Hot air impingement on a flat plate using Large Eddy Simulation (LES) technique

C. Plengsa-ard* and M. Kaewbumrung
Mechanical Engineering Department Kasetsart University, Chatuchak, Bangkok 10900, Thailand

* E-mail: fengecpp@ku.ac.th, Tel: 02-797-0999 Ext 1833

Abstract. Impinging hot gas jets to a flat plate generate very high heat transfer coefficients in the impingement zone. The magnitude of heat transfer prediction near the stagnation point is important and accurate heat flux distribution are needed. This research studies on heat transfer and flