World Academy of Science, Engineering and Technology
International Journal of Mechanical and Mechatronics Engineering
Vol:4, No:12, 2010

Investigation of Increasing the Heat Transfer from Flat Surfaces Using Boundary Layer Excitation

M.H.Ghaffari

Abstract**—**The present study is concerned with effect of exciting boundary layer on increase in heat transfer from flat surfaces. As any increase in heat transfer between a fluid inside a face and another one outside of it can cause an increase in some equipment's efficiency, so at this present we have tried to increase the wall's heat transfer coefficient by exciting the fluid boundary layer. By a collision between flow and the placed block at the fluid way, the flow pattern and the boundary layer stability will change. The flow way inside the channel is simulated as a 2&3-dimensional channel by Gambit^{TM} software.

With studying the achieved results by this simulation for the flow way inside the channel with a block coordinating with Fluent^{TM} software, it's determined that the figure and dimensions of the exciter are too important for exciting the boundary layer so that any increase in block dimensions in vertical side against the flow and any reduction in its dimensions at the flow side can increase the average heat transfer coefficient from flat surface and increase the flow pressure loss. Using 2&3-dimensional analysis on exciting the flow at the flow way inside a channel by cylindrical block at the same time with the external flow, we came to this conclusion that the heat flux transferred from the surface, is increased considerably in terms of the condition without excitation. Also, the k-e turbulence model is used.

Keywords**—**Cooling, Heat transfer, Turbulence, Exciting boundary layer.

I. INTRODUCTION

INCREASING and decreasing the heat transfer rate from a solid surface, is one of an important problems for the technical scientists in energy field most of the time. Today, at the most industrial applications such as electronic cooling components, cooling the plasma reactor shell, gas turbine components cooling such as combustion chamber and the other ones, we need to increase the heat transfer coefficient by the different methods because of space limitation and cost.

All of the different kinds of processes and equipments need to get energy to do their best; but as always it's impossible to convert all of the entrance energy to the useful work, some parts of this energy is lost in type of heat. So to reuse this heat loss, the different kinds of systems are developed and widely used in industrials which some equipment such as different types of heat exchangers are used to recover the heat and optimize the energy consumption. The efficiency of these equipments is so important to minimize the energy consumption from industries. In addition, a higher efficiency can cause a reduction in equipment sizes, material usage and manufacturing costs. Happening a turbulence inside these equipments to increase their heat transfer coefficient, is one of the best methods to increase their efficiency [1]. Recently a large amount of researches are done to improve the heat transfer by this method which are established as some papers[2]. One of the newest methods to change the flow regime, is using the boundary layer exciting by creating some blocks in different dimensions and sizes inside the ways, channels or tubes.

Drust and Becker[3], inserted a rectangular block on a flat surface inside a laminar boundary layer and by measuring the flow turbulence intensity, the transfer zone length and the area dimensions, showed that even with a laminar boundary layer, inserting a rectangular block can create a turbulency with a sensible intensity. These results combining with the results by the other researchers have the main rules to develop the electronic components cooling using these kinds of technologies [4].

Also Kahrom et. al. [5,6], studied the boundary layer exciting for the different blocks by creating them on and close to a hot surface with a hole on it at the flow on horizontal and vertical flat surfaces. Beredberg [6] at his doctorate thesis, at first assumed the turbine fines internal cooling way as a flat surface and then he studied the increase in cooling air flow turbulence effect on the surface average heat transfer coefficient rate using some general turbulence modeling methods. He assumed that the cooling flow rotating, a block created with sharp edges and roughening the surface are some factors to increase the flow turbulency and therefore he studied these factor's effects on surface heat transfer coefficient rate.

For a forced fluid flow on a surface, we can use the flow velocity, fluid viscosity and the contact area between the flow and solid faces, as some effective factors on heat transfer coefficient. Through these factors, the fluid velocity can be an effective factor to reduce the heat transfer coefficient because of the reduction in fluid velocity beside the solid surfaces and configuring the boundary layer. Therefore, the heat transfer mechanism at the layer coherent to the wall would be a conductional type heat transfer which can cause an inclement reduction in heat transfer. This layer can have a resistant layer rule against the forced heat transfer coefficient. We can increase the heat transfer coefficient by changing flow pattern inside the boundary layer and changing the velocity field

M.H.Ghaffari is Bsc in Mechanical Engineering and Msc Student of Energy Egineering, Sarkhoon and Qeshm Gas Treating Co, Bandarabbass, Iran (e-mail: hos.proeng@gmail.com).

change. There are some different types of researched and examined methods with obtained different results called the boundary layer exciting. The main parts of the boundary layer excitation is applying the fluidic phenomena such as stagnation point, vortex, the boundary layer discrete and the fluid jet creating inside the boundary layer.

Using these phenomena, the flow pattern inside the boundary layer and its thickness will change. Increasing external blocks inside the boundary layer or creating some convexities or downthrows on the surface are some general methods to have this effect [7]. At this research, to study the exciting effect for a cooling fluid flow inside a surface to increase the wall conventional heat transfer coefficient due to block placement, the cooling fluid way is simulated as a cylindrical channel in 20x1cm dimensions and then with creating this block with different dimensions and laying in this way, one can study the wall temperature change rates, the conventional heat transfer coefficient and the cooling air pressure loss due to boundary layer excitation by block placement. Then, with using the different types of geometrics, one can study and compare the block's placement effect in different types of geometrics, dimensions and laying on heat transfer coefficient values and the temperatures of the walls.

II. THE STUDIED MODEL SPECIFICATIONS AND THE GRIDDING METHOD

By placing a block in an air flow line, we can increase the heat transfer coefficient and decrease the surface temperature. At this time, to study the effect of exciting the cooling air flow to increase the heat transfer coefficient and to decrease the flat surface temperature using a block, at first we consider and simulate the flat surface as a 10x1cm rectangular channel and then to study the rate of wall temperature changes, the convection heat transfer coefficient of the cooling air flow and also the cooling air heat loss due to the boundary layer excitation because of the block, we will place a rectangular type block at the cooling air flow way with different dimensions that is shown in figure 1. After that, by using the different geometrics, we will study and compare the effect of block placement on both heat transfer coefficient and the wall temperature by using the different geometrics of the blocks such as circle, triangle and ellipse with the help of the governing equations. At the end, by determining a proper block, the boundary excitation effect in cooling of a channel in 3-dimensional condition is studied.

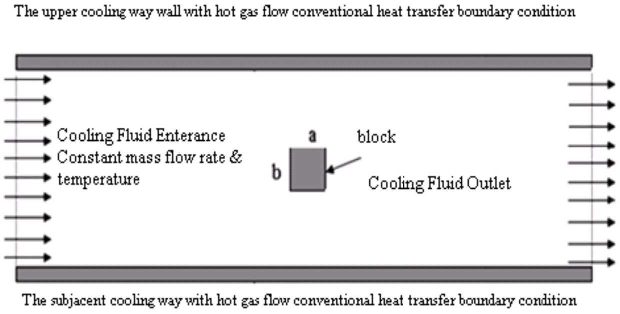


Fig. 1: The geometry used to study the rectangular block placement

The primary values for the hot gas and cooling air which are used to simulate the flat surface cooling, are mentioned on table 1.

TABLE I THE VALUES FOR THE BOUNDARY CONDITION PARAMETERS

The hot gas static temperature around the wall (in ^{o}K)	1000
The cooling air mass flow rate	0.5
The cooling air temperature (in ^{o}K)	500

A. Study of the gridding effect

Creating a proper gridding is one of the best aspects in numerical solution methods for equations with partial derivation. Minimizing the grid size is the main available method to improve the accuracy of the simulation. First of all, to achieve a general result from the solution, a primary simulation will be implemented on a large size grid. Then, the grid will be optimized during the solution circuit to forbid the differences between the obtained results to get over from the expectative tolerances. On the other hand, another most important parameter to select a good grid, is increasing the calculations time due to the grid density increasing. After determining a proper grid(depend on the calculation's accuracy and the proper solution time), the remained calculations will be done using this grid. At this paper to study the grid effects on results and selecting a proper grid, the temperature changes and the average conventional heat transfer coefficient are studied for the different cell numbers. To achieve this goal, at first, we created a grid include of 10744 cells which is illustrated in figure 2.

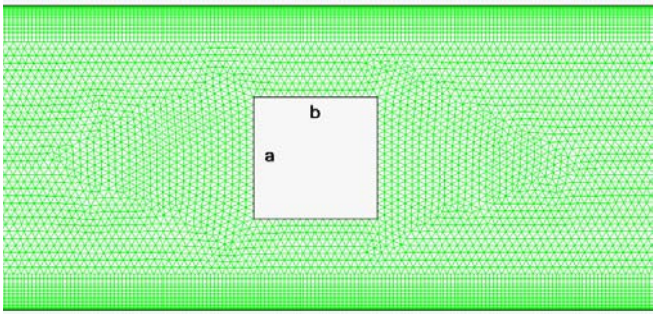


Fig. 2: The cooling way gridding using rectangular exciting

III. GOVERNING EQUATION

The average gas phase equations are as follows:

-Continuity:

— — — (1)

-Momentum:

$$\left[\frac{\partial}{\partial x}(\rho uu)+\frac{\partial}{\partial y}(\rho uv)\right]=-\frac{\partial p}{\partial x}+\mu\nabla^{2}u \\ -\rho\overline{u'u'}-\frac{\partial}{\partial y}(\rho\overline{u'v'})-\frac{\partial}{\partial z}(\rho\overline{u'w'}) \tag{2}$$

$$\left[\frac{\partial}{\partial x}(\rho uv)+\frac{\partial}{\partial y}(\rho vv)\right]=-\frac{\partial p}{\partial y}+\mu\nabla^2 v \qquad (3)$$

$$-\rho\overline{v'v'}-\frac{\partial}{\partial x}(\rho\overline{u'v'})--\frac{\partial}{\partial z}\rho\overline{v'w'}$$

$$\left[\frac{\partial}{\partial x}(\rho uw)+\frac{\partial}{\partial z}(\rho vw)\right]=-\frac{\partial p}{\partial z}+\mu\nabla^2 w - \qquad (4)$$

$$\frac{\partial}{\partial x}(\rho\overline{u'w'})-\frac{\partial}{\partial y}(\rho\overline{v'w'})-\rho\overline{w'w'}$$

The simplest "complete models" of turbulence are two-equation models in which the solution of two separate transport equations allows the turbulent velocity and length scales to be independently determined. The standard $k-\varepsilon$ model falls within this class of turbulence model and has become the workhorse of practical engineering flow calculations in the time since it was proposed by Launder and Spalding [8]. Robustness, economy, and reasonable accuracy for a wide range of turbulent flows explain its popularity in industrial flow and heat transfer simulations. It is a semi-empirical model, and the derivation of the model equations relies on phenomenological considerations and empiricism. The standard $k-\varepsilon$ model [9] is a semi-empirical model based on model transport equations for the turbulence kinetic energy k and its dissipation rate (ε). The model transport equation for k is derived from the exact equation, while the model transport equation for ε was obtained using physical reasoning and bears little resemblance to its mathematically exact counterpart. In the derivation of the $k-\varepsilon$ model, the assumption is that the flow is fully turbulent, and the effects of molecular viscosity are negligible. The standard $k-\varepsilon$ model is therefore valid only for fully turbulent flows. The turbulence kinetic energy, k, and its rate of dissipation, ε, are obtained from the following transport equations[10]:

$$\frac{\partial(\rho k)}{\partial t}+div\,(\rho kU)=div\left[\mu+\frac{\mu_t}{\sigma_k}gradk\right]+ \qquad (5)$$

$$G_k+G_b-\rho\varepsilon-Y_M+S_K$$

$$\frac{\partial(\rho\varepsilon)}{\partial t}+div\,(\rho\varepsilon U)=div\left[(\mu+\frac{\mu_t}{\sigma_\varepsilon})grad\,\varepsilon\right] \qquad (6)$$

$$+C_{1\varepsilon}\frac{\varepsilon}{K}(G_K+C_{3\varepsilon}G_b)-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k}+S_\varepsilon$$

In these equations, G_k represents the generation of turbulence kinetic energy due to the mean velocity gradients. G_b is the generation of turbulence kinetic energy due to buoyancy. Y_M represents the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate. $C_{1\varepsilon}, C_{2\varepsilon}$, and $C_{3\varepsilon}$ are constants. σ_k and σ_ε are the turbulent Prandtl numbers for k and ε, respectively. S_k and S_ε are user-defined source terms. The turbulent (or eddy) viscosity, μ_t, is computed by combining k and ε as follows[9]:

$$\mu_t=\rho C_\mu\frac{k^2}{\varepsilon} \qquad (7)$$

Where C_μ is constant.

The model constants $C_{1\varepsilon}, C_{2\varepsilon}, C_\mu, \sigma_k$ and σ_ε have the following default values [8]:

$$\sigma_{1\varepsilon}=1.44\ \sigma_{2\varepsilon}=1.92\ \sigma_k=1\ C_\mu=0.09\ \sigma_\varepsilon=1.3$$

IV. FLOW SIMULATION

The geometry of the air cooling way and the structured grid is generated using Gambit™ 2.3.16 software. Different grid sizes have been tested and Grid independency has been verified. This geometry offers the best compromise between precision and computational effort. When out flow converged, for increasing efficiency, cells of cooling way are doubled. Simulations are performed with the commercial CFD software Fluent™ [10]. The resultant systems of discretised linear algebraic equations are solved by using the density-based explicit solver [10].The SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) [11] algorithm is used for the pressure-velocity coupling, whereas the power-law [11] scheme is used for the convection-diffusion formulations. Fluid flow rate at the inlet of cooling way is 0.5kg/s. In figure 2, the location of block in cooling way is shown.

V. RESULTS FOR 2-DIMENSIONAL ANALYSIS PART

The rate of changes for wall's average heat transfer coefficient is studied due to increasing the cell's number.

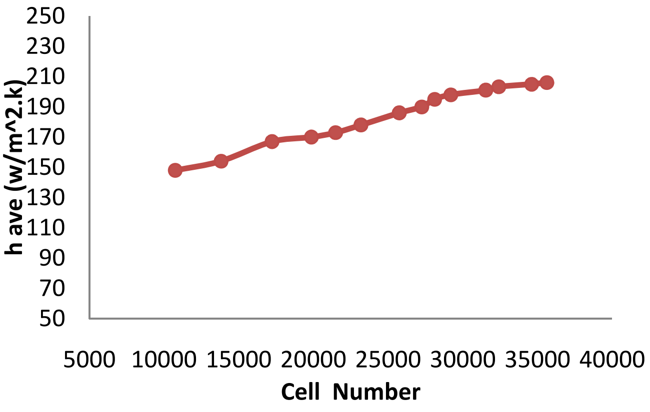


Fig. 3: Grid effects study on surface's average heat transfer coefficient effects

As shown in figure 3, with increasing the grid density to about 28000 cells, the heat transfer coefficient changes will be sensible. But, by increasing more, this grid density will tend to a constant value, not to many changes expected. So, at this research, a grid with 34600 cells is selected as a suitable grid because of its high accuracy equations and the proper solution time.

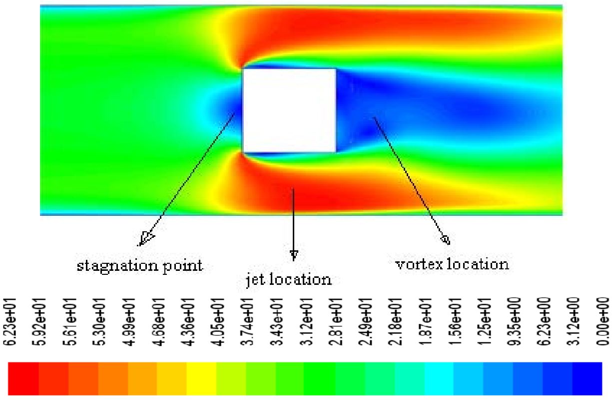


Fig. 4: The velocity vectors around the square block

In figure 4, we illustrated the effects of block placement on cooling air flow way which can cause a stagnation point at fluid-block collision situation, vortex creation at behind the block and a jet around the block close to the wall. The jet creation around the block can decrease the boundary layer thickness and increase the velocity of the flow beside the walls which they cause the boundary layer ablutions and increase the heat transfer coefficient. Also, it can create a great vortex behind the block that the figure and dimensions of this vortex, are depend on the geometry and the velocity of the flow.

Due to vortex constitution, the upper and subjacent fluid layers will directly stand beside the surfaces and the temperature gradient beside the wall and the side's heat transfer coefficient will increase. In continue, the fluid beside the wall will get warm step by step and the heat transfer coefficient will be decreased. In figure 5, a curve for the changes of conventional heat transfer coefficient between the wall and the cooling air flow due to the block placement as compared to the condition without exciting is illustrated.

