Numerical Simulation of the Impinging Jets and Study the Effect of the Existence of the Pedestal on Flow Field and Heat Transfer Characteristics

Maryam Moeinfar[1], Mohammad Moghiman[2], Morteza Abdolahian[3] Rasool Elahi[4]
Department of Mechanical and Aerospace Engineering, Ferdowsi University of Mashhad

Abstract
In this study the numerical simulation of three dimensional turbulent impinging round jets are conducted using the CFD method. The two jets one impinging on a simple flat plate and the other impinging on a circular pedestal heated and mounted on a flat plate are investigated. The flow fields and heat transfer characteristics of the two cases are compared at a Reynolds number of 23000 and nozzle to target distance of six times the diameter of the jet. The purpose of this paper is studying the effect of existence of the pedestal on rate of heat transfer. Turbulent fluctuations in the velocity field are modeled using the Reynolds Averaged Navier-Stokes (RANS) methodology. Turbulence is assumed to be isotropic. The buoyancy and radiation heat transfer effects are neglected and the flow is considered to be incompressible. The simulations are performed using various turbulence models such as the *Realizable* $k-\varepsilon$, $RNG\ k-\varepsilon$, $SST\ k-\omega$ and v^2-f. There is a good agreement between the convective heat transfer results of the v^2-f model and experimental data. The results show that existence of pedestal increases the heat transfer coefficients and causes a local minimum in the heat transfer coefficient profile on the symmetry axis, which is in contrast to the results of the simple flat plate.
Keywords: *Impinging jet - Numerical simulation - Circular pedestal - Heat transfer characteristics.*

Introduction

Impinging jets are widely used in many engineering and industrial problems. They have been taken into consideration as heating or cooling techniques, due to their high convective heat transfer rates. Some of their applications are annealing of metals, cooling in grinding processes, [1] cooling of gas turbine blades, [2-4] glass manufacturing, [5] cooling of microelectronic equipments, [6, 7] drying film and textile [8] and etc. The high rate of heat transfer generated by turbulent impinging jet flows is also used in aircraft industry. The flow field generated by the fan-powered vertical take-off and landing vehicles can be simulated by considering impinging jet flows [9].

The jets strike the target plate and its structure is broken down into three parts, the potential core, the shear layer, and the wall jet. Some of the key parameters determining the heat transfer characteristics of an impinging jet are the jet Reynolds number, Prandtl number, the nozzle to target distance, the geometry of the target plate, the flow conditions in the pipe, the nozzle geometry, the crossflow and etc.

Despite the simple geometry of the impinging jet, its flow characteristics are extremely complicated and this complexity has made it to be a challenging case for turbulence models and the investigation of such flow has attracted the attention of many researchers.

Since most experiments are performed at conditions suitable for accurate testing and not actual conditions, it is better to use the dimensionless parameters to scale the results. Impinging jets are often characterized by the jet Reynolds number, while the heat transfer is characterized by the Nusselt number defined as:

$$Nu = \frac{hd}{k} \tag{1}$$

where D, h and k represent the jet diameter, convective heat transfer coefficient and thermal conductivity respectively.

Numerous studies have been conducted in the literature investigating the influence of different parameters on the flow field and heat transfer characteristics of impinging jets. These studies include experiments, analytical solutions, and numerical simulations. For example, Mesbah [10] and Choi et. al [11] investigated the heat transfer characteristics of the jet on concave surfaces experimentally and concluded that the increase in the relative curvature increased the heat transfer of the stagnation point. Lee [8] studied the effect of the aspect ratio of elliptic turbulent jets on the jet structure and heat transfer of the stagnation region. Dano et. al [12] researched the effects of nozzle geometry on the flow characteristics and heat transfer performance. Kim and Giovannini[13] also investigated the turbulent round jet flow, impinging on a square cylinder laid on a flat plate experimentally.

