Computational investigation of slot jet impinging on a heated flat plate using low-Reynolds number modeling

Sushil Kumar Rathore¹*, Sanjay Singh Rathore², Sudarshan Singh³

¹Department of Mechanical Engineering, NIT Rourkela, Rourkela-769008, Odisha
²Department of Mechanical Engineering, NIT Patna, Patna-800005, Bihar
³Former Professor, Department of Mechanical Engineering, NIT Patna, Patna-800005, Bihar

*Corresponding author’s e-mail address: isushilrathore@gmail.com

Abstract. The present work reports the computational investigation of turbulent slot jet impinging on a heated plate subjected to isothermal and isoflux boundary conditions. The eddy viscosity based 2-equation low-Reynolds number $k-\varepsilon$ model of Yang and Shih has been considered. Low-Reynolds number (LRN) modeling employ near-wall integration which solves the entire boundary layer using very fine grid near the wall to handle the sharp gradient of variables prevailing in these regions. The Reynolds number of jet at the inlet is 9900 and the non-dimensional distance between the top and bottom plate i.e. $H/w$ is 7.5. The variation of centerline axial velocity, variation of local maximum axial velocity, profile of non-dimensional wall shear stress along impingement and confinement wall, distribution of local Nusselt number along impingement wall and confinement wall, and distribution of Reynolds shear stress in the domain are discussed.

1. Introduction

The jet impingement heat transfer is very efficient technique for cooling or heating the target surface. The excellent heat transfer coefficient associated with impingement heat transfer makes it suitable for wide ranges of practical and engineering applications. Some of the important applications areas are: cooling of turbine blades, boiler combustion chambers, cooling of hot metal plates and glass, food processing industries, thrust augmentation during short take-off and landing of fighter aircraft, automobile defroster, etc. [1-6].

Figure 1 shows physical domain of slot impinging jet flow problem under consideration. There are three regions of flow: free jet region, stagnation/impingement region and wall jet region. As the name implies, the free jet region remains unaffected from the effects of wall. The free jet region further comprises of two regions namely potential core region and shear layer region. The potential core region remains unaffected from the effect of viscosity and the velocity in this region is approximately equal to the jet inlet velocity. The impingement region forms when the jet strikes to the bottom plate and comes to the stagnation with subsequent conversion of Y-component of velocity into X-component of velocity. Sufficiently away from the impinging point, flow field resembles to characteristics of that of a wall jet flow.
The present numerical simulation reports the turbulent jet impingement flow and heat transfer using LRN $k-\varepsilon$ model of Yang and Shih [7] (YS). The YS model is reported to perform well in many complex flow and heat transfer problems [8, 9, 10]. Yang and Shih model takes into account Kolmogorov time scale in the near-wall region and conventional time scale ($k/\varepsilon$) in the off-wall region which makes the model more appropriate for complex flow situations. Apart from composite time scale feature, this model correctly mimics the variation of near-wall shear stress. The correct prediction of near-wall behavior is key for the success of low-Reynolds number model. Very limited works have been reported on flow and heat transfer behavior of turbulent impinging slot jet using LRN turbulence models. The present study is an attempt to fill this gap. The advantage of LRN turbulence model is that it does not require wall functions unlike the high-Reynolds number turbulence model. The first grid is placed in viscous sublayer with $y^+ = yu_t/v$ value close to 1 where $u_t = \sqrt{\tau_w/\rho}$ is friction velocity, $\tau_w$ is wall shear stress, $\rho$ is density of fluid, $v$ is kinematic viscosity of fluid. The LRN turbulence model is more suitable for the flow situations e.g. flow involving separation, presence of transition and/or laminar regions of flow, natural and mixed convection flow, etc.

![Diagram](image_url)  

**Figure 1.** The schematic diagram of present problem (slot jet impingement) under consideration.

2. Mathematical Formulation

2.1. Governing Equations

For present computations, RANS equations are used. For turbulence closure, LRN $k-\varepsilon$ model of Yang and Shih (YS-model) [7] has been considered.