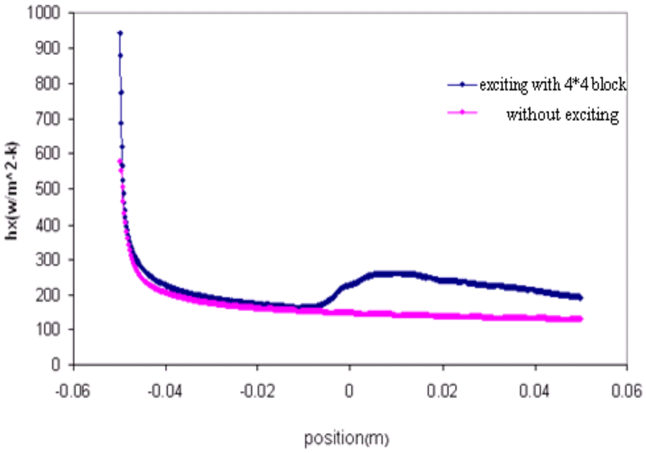


Fig. 5: the block placement effect on increasing the heat transfer coefficient

Due to increasing the conventional heat transfer coefficient, the heat flux between the cooling air and wall will increase which will cause the wall temperature reduction. The wall temperature changes curve due to a square block placement as compared to the condition without exciting, is shown in figure 6. It's important to know that another important effect of block placement, is cooling air pressure drop which its quantity is 6.7 times more than as compared to the condition without exciting.

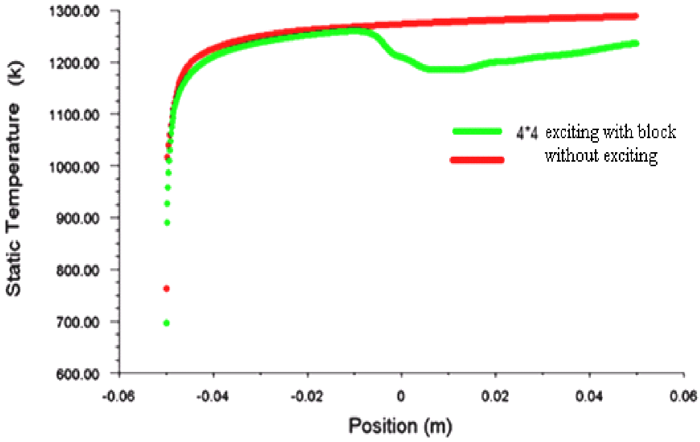


Fig. 6: The square block placement effect on local temperature decreasing in comparing to the condition without exciting

VI. USING RECTANGULAR BLOCK IN DIFFERENT DIMENSIONS

To study how much the cooling air exciting intensity depends on the block dimensions, at first assume that the block length is constant and then, with changing the block height, the increasing amount of block dimension at vertical position to the flow (a), will be compared to the first condition (constant height) at the amount of exciting rate. After that, by assuming the block height constant, the increasing amount of block dimension at the same side to the flow (b), will be compared to the previous condition (constant length) at the amount of exciting rate.

A. Using Rectangular Block with Constant Length and Variable Height

To study increasing block length effect in a constant height, consider 5 blocks in 4xb millimeters with block height changes rate from 1 to 5mm. in figure 7, the flow velocity distribution in cooling way with two blocks in 4x5 and 4x2mm are presented. With comparing this factor between these two channels, we can come to this conclusion that the jet velocity and length close to the walls will increase due to any increase in block height. One can see that the maximum jet flow velocity for a block with 5mm height is about 2 times more than a block with 2mm height which causing heat transfer coefficient increasing between wall and the cooling air flow.

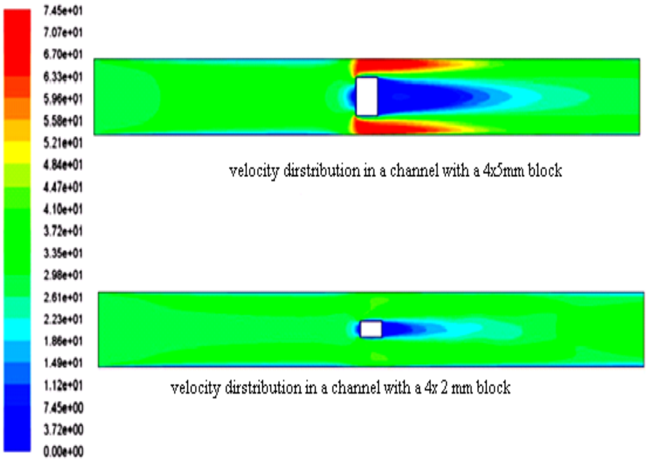


Fig. 7: The flow velocity distribution for the cooling way with 2blocks in 4x5 & 4x2mm dimensions

Also with comparing the turbulency distribution between these two channels, one can find that any increase in block height can cause an increase in exciting region length and the flow turbulence intensity. Because of this matter, the maximum turbulence intensity at the channel with larger block, will be 1.97 times more than one for the channel with smaller block. Turbulence intensity increasing will cause temperature gradient increase close to the walls too. In figure 8, the flow turbulence intensity distribution for the cooling way with two blocks in 4x5 and 4x2mm dimensions is shown.

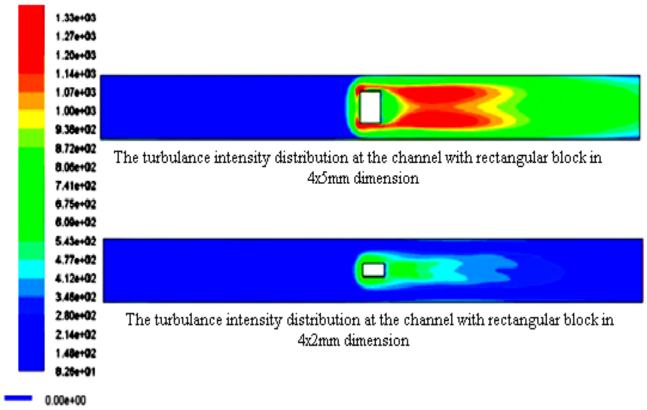


Fig. 8: The flow turbulence intensity distribution for the cooling way with 2blocks in 4x5 & 4x2mm dimensions

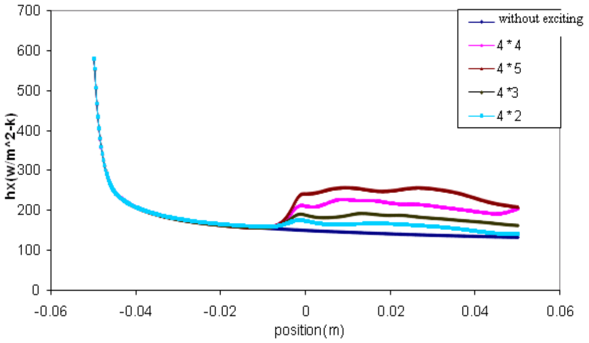


Fig. 9: Curve for the wall local conventional heat transfer coefficient changes in terms of the change in block height

In figure 9, the wall's local conventional heat transfer coefficient changes curve in terms of increase in block height is illustrated. Studying this curve can find that increase in block height with constant length can cause increase in wall heat transfer coefficient so that with a block in 4x5mm dimension, the wall local heat transfer coefficient at jet position, is 2.7 times more than the condition without exciting. Any increase in wall heat transfer coefficient will lead to a reduction in wall temperature so that a change in block height from 2 to 5mm, will reduce the wall average temperature from 800^{o}K to 537^{o}K. In table 2, the cooling air pressure drop quantity, wall average temperature and wall conventional average heat transfer coefficient in terms of increase in block height is presented.

TABLE II THE AIR PRESSURE DROP CHANGES, TEMPERATURE REDUCTION AND THE WALL HEAT TRANSFER INCREASING WITH INCREASE IN BLOCK HEIGHT

Block dimension (mm)	$\Delta P(pa)$	$\bar{T}_{wall}(o_k)$	$\bar{h}_{wall}({}^{W}/_{m^2k})$	$\Delta\bar{T}_{wall}$	$\Delta\bar{h}_{wall}$
Channel without exciting	162	800	164	0	0
1×4	270	794	167	6	3
2×4	465	773	176	21	12
3×4	660	735	185	38	21
4×4	1099	667	205	68	41
5×4	1986	537	250	130	86

$\Delta\bar{T}_{wall}$: the wall temperature difference as compared to the condition without exciting

$\Delta\bar{h}_{\text{wall}}$: the wall heat transfer coefficient difference as compared to the condition without exciting

With study the results from table 2, one can find that the effect of any increase in block dimension in vertical position of flow (block height), can increase the flow exciting intensity and the wall average heat transfer coefficient which can reduce the wall temperature. Also the other effect of block height increasing, is the cooling air flow pressure drop so that with block height increasing from 2 to 5mm, the flow pressure drop will be 4.26 times more than the previous condition. In figure 10, the wall average heat transfer coefficient changes and the flow pressure drop as compared to the condition without exciting in terms of increase in block height is illustrated as a curve.

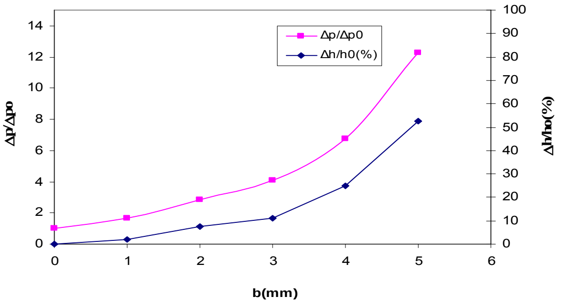


Fig. 10: The curve for wall average heat transfer coefficient changes and the flow pressure drop as compared to the condition without exciting in terms of increase in block height

B. Using the Rectangular Block with Variable Length and Constant Height

To study the block length increase effect in a constant height (increase in block side at the same side of the flow), we have considered 5 blocks in ax4mm dimension with the block length changes from 1 to 5mm. By comparing the flow velocity distribution, it has determined that the increase in block length had just a little effect on velocity and jet length close to the wall. But by comparing the turbulency between the two channels, one can find that an increase in block length, can decrease the excitation region length and decrease the turbulence intensity of the flow at the region behind the block so that the maximum turbulence intensity at the channel with the smaller block, is about 1.2 times more than that of for the greater channel. Also, an increase in turbulence intensity, can increase the temperature gradient beside the walls. Similar this, with flow turbulence intensity reduction because of increase in block length, the wall's heat transfer coefficient will be reduced and the wall's temperature will get increased.

To compare the effect of increase in block dimension in vertical and parallel position as compared to the flow, the wall's average heat transfer coefficient changes curve is obtained. Studying this curve can determine that the effect of increase in block height in increasing the surface average heat transfer coefficient is more than the effect of decrease in block length in increasing the surface average heat transfer coefficient and in increasing the flow pressure loss. Also, by an increase in rectangular block height and decreasing its length, the surface average heat transfer coefficient and the flow pressure loss will get increased and the walls will more cool. $\bar{h} \approx \frac{1}{a}$, $\bar{h} \approx b$

VII. SUMMATION FOR THE 2-DIMENSIONAL ANALYSIS PART

At this time, the cooling way exciting inside the channel in 2-dimensional condition using the rectangular, circular and triangle blocks in different dimensions is studied to achieve an increase in heat transfer from the channel surface. Also with studying the results from changes occurred in block dimensions in flow side and in vertical position as compared to the flow side, one can find that we have a reduction in exciting intensity due to an increase in blocks dimensions at flow side and we have increase in exciting intensity due to increasing in blocks dimensions at the vertical position as compared to the flow side so that for every reduction in thickness and any increase in height of the block, we would have more exciting intensity.

The exciting intensity and the cooling air flow pressure drop, are two important variables to select a proper block. To compare the different exciting intensity, different parameters such as channel's wall average temperature, cooling fluid temperature in outlet of the wall, and the channel's wall average heat transfer coefficient can use. In figure 11, the wall average heat transfer coefficient changes as compared to the condition without exciting for different blocks in terms of block dimension change in vertical position flow with constant block dimension assumption at flow side is illustrated. In this figure, one can find that with increase in block dimension, the wall average heat transfer coefficient, will increase specially on rectangular block which by an increase in block height to over 4mm, the heat transfer coefficient will suddenly increase.

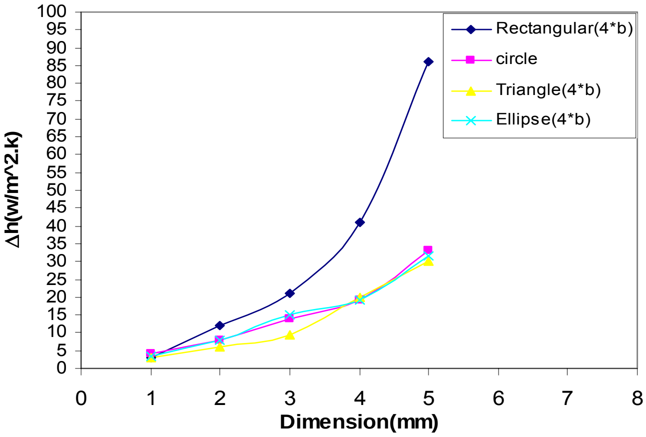


Fig. 11: The wall average heat transfer coefficient changes for different blocks in terms of block dimension increase in a vertical position against the flow side

Also in figure 11, it's shown that in similar dimensions, the rectangular block exciting is more than the other blocks so that an increase in wall average heat transfer coefficient, with rectangular exciting in 4x5mm dimension, is about 2.8 times more than the flow exciting with triangle block at the same dimensions.

According to the previous parts, increase in cooling air flow pressure drop is one of important effects of flow exciting intensity because of block placement, so that selecting a proper block is depends on the allowable pressure drop design and the importance of increase in heat transfer. Figure 12, is presented the wall heat transfer coefficient changes for the different blocks in terms of flow pressure drop. With studying this figure, one can find that at a similar pressure drop, the circular block exciting intensity (increase in wall heat transfer coefficient) is more than the other kinds of blocks.

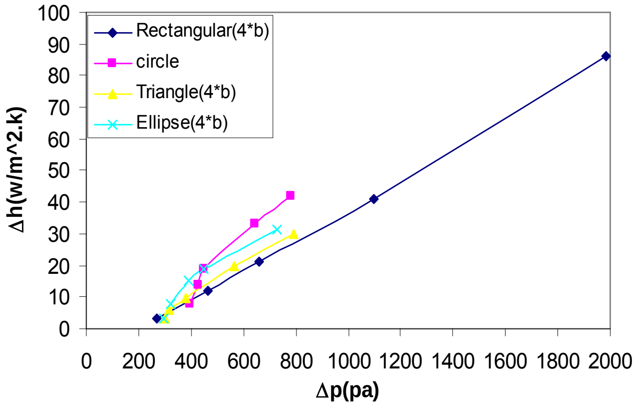


Fig. 12: The wall heat transfer coefficient changes of different blocks in terms of flow pressure drop

VIII. RESULTS FOR THE 3-DIMENSIONAL ANALYSIS PART

A. Analysis of flow excitation using cylindrical and cubic blocks

In previous parts, we mentioned the boundary layer excitation on flat surfaces as a channel in 2-dimensional analysis by some blocks in different geometrics. At this time, to have a better sense about the analysis, a 3-dimensional analysis has considered to study the excitation by some different types of blocks. To have a good view, a tube is assumed for exciting. The fluid, temperatures and the other specifications are the same as before. At first, we compare the flow way around a cylindrical block with a cubic block in 4mm dimension. In figure 13, the channel geometry, inlet, outlet and the block position are shown. By considering the gridding importance in numerical solution of equations, to analyze the effect of grid on results and to select a proper grid, the tube wall average heat transfer coefficient for different numbers of cells is studied. To achieve this goal, at first a grid with 76431 cells is created. After analysis, it has determined that by increasing the number of cells to about 101274, the average heat transfer from the tube surface is sensible. But by increasing more, this parameter tend to a constant value so, a grid with 113258 cells is selected because of high accuracy and proper solution time for the calculations.

Fig. 13: Geometry of the flow inside the tube and excitation using 4mm diameter cylindrical block

Fig. 14 is shown the flow lines path inside the tube. Because of vortex existence behind the block, the fluid entrance the way, doesn't fill the space behind the block. Figure 15, is shown the flow lines path and the vortexes created behind the cylindrical block. As it has shown at the upward figure, two vortexes are created at the cylinder-channel connecting location in addition to the vortex behind the block.

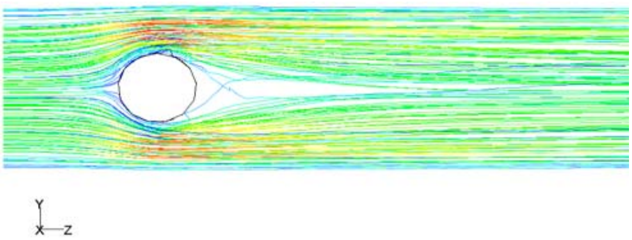


Fig. 14: Direction of cold fluid inside the tube with excitation by 4mm diameter cylindrical block