Since the reported experimental results show significant differences in the rate of heat transfer, there have been many studies, trying to numerically predict the flow and heat transfer from the impingement of the jet. For example Behnia et. al [14] computed the heat transfer of a turbulent impinging jet numerically. The

1. M.Sc. Student of Aerospace Engineering, Tel: +989375371702, Email:Eng_m_moeinfar@yahoo.com.
2. Professor of Mechanical Engineering
3. M.Sc. Student of Aerospace Engineering
4 Graduate Student of Aerospace Engineering

unsteady simulation of a jet impinging on a flat plate was conducted by Cziesla et. al [15] using the large eddy simulation (LES) methodology and by Chung et. al [16] using direct numerical simulation (DNS). Merci et. al [17] studied the heat transfer of the turbulent jet impinging on a pedestal both experimentally and numerically. Rundstrom et. al [5] compared the performance of the $v^2 - f$ and Reynolds stress turbulence models with a two-layer wall treatment for the prediction of the mean velocity field, the turbulence characteristics and the heat transfer rate of the normal impinging jet and also impinging jet in a cross-flow configuration. Moeinfar et. al [18] investigated the effect of the nozzle geometry of the jet impinging on a circular pedestal on the rate of heat transfer numerically. They concluded that on the top of the pedestal the heat transfer of the round jet is higher compared to the rectangular and triangular jets with the same hydraulic diameters while on the flat plate rectangular jet has the maximum amount of heat transfer.

Apart from these experimental and numerical studies, Parameswaran and Alpay [19] presented an analytical model for the flow field on a normal impinging jet.

As much information of the effect of different parameters on impinging jets is available, the ways of increasing the jet effectiveness especially through modification of target plate is an important issue.

In the present study, the turbulent jets, one impinging on a simple heated flat plate and the other impinging on a circular pedestal with height (*h*) of D /1.06 and radius of *h*/2, heated and mounted on the plate, are investigate numerically using the CFD method. Herein all lengths are made dimensionless using the jet diameter (D). The fully developed condition is considered at the emergence of the pipe. The heat transfer characteristics of the two cases are compared to study the effect of the existence of the pedestal on the rate of heat transfer both qualitatively and quantitatively. Different turbulent models are used at Reynolds number of 23000, and dimensionless nozzle to target distance of six, and the predictive capabilities of these models are evaluated comparing the simulation results to the available experimental data [10]. A schematic of one of the studied computational domains is shown in Fig. 1.

Governing Equations

The full equations of motion governing Newtonian fluid flows are very complex. In the present work the flow field can be considered to be incompressible. The buoyancy and radiation heat transfer effects are neglected. The velocity and pressure are decomposed into the mean and fluctuating components. The continuity equation is:

$$\frac{\partial U_i}{\partial x_i} = 0 \tag{2}$$

where U_i is the velocity component. The momentum and energy equations are decoupled. Assuming the constant fluid properties these two equations are:

$$\frac{\partial U_i}{\partial t} + U_j \frac{\partial U_i}{\partial x_j} = -\frac{\partial P}{\partial x_i} + \nu \frac{\partial}{\partial x_j}\frac{\partial U_i}{\partial x_j} - \frac{\partial \overline{u_i' u_j'}}{\partial x_j} \tag{3}$$

$$\frac{\partial T}{\partial t} + U_j \frac{\partial T}{\partial x_j} = \alpha \frac{\partial}{\partial x_j}\frac{\partial T}{\partial x_j} - \frac{\partial \overline{u_j' T'}}{\partial x_j} \tag{4}$$

where P, T, ν and ρ are the pressure, temperature, kinematic viscosity and fluid density respectively.

$$\alpha = \frac{k}{\rho c_p} \tag{5}$$

k is the thermal conductivity of the fluid and c_p is the specific heat at constant pressure. In equation 3, $\overline{u_i' u_j'}$ is the kinematic form of the Reynolds stress tensor and $\overline{u_i' T'}$ is the turbulent heat flux. In order to simulate Eqns. (2), (3) and (4), relationships between the Reynolds stress tensor and turbulent heat flux with the mean velocity, temperature and pressure field must be solved. The main challenge in turbulence modeling is to devise a relationship between U_i and T with $\overline{u_i' u_j'}$ and $\overline{u_i' T'}$ that best reflects the physics of turbulence in the flow. Using the Boussinesq eddy-viscosity assumption, the Reynolds stresses are approximated as:

$$\overline{u_i' u_j'} = \nu_T \left(\frac{\partial U_i}{\partial x_j} + k\delta_{ij} \right) \tag{6}$$

ν_T is the eddy viscosity and k is the turbulent kinetic energy. The mean flow field obtained from making this assumption is acceptable for the calculations of engineering design. Additional differential equations must also be solved in order to calculate ν_T .

Heat Transfer Modeling

In order to solve Eqn. (4), the turbulent heat flux is related to the mean velocity field by using the Boussinesq approximation where

$$\overline{u_i' T'} = \frac{\nu_T}{\Pr_T}\frac{\partial T}{\partial x_i} \tag{7}$$

in all calculations, air is assumed to be the working fluid hence, the molecular Prandtl number is set to a constant value of 0.7. Viscous heating is neglected and a constant turbulent Prandtl number is introduced to model the turbulent heat flux,

$$k_T = \nu_T / \Pr_T \tag{8}$$

values of $\Pr_T$ vary from 0.73 to 0.92. In all calculations presented here, a constant value of $\Pr_T = 0.85$ is used.

Numerical Method

In the present study the three-dimensional grids made up of approximately 2342375 cells have been used. This grid is sufficient to ensure that all steep gradients in the flow and temperature fields are well resolved. Turbulent fluctuations in the velocity field are modeled using the RANS methodology. Assuming the turbulence to be isotropic, the wall effects are approximated using various turbulence models such as $RNG\ k-\varepsilon$, $SST\ k-\omega$, $Realizable\ k-\varepsilon$ and $v^2 - f$. Since wall functions are not considered for the $v^2 - f$ turbulence model in the simulations, all grids have

strong clustering close to the walls to ensure that the y^+ of the first computational node is almost equal to one. The SIMPLE algorithm couples the velocity and the pressure fields. The second-order upwind discretization method is applied to the convective fluxes in the momentum, energy and turbulence equations. Only steady-state calculations have been performed.

Boundary Conditions

The atmospheric static pressure and temperature of 295.2K are considered at the inlet boundary (velocity inlet). A turbulent intensity of 1% and a turbulent length scale equals to the nozzle diameter are also used in the simulations. The uniform temperature of 321.2K is imposed on the pedestal and the flat plate and the Nusselt number is computed from the local steady surface heat flux. At the solid boundaries velocity components and turbulent kinetic energy are set to zero and a zero derivative is also considered for the static pressure. At the upper boundary, atmospheric static pressure is imposed and radial derivatives for the other quantities are set to zero (pressure outlet condition).

Results and Discussions

In order to validate the numerical method, the results of the heat transfer coefficients on the top of the pedestal and the flat plate are compared with the available experimental data reported by Mesbah [10], in Figs. 2 and 3 respectively. The Reynolds number is 23000 and the dimensionless jet to target spacing is equal to six. According to these figures it can be seen that with the SST turbulence model, the heat transfer coefficients are underpredicted while they are overpredicted with the two $k-\varepsilon$ models, especially with realizable $k-\varepsilon$ model. The results show that the v^2-f turbulence model predicts the experimental results better than the three other investigated models in this study. This model provides better agreement with the experimental data for heat transfer coefficients at the top of the pedestal (Fig.2) with respect to those along the flat plate (Fig.3). Comparing the results of the present paper with those of the other investigators shows that these results are acceptable for the engineering design although the Nu numbers are not predicted very well along the flat plate (Fig. 3). Thus the same turbulence model is also used to simulate the turbulent air jet impinging normally on a simple flat plate. Fig. 4 shows the profile of surface heat transfer coefficient along the plate at the Reynolds number of 23000 and dimensionless nozzle to plate distance of six. Comparing this figure with the two previous figures, it can be concluded that existence of the pedestal increases the heat transfer coefficients and causes a local minimum in the heat transfer coefficient profile on the symmetry axis, which is in not similar to the results of the simple flat plate. It is evident from the results that for the simple flat plate, the maximum values of Nu occurs in vicinity of the stagnation point and approximately at dimensionless radial location of 2 (R/D=2) which is due to the high level of turbulence forming in the mixing layer at the edge of the jet producing a local maximum of turbulent kinetic energy at dimensionless radial location of 1.7 (R/D=1.7).

The normalized velocity ($\sqrt{u^2+v^2}/U_{bulk}$) at Reynolds numbers of 23000 and six different radial locations for jets impinging on the simple flat plate and pedestal are presented in Fig. 5 in order to compare the velocity profiles of the two cases. As it can be seen the normalized velocity magnitude increases from a value of zero to a maximum velocity and subsequently decays to a very small value at almost all the radial locations for the jet impinging on the simple flat plate while the velocity of the case with the pedestal has the same behaviour just at radial locations of 2, 2.5 and 3. The simulations also show that the maximum velocity magnitude of the case without the pedestal is larger than that of the other one (with the pedestal) at almost all radial locations except the dimensionless location value of 0.5. As it is expected, moving away from the stagnation point, the profiles of the velocity magnitude and maximum velocity of the two cases approach each other.

The velocity fields for jets impinging on a circular pedestal and simple flat plate are shown in Figs. 6 and 7 respectively. These velocity fields are illustrated by the contour plots of the velocity magnitude in the xy-plane and show a complex behavior around the stagnation points. Fig. 6 shows that the stagnation point is at the top of the pedestal on the symmetry axis, since the air emanating from the jet, decelerates in the axial direction. Then the flow turns sharply and forms a radial wall jet along the upper surface of the pedestal. At the corner of the pedestal, the flow separates and reattaches downstream on the plate. It creates a recirculation region that has a significant effect on the wall heat transfer. After reattachment, the flow develops into a wall jet along the plate. While according to Fig. 7 for the jet impinging on a simple flat plate the flow field is not so complex as when there is a pedestal on the plate. In this case it can be seen that the turbulent air jet strikes the plate and forms stagnation region directly below the pipe. The flow then moves radially away from the stagnation point creating an axisymmetric wall jet.

The contours of turbulent kinetic energy around the stagnation point are shown for the two kinds of jets impinging on plates with and without the pedestal in Figs. 8 and 9 respectively. As Fig. 8 shows the maximum value of turbulent kinetic energy (k) occurs near the pedestal top corners, due to the large turbulent shear stresses in the curved streamlines in those regions, and at the edge of the jet because of the high level of turbulence forming in the mixing layer. The recirculation region near the pedestal has also a

high value of the turbulent kinetic energy. While for the simple flat plate as the Fig. 9 shows the maximum value of turbulent kinetic energy takes place nearly at the dimensionless radial location of 1.7. It can also occur due to the large amount of turbulence in the mixing layer at the edge of the jet. One of the other results is that the maximum value of the turbulent kinetic energy of the jet impinging on a simple flat plate is higher than that of the other case.

Conclusions

In this study a numerical investigation has been presented to predict the flow fields and heat transfer characteristics of the jets, impinging on a simple flat plate and a circular pedestal with the purpose of studying the effect of the existence of the pedestal on the heat transfer characteristics. The flow is assumed to be incompressible. The Reynolds number is 23000 and the nozzle to target distance equals to six times of the jet diameters. The radiation heat transfer and buoyancy effects are neglected. Different turbulence models have been also studied to compare the ability of these models in prediction of the experimental results. The results show that the $v^2 - f$ model provides better agreement with experimental data especially at the top of the pedestal compared to the other investigated turbulence models. The simulations also show that the pedestal increases the amount of heat transfer and also causes a local minimum in profile of the heat transfer coefficient at the stagnation point which is in contrast to the results associated with jet impinging on the simple flat plate. One of the other results is that the maximum values of the velocity and turbulence kinetic energy of the jet impinging on the simple flat plate are larger than those of the jet impinging on the pedestal.

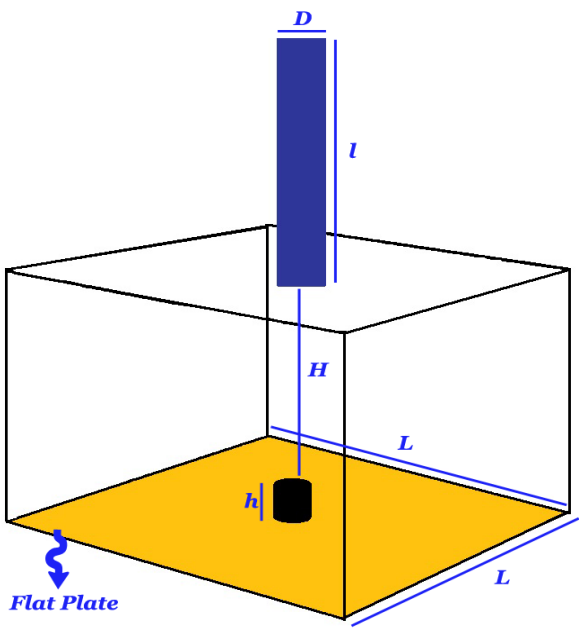

Fig. 1: Computational domain
(D): The jet diameter, (H): The jet to target distance.
(h): The height of the pedestal (h=2r=D/1.06 and r is the radius of the pedestal), (l): The jet length, (L): The flat plate length.

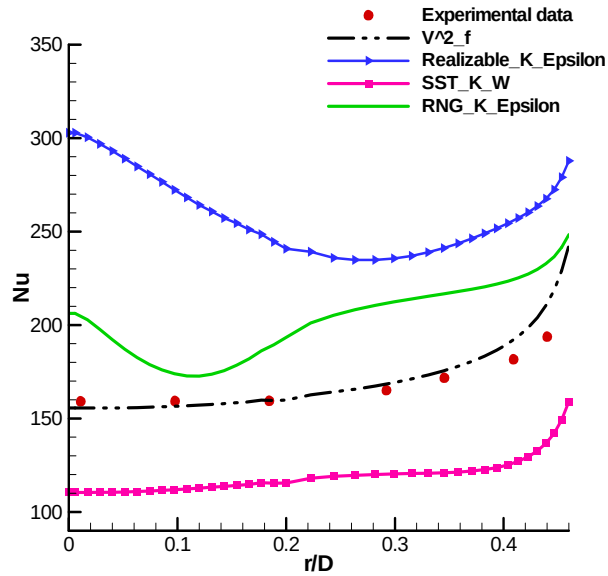

Fig. 2:The profiles of surface heat transfer coefficient along the top face of the pedestal (H/D=6, Re=23000)

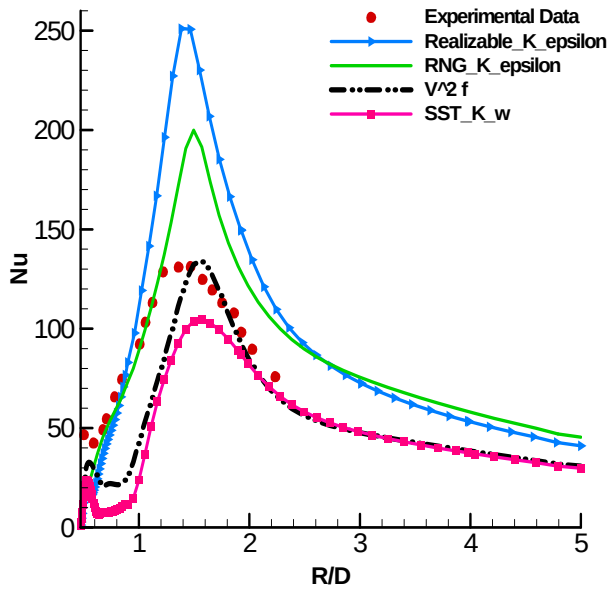

Fig. 3: The profiles of surface heat transfer coefficient along the flat plate (with the existence of the pedestal, H/D=6, Re=23000)

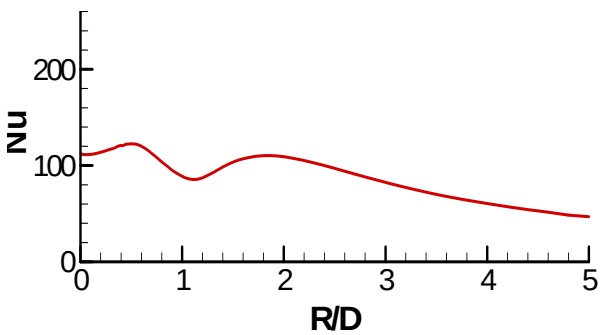

Fig. 4:The profile of surface heat transfer coefficient along the simple flat plate (without the pedestal, $v^2 - f$ model, H/D=6, Re=23000)