Non-dimensional variables

\[ U = \frac{u}{U_0}, \quad V = \frac{v}{U_0}, \quad X = \frac{x}{w}, \quad Y = \frac{y}{w}, \quad P = \frac{p - p_0}{\rho U_0^2}, \quad k_n = \frac{k}{u_0^2}, \quad \varepsilon_n = \frac{\varepsilon}{u_0^2/w}, \quad \nu_n = \frac{\nu_n}{v} \]  

(1)

Here $U_0$ is the velocity at the jet inlet, $w$ is width of nozzle at the inlet, $p_0$ is ambient pressure, $v$ is kinematic viscosity of fluid.

Non-dimensionalized governing equations

Continuity equation:
\[ \frac{\partial U}{\partial X} + \frac{\partial V}{\partial Y} = 0 \] (2)

Turbulent kinetic energy \( (k_n) \) equation:

\[ \frac{\partial (Uk_n)}{\partial X} + \frac{\partial (Vk_n)}{\partial Y} = \frac{1}{Re} \frac{\partial}{\partial X} \left[ \left( 1 + \frac{\nu_{t,n}}{\sigma_k} \right) \frac{\partial k_n}{\partial X} \right] + \frac{1}{Re} \frac{\partial}{\partial Y} \left[ \left( 1 + \frac{\nu_{t,n}}{\sigma_k} \right) \frac{\partial k_n}{\partial Y} \right] + G_n - \varepsilon_n \] (3)

Rate of dissipation \( (\varepsilon_n) \) equation:

\[ \frac{\partial (U\varepsilon_n)}{\partial X} + \frac{\partial (V\varepsilon_n)}{\partial Y} = \frac{1}{Re} \frac{\partial}{\partial X} \left[ \left( 1 + \frac{\nu_{t,n}}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon_n}{\partial X} \right] \left( \left( 1 + \frac{\nu_{t,n}}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon_n}{\partial Y} \right] + C_1 f_1 \frac{\varepsilon_n}{k_n} G_n - C_2 f_2 \frac{\varepsilon_n^2}{k_n} + E_n \] (4)

Production by shear \( (G_n) \):

\[ G_n = \frac{\nu_{t,n}}{Re} \left[ \left( \frac{\partial U}{\partial X} \right)^2 + 2 \left( \frac{\partial V}{\partial Y} \right)^2 + \left( \frac{\partial U}{\partial Y} + \frac{\partial V}{\partial X} \right)^2 \right] \] (5)

Eddy viscosity \( (\nu_{t,n}) \):

\[ \nu_{t,n} = C_\mu f_\mu Re \frac{k_n^2}{\varepsilon_n} \] (6)

The model functions are defined as follows:

\[ f_\mu = \left( 1 + \frac{1}{\sqrt{Re}} \right) \left[ 1 - \exp\left( -aR_{ey} - bR_{ey}^2 - cR_{ey}^3 \right) \right] \] (7)

Where, \( a = 1.5 \times 10^{-4}, \quad b = 5 \times 10^{-7}, \quad c = 1 \times 10^{-10} \)

\[ f_1 = f_2 = \frac{\sqrt{Re}}{1 + \sqrt{Re}} \] (8)

\[ E_n = \frac{2 \nu_{t,n}}{Re} \left( \frac{\partial^2 U}{\partial X^2} \right) \] (9)

Where \( \sqrt{Re} = \frac{k_n^2}{\nu} \) and \( \sqrt{Re_y} = \frac{\sqrt{k} y}{\nu} \)