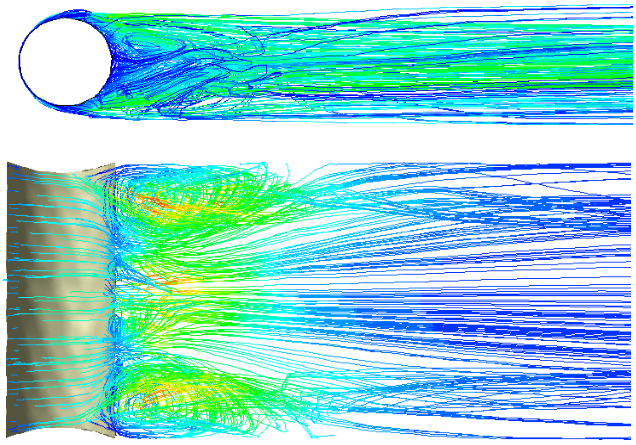


Fig. 15: Path lines of flow and vortex formed at behind the cylindrical block in two different view

Another compared block is a cubic block in 4x4mm dimension. In figure 16, the vortex and the flow motion path behind the block has shown. At this figure, it's determined that there are two vortexes on each other behind the block that have a rotation against each other (clockwise and counter clockwise). As it has shown, the created vortex behind the cubic block is greater than that of the cylindrical block and its dimension in block length is constant. Also, the two vortexes created at the wall-block connecting location for the cubic block is so smaller than that for the cylindrical block. (Figure 17)

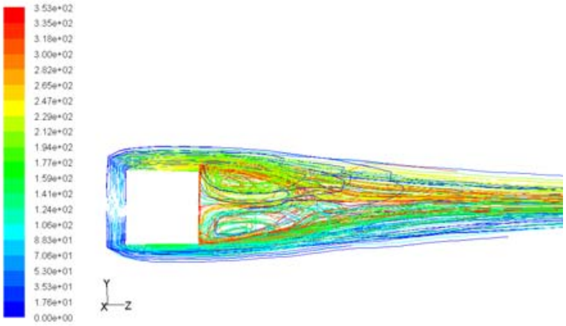


Fig. 16: Direction of flow behind the cubic block in two different views

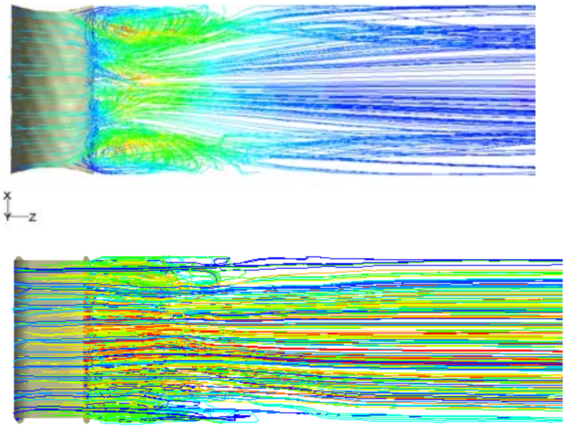
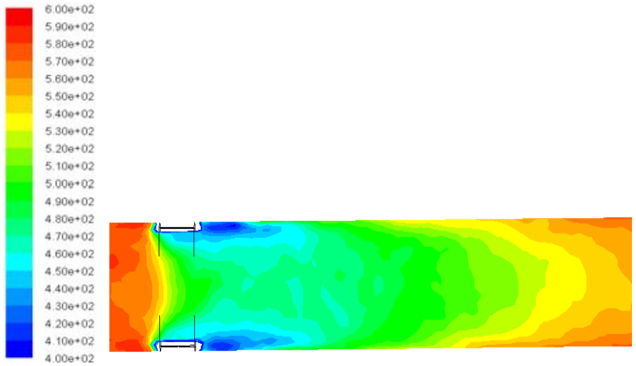
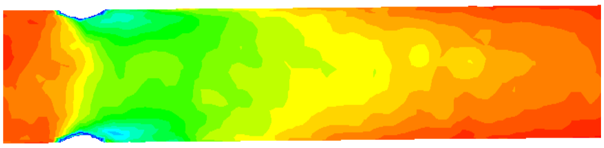


Fig. 17: compare side vortex behind the cylindrical block (up) with cubic block (down)

In figure 18, the temperature distribution and the block's excitation region length is compared for both cubic and cylindrical blocks. At exciting by cubic block, one can see that the excitation region length is larger because of the greater vortex and also we have more reduction in surface temperature as compared to the excitation by cylindrical block. It's found that the length and intensity of the created jet around the cubic block is more than one for cylindrical block so the exciting intensity because of the cubic block placement is too more than one for cylindrical block.



Cubic block



Cylindrical block

Fig. 18: compare temperature distribution and excitation region length behind the cubic and cylindrical block

In figure 19, the heat transfer from the tube wall to the fluid behind the cubic block has shown. One can find that the heat transfer rate is increasing behind the block.

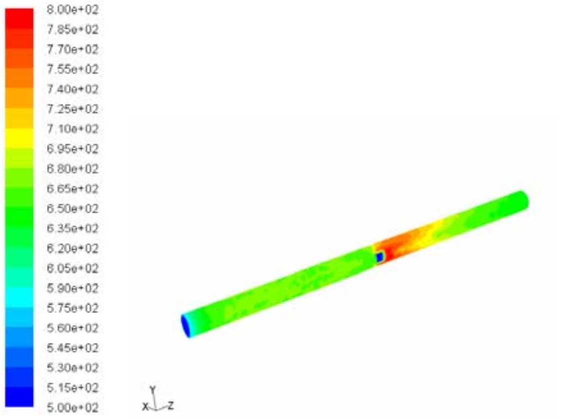


Fig. 19: Heat transfer distribution on the tube wall at excitation by 4x4mm cubic block

IX. CONCLUSION

At this research, the exciting boundary layer at first for 2-dimensional and then for 3-dimensional condition by placing difference type of blocks is studied. The way pressure loss and exciting intensity, are two important variables to select a proper block. To compare this intensity, one can assess different parameters such as surface average temperature, fluid temperature at outlet of the channel, the channel wall's average heat transfer coefficient and the heat transfer from the surfaces.