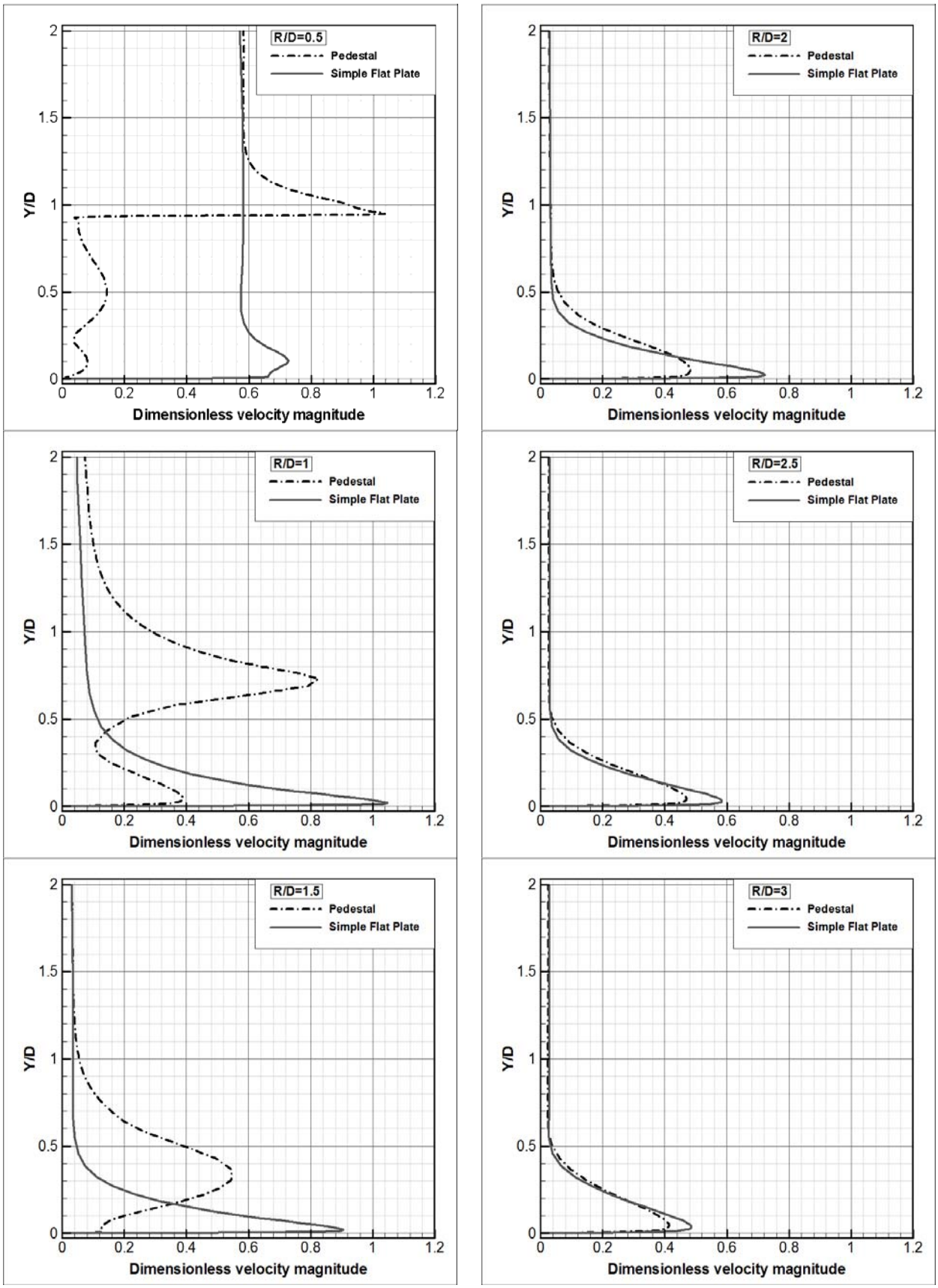

Fig. 5: The profiles of normalized velocity magnitude at different radial locations ($v^2 - f$ model, Re=23000, H/D=6).

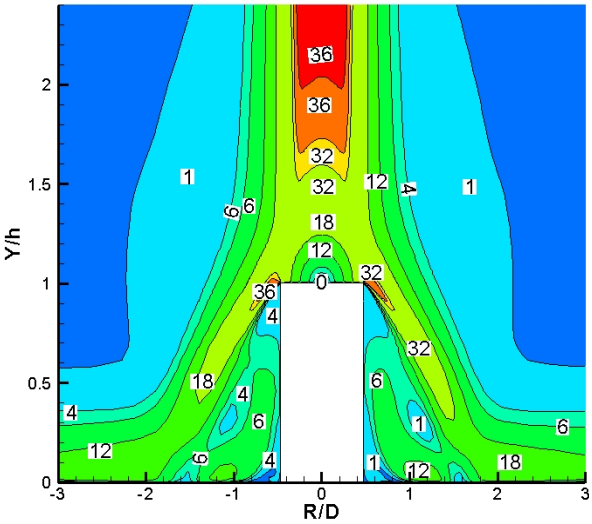

Fig. 6: The velocity contour around the pedestal ($v^2 - f$ model, H/D=6, Re=23000)

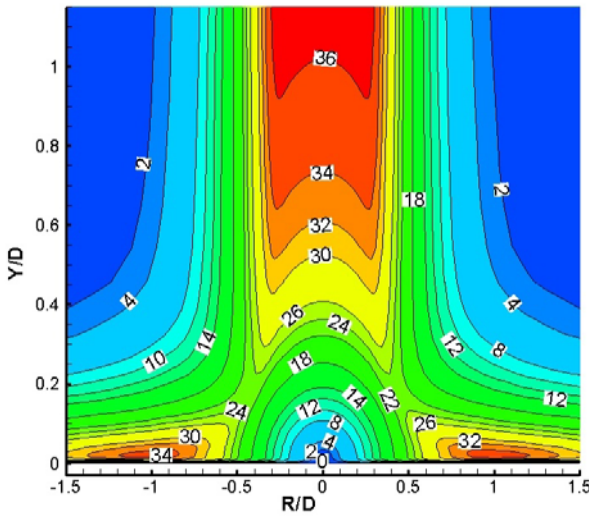

Fig. 7: The velocity contour around the stagnation point of the simple flat plate ($v^2 - f$ model, H/D=6, Re=23000)

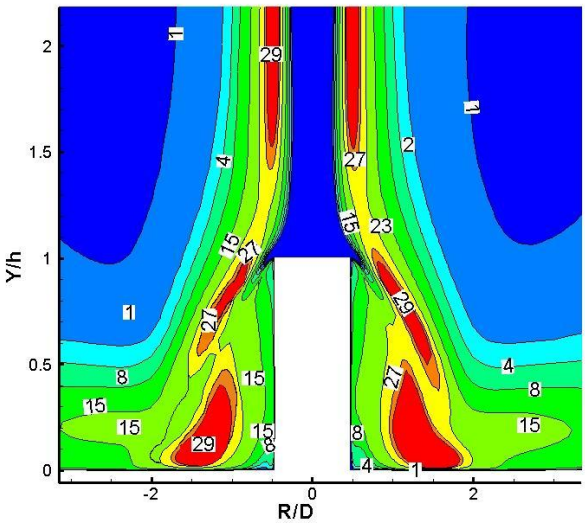

Fig. 8: The contour of turbulent kinetic energy around the pedestal ($v^2 - f$ model, H/D=6, Re=23000)

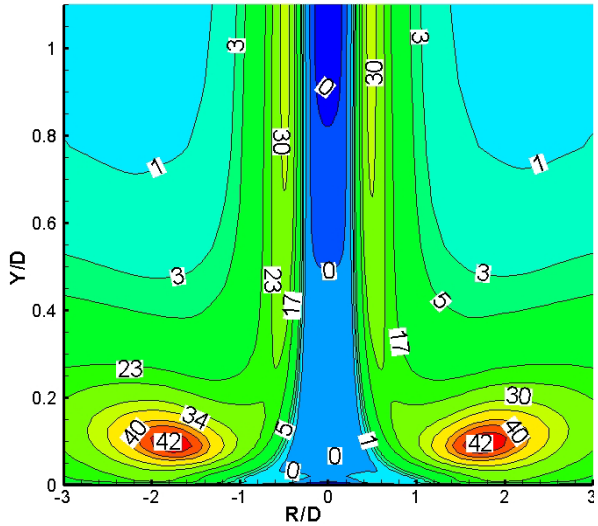

Fig. 9: The contour of turbulent kinetic energy around the stagnation point of the simple flat plate ($v^2 - f$ model, H/D=6, Re=23000)

References

1- Babic, D. Murray D.B. and Torrance A.A., Mist jet cooling of grinding processes, *Int. J. Mach. Tools Manufact*., Vol. 45, 2005, pp. 1171-1177.