2.2 Nondimensional Boundary Conditions

Figure 2 shows the computational domain of present problem under consideration. It the present work, it would be sufficient to consider half portion of domain for computation due to symmetry. At the jet inlet \( V=1 \) and \( k_n = 1.5I^2 \). The turbulence intensity \( I \) is taken as 0.02 for the present study. The value of turbulence intensity is taken as 0.02 following the earlier works [11-12]. The turbulent kinetic energy \( k_n \) is taken as zero on the solid walls. The dissipation rate at the jet inlet is taken as \( \varepsilon_n = \left( k_n^{3/2} \cdot C_\mu^{3/4} \right)/0.07 \). Whereas, the dissipation rate at the solid wall is taken as \( \varepsilon_n = \frac{2}{Re} \left( \frac{\partial k_n}{\partial n} \right)^2 \). The symmetry boundary condition is applied along symmetry axis which implies, X-component of velocity is zero along this axis and Neumann boundary condition \( \partial \phi / \partial X = 0 \) is applied. A fully developed flow of \( \partial \phi / \partial X = 0 \) is imposed at outlet, where \( \phi = U, V, k_n, \varepsilon_n \). The temperature of cold jet coming out from the nozzle is same as that of the ambient. The heated impingement plate is subjected to either isothermal boundary condition or isoflux boundary condition. The confinement surface is taken as adiabatic wall.
3. Numerical Scheme
The present CFD simulation has been done using computer code written in C++. Finite volume method and the staggered grid arrangement are used. The discretization of convection and diffusion terms has been carried out using Power law upwind scheme (due to stability of the scheme) and central difference scheme, respectively. The SIMPLE algorithm [13] has been used for velocity pressure coupling. In present work, TDMA and line-by-line solver are used to find the variables of interest. The pseudo-transient approach [14] has been implemented for under-relaxation of momentum equations and turbulence equations. The domain size chosen for the present study is 66 × 7.5 (non-dimensional). The Reynolds number at the jet inlet is 9900.

4. Validation and Grid Independence Study
Figure 3 (a) shows comparison of the centerline axial velocity ($V_c$) with the published results of Seyedein et al. [5]. In order to further validate, the profile of local Nusselt number is compared with the work of Cadec [15] and Gardon and Akfirat [16] as shown in Figure 3(b). Overall, good agreements have been noticed. The grid sizes of 351 × 171, 381 × 191 and 401 × 201 are considered for grid independence study. Figure (4) presents the grid independence study where non-dimensional variation of wall shear stress is plotted for grid sizes 351 × 171, 381 × 191 and 401 × 201. The variation of non-dimensional wall shear stress with three grid densities is very small however to be on safer side, grid size of 381 × 191 is chosen for the present computation. The grid layout for the present problem is shown in Figure 5.

Figure 2. The computational domain of present problem under consideration.

Figure 3. Validation of present computational study: (a) variation of centreline axial velocity along spacing, (b) Variation of local Nusselt number along impingement wall.
5. Results and Discussion

The flow pattern of the present problem is shown with the help of streamlines plot as shown in Figure 6. A recirculation region is evident below the top confinement wall with the vortex center (12, 3.4). The variation of centerline velocity ($V_c$) for $H/w=7.5$ is demonstrated in Figure 7. It can be observed from Figure 7 that the maximum value of 1 is observed at the jet inlet and the minimum value of 0 at the stagnation point. The value of centerline axial velocity ($V_c$) is equal to jet inlet velocity equal to 1 (one) up to approximately $y/H = 0.4$ due to the presence of potential core region. Figure 8 shows the axial variation of local maximum axial velocity $U_{max}$. At the jet inlet, $U_{max}$ is zero as the jet is directed to the impingement surface at $90^\circ$. It reaches a maximum value of 0.95 at axial location $X = 3$ and then decreases very rapidly in the downstream due to interaction with recirculation region and finally decays steadily in the wall jet region.

The distribution of local Nusselt number along the heated impingement plate is illustrated in Figure 9. The heated impingement plate is at higher temperature than the cold impinging jet. The temperature of the jet at inlet is same as that of the ambient. The local Nusselt number distribution for isothermal boundary condition is shown by solid lines whereas dashed lines are used to show the local

![Figure 4](image4.png)

**Figure 4.** Grid independence test showing variation of non-dimensional wall shear stress along impinging wall

![Figure 5](image5.png)

**Figure 5.** Grid layout for the present problem (zoomed)
Nusselt number distribution for isoflux boundary condition. The peak value (maxima) of local Nusselt number is observed in the stagnation region, as the normal velocity gradient in the X-direction is high resulting in better heat transfer. The off-stagnation minima of local Nusselt number occurs due to interconversion of normal component of velocity into axial component of velocity. The zone of high heat transfer is area of interest for the heat transfer problem. The qualitative variation of local Nusselt number for isothermal and isoflux boundary condition is almost similar.