In figure 20, the changes for the wall's average heat transfer coefficient for the different composing in terms of block dimension is illustrated. It has shown that there's a suddenly changing at the curve inclination for increasing in wall's heat transfer coefficient because of placing a block in 3mm dimension. By considering figure 21, one can find that with increase in block dimension, the flow pressure loss will increase and the outlet flow average temperature changes would be decreased.

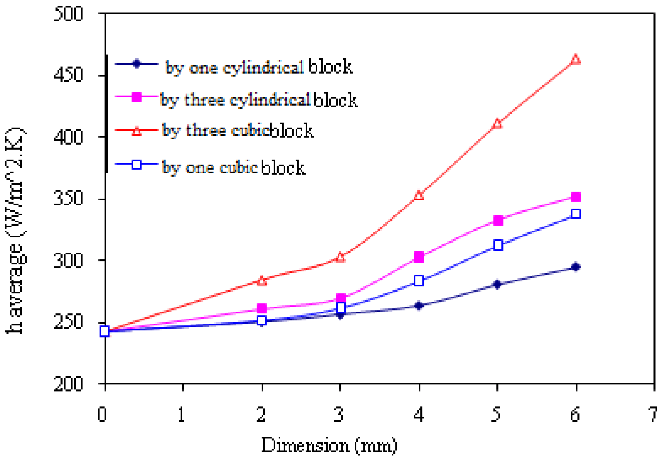


Fig. 20: compare tube wall's average heat transfer coefficient changes at excitation by cylindrical and cubic blocks in different dimension

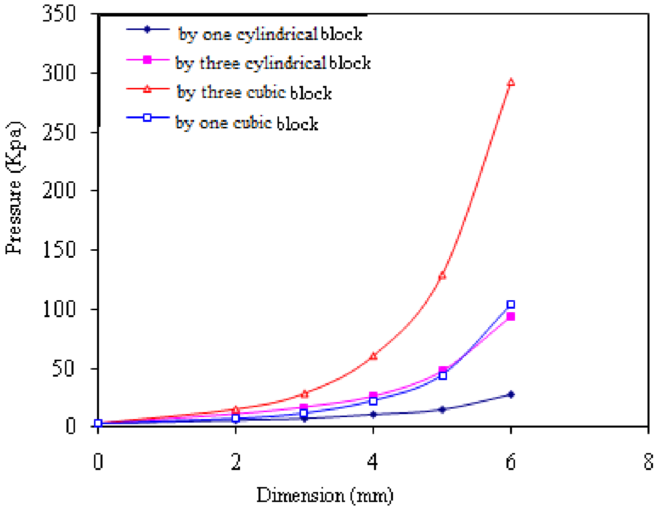


Fig. 21: compare flow pressure drop at excitation by cylindrical and cubic blocks in different dimension

Therefore by considering the results combining with the other ones, the main results obtained from solution are as follows:

1. The average heat transfer coefficient and the flow pressure drop, have considered as the evaluating criteria excitation intensity so that by increase the

heat transfer coefficient, the outlet fluid average temperature would be decreased. Therefore, one can use these two parameters to assess the exciting intensity.

2. By studying the flow excitation with different types of blocks, one can find that excitation by cubic block as compared to the cylindrical block has more effects to increase the heat transfer at the same dimension and in a line perpendicular to the flow and also it has more pressure drop.
3. Selecting the appropriate block is depends on the allowable design pressure drop and the importance of increase in heat transfer.
4. Considering all the parameters such as heat transfer, the temperatures, excitation intensity and the pressure drop, the cylindrical block is selected as the most suitable choice to have a good exciting intensity.

REFERENCES

[1] Jafari Nasr, M.R. and Polley, G.T., "An algorithm for cost comparison of optimized shell and tube heat exchanger with tube insert and plain tubes", *Chem. Eng. Tech.*, Vol.23, pp.267-272, 2000.

[2] Tijing, L.D., Pak, B.C., "A study on heat transfer enhancement using straight and twisted internal fin inserts", *International Communications in Heat and Mass Transfer*, xx (2006) xxx–xxx.

[3] Drust,.F., and Becker,S., FY, "Boundary layer transition included by a roughness element", *international report*, University Erlangen, Germany, INEEL LDRD Program, 1973.

[4] Chiriac, Y.A. and Ortega, A., "A numerical study of the unsteady flow and heat transfer in a transitional confined slot jot impinging on a isothermal surface", *Int., j., heat and mass transfer*. Vol.45, pp1237-1248, 2002.

[5] Kahrom M., Haghparast P., Javadi M., "Neural Network Application in Boundary Layer Phenomenon for Cooling the Turbine Blades to Improve the Cooling Process", *16th International Conference on Mechanical Engineering*, Iran, 2008.

[6] Beredberg, J., "Turbulence Modeling For Internal Cooling Gas – Turbine Blades", *PhD Thesis*, Chalmers University of Technology, Sweden, 2002.

[7] Ghobadi, A., Kahrom, M., "Cooling Turbine Blades using Exciting Boundary Layer", *International conference on computational and mathematical engineering*, Penang, Malaysia, 2010.

[8] B. E. Launder, D. B. Spalding. "Lectures in Mathematical Models of Turbulence", *Academic Press.*, London, England, 1972.

[9] T.B Gatski, M.Y.Hussaini, J.L. Lumley, "Simulation and Modeling of Turbulent flows", *Oxford University Press*, New York, 1996.

[10] "Fluent User's Guide", Version 4.3, vols, 1-4, *Fluent Incorporated, Lebanon*, NH, 1995.

[11] S. V. Patankar, "Numerical Heat Transfer and Fluid Flow", *McGraw-Hill*, 1980.