Table1. Mean heat transfer coefficient at each face of the cube .

Cube face	Cube without VG $h_m(W/m^2K)$	Cube with VG $h_m(W/m^2K)$	Percentage increase
Front	11.5	15.2	32%
Top	10.4	12.16	17%
Lee – side	4.61	5.71	24%
Lateral – side	6.91	8.29	20%
Cube average	8.35	10.4	23.3%

6-Conclusion

In this paper, we have employed Large-eddy simulation to explore the fluid flow and the influence of a rib-shape vortex generator, on the surface of heated cube mounted in the middle of a channel, on the local heat transfer coefficient. LES results for flow and temperature distribution were obtained for a surface mounted cube with and without vortex generator. The relation between the flow structure around the cube and the temperature distribution is explained in the paper. The streamwise face of the cube was found to have high heat transfer coefficient due to the downwash flow that, efficiently, carries the heat from the face. Lower local heat transfer coefficient is reported on the places where flow separation occurs such as on the topside face and the lee-side face of the cube . It was found that the VG on the top and lateral faces of the cube altered the boundary layer on the surface of the cube. As a result, vortices are shed from the wake of the VG at a frequency higher than the dominant vortex shedding in the flow.

Globally these vortices enhanced mixing of heat in the boundary layer which appeared as high mean heat transfer coefficient. The present investigation showed that the percentage increase of the heat-transfer coefficient is considerable on the front and lateral-side faces of the cube. Local high temperature spots are found in the circulation region behind the VG. There was an increase of the mean heat-transfer coefficient of about 32% in the front face while 20% increase was found in the lateral-side faces. The minimum enhancement was found on the top-side faces about 17 % increase. Globally, there was an enhancement of the heat-transfer coefficient of about 23.3% due to the attachment of the vortex generator on the surface of the cube.

REFERENCES :

[1] S. Krajnovic´, L. Davidson. "Flow Around a Three – Dimensional Bluff Body" , 9th International Symposium on Flow Visualization, Heriot–WattUniversity, Edinburgh, G.M. Carlomagno and I. Grant(Eds.), 2000.

[2] S. Krajnovic´, L. Davidson. "Large – Eddy Simulation of the Flow Around a Bluff Body", AIAA Journal, Vol. 40(5), pp.927 – 936,2002 .

[3] A. Yakhot, H. Liu, and N. Nikitin, "Turbulent flow around a wall – mounted cube: A direct mumberical simulation" , International Journal of Heat and Fluid Flow (in press), 2006 .

[4] R. Martinuzzi and C. Tropea. "The flow around surface – mounted prismatic obstacles placed in a fully developed channel flow" ASME: Journal of fluids Engineering, Vol, .115,pp. 85–91,1993.

[5] Smagorinsky J, "General Circulation Experiments with the Primitive Equations I", The Basic Experiment, Mon. Weather Rev., Vol. 91,No.(2003),pp. 99 – 164 .

[6] E. Meinders, and K. Hanjalic´," Experimental study of heat transfer from in – line staggered configurations of two wall – mounted cubes", International Journal of Heat and mass transfer, Vol,. 45(3)(2002), pp. 465 – 482 .

[7] D. Barhaghi and L. Davidson and R. Karlsson Large Eddy Simulation of Natural Convection Boundary Layer on a Vertical Cylinder", Selected paper, International Journal of Heat and Fluid Flow, to appear,2006 .

[8] H. Hemida, S. Krajnovic´ and L. Davidson, "Large – Eddy Simulations of the Flow Around a Simplified High Speed Train Under the Influence of a Cross - Wind", 17th AIAA Computational Fluid Dynamics Conference, Toronto, Ontario, Canada 6 – 9 June, 2005 .
[9] S. Krajnovic´, L. Davidson. "Flow Around a Simplified car, Part (1): Large – Eddy Simulation", Journal of Fluid Engineering, Vol. 127(2005),pp. 907 – 918 .

Nomenclature

h heat transfer coefficient [W/m^2.K]

p pressure [Pa]

Pr Prandtl number

q° heat flux [W/m^2]

T temperature [K]

VG vortex generator

Subscripts

b bulk

m mean

s surface