Figure 10 illustrates the non-dimensional temperature profile at various axial locations for isothermal and isoflux boundary conditions on the impingement wall. The higher temperature is observed in the thermal boundary layer close to the wall. The temperature gradient along Y direction is very small away from the impingement wall. The temperature decreases in the downstream locations for both isothermal and isoflux wall as evident from Figure 10. Figure 11 shows the distribution of Reynolds shear stress in the domain. The Reynolds shear stresses (\( -\overline{u'v'} \)) are higher in the surroundings region of stagnation point due to turning of jet and the interaction of jet with the recirculating fluid. The Reynolds shear stresses outside the recirculation region are moderate.

![Figure 6. Streamlines plot showing the flow pattern](image1)

![Figure 7. Profile of centreline axial velocity.](image2)
Figure 8. Profile of $U_{\text{max}}$ along the X direction

Figure 9. Profiles of local Nusselt number variation along impingement wall.

Figure 10. (a) Variation of Temperature at various axial locations for isothermal condition, (b) Variation of Temperature at various axial locations for isoflux condition.
6. Conclusion

The computational study of two-dimensional turbulent slot jet impinging on a heated plate whose temperature is higher than jet inlet temperature is presented. The Reynolds number of the flow considered is 9900 and non-dimensional spacing between the top and bottom wall i.e. H/w is 7.5. Low-Reynolds number $k-\epsilon$ model proposed by Yang and Shih [7] has been chosen for turbulence closure. The capability of low-Reynolds number modelling has enabled to capture the various parameters like velocity, wall shear stress, temperature, local Nusselt number profile, etc. in the viscous sublayer region also which otherwise cannot be captured by high-Reynolds number modeling. The higher value of Local Nusselt number is observed in the impingement region indicating the zone of higher heat transfer. Wall shear stress is zero at stagnation point and the maximum at $X = 1$ as observed in case of impingement surface. The Reynolds shear stresses $(-u'v')$ are higher in the impingement region.

References

[1] Bovo M and Davidson L 2013 Numerical Heat Transfer, Part A: Applications 64(4) 290-316
[2] Inada S, Miyasaka Y and Izumi R 1981 Bulletin of the JSME 24 1803-1810.
[3] Vader D T, Incropera F P and Viskanta R 1991 International Journal of Heat and Mass Transfer 34 611-623.
[4] Bula A J, Rahman M M and Leland J E 2000 Numerical Heat Transfer 38 45-66.
[5] Seyedein S H, Hasan M and Mujumdar A S 1994 Appl. Math. Modelling 18 526-537.
[6] Sahoo D and Sharif M A R 2004 International Journal of Thermal Sciences 43 877–887.
[7] Yang Z and Shih T H 1993 Am. Inst. Aeronaut. Astronaut. Journal 31 1191–1198.
[8] Rathore, S K, Das M K 2013 International Journal of Heat and Mass Transfer 61 365–380.
[9] Rathore, S K, Das M K 2015 International Journal of Thermal Sciences 89 337–356.
[10] Rathore, S K, Das M K 2016 Heat and Mass Transfer 52 437–457.
[11] Pramanik S, Achari A M and Das M K 2012 Industrial & Engineering Chemistry Research 51 9153-9163.
[12] Afroz A and Sharif M A R 2013 International Journal of Thermal Sciences 74 1-13.
[13] Patankar S V 1980 Numerical Heat Transfer and Fluid Flow (New York: Hemisphere Publishing Co.).
[14] Versteeg H K and Malalasekera W 1996 An Introduction to Computational Fluid Dynamics: The Finite Volume Method (England: Addison Wesley Longman Limited).
[15] Cadek F F 1968 *A fundamental investigation of jet impingement heat transfer* Ph.D. Thesis University of Cincinnati.

[16] Gardon R and Akfirat J C 1965 *International Journal of Heat and Mass Transfer* 8 1261-1272.