2- Esposito E.I., Jet Impingement cooling configurations for gas turbine combustion. Master of Science in Mechanical Engineering, Mechanical Engineering Dept., Louisiana State Univ., Agricultural and Mechanical College, 2006.

3- Grady J.O., Jet impingement cooling of turbine blades by forced convection, *MAE 221A heat transfer*, Mechanical Department of Mechanical and Aerospace Engineering, University of California, San Diego, December 7, 2007.

4- Liu Q. Sleiti A.K. and Kapat J.S., Application of pressure and temperature sensitive paints for study of heat transfer to a circular impinging air jet, *International Journal of Thermal Sciences*, Vol. 47, 2008, pp. 749-757.

5- Rundstrom, D. Moshfegh B. and Ooi A., Rsm and *v*2 -*f* predictions of an impinging jet in a cross flow on a heated surface and on a pedestal, *16th Australasian Fluid Mechanics Conference*, Crown Plaza, Gold Coast, Australia, 2-7 December, 2007.

6- Kercher D.S. Lee J.B. Brand O. Allen M.G. and Glezer A., Microjet cooling devices for thermal management of electronics, *Transactions of the ASME*, Vol. 125, No. 2, 2003, pp. 359-366.

7- Cheng T.C. Chiou P.H. and Lin T.F., Visualization of mixed convective vortex rolls in an impinging jet flow of air through a cylindrical chamber, *International Journal of Heat and Mass Transfer*, Vol. 45, 2002, pp. 3357-3368.

8- Lee, J., Lee, S., The effect of nozzle aspect ratio on stagnation region heat transfer characteristics of elliptic impinging jet. *International Journal of Heat and Mass Transfer*, Vol. 43, 2000, pp. 555-575.

9- Siclari, M. and Migdal D., Development of theoretical models for jet-induced effects on vlstol aircraft. *Journal of Aircraft*, Vol. 13, No.12, 1976, pp. 938-944.

10- Mesbah, M., An Experimental Study of Local Heat Transfer to An Impinging Jet on Nonflat Surfaces: A Cylindrical Pedestal and A Hemispherically Concave Surface. Ph.D thesis, University of California, Davis, CA, 1996.

11- Choi M. Yoo H. Yang G. J.S. Lee J. and Sohn D., Measurements of impinging jet flow and heat transfer on a semi-circular concave surface. *International journal of heat and mass transfer*, Vol. 43, 2000, pp. 1811-1822.

12- Dano, Bertran, Flow Characteristics and heat transfer performance of a semiconfined impinging array of jets: effect of nozzle geometry, *International journal of heat and mass transfer*, Vol. 48, 2005: 691-701.

13- Kim N. S. and Giovannini A., Experimental study of turbulent round jet flow impinging on a square cylinder laid on a flat plate, *International journal of heat and fluid flow*, Vol. 28, 2007, pp. 1327-1339.

14- Behnia M. Pameix S. Shabany Y. and Durbin P., Numerical study of turbulent heat transfer in confined and unconfined impinging jets. *Int Journal of Heat and Fluid Flow*, Vol. 20, 1999, pp. 1-9.

15- Cziesla T. Biswas G. Chattopadhyay H. and Mitra N., Large-eddy simulation of flow and heat transfer in an impinging slot jet. *Int Journal Heat and Fluid Flow*, Vol. 22, 2001, pp. 500-505.

16- Chung Y. M. and Luo K.H., Unsteady heat transfer analysis of an impinging jet, *Journal of Heat Transfer, Transactions of the ASME*, Vol. 124., December 2002, pp. 1039-1048.

17- Merci B. Mesbah M.P.E. and Baughn J.W., Experimental and numerical study of turbulent heat transfer on a cylindrical pedestal, *International journal of heat and fluid flow*, Vol. 26, 2005, pp. 233-243.

18- Moeinfar M. Abdolahian S.M. Kahrom M. and Elahi. R., Investigating the Effect of Nozzle Geometry of a Jet Impinging Normally on a Pedestal on the Heat Transfer Characteristics, *Ankara International Aerospace Conference(AIAC2012)*, Turkey, Ankara, 14-16 September. 2011.

19- Parameswaran V. and Alpay S., Normal impingement of jets. *Journal of Aircraft*, Vol.11, No.3, 1974, pp. 189-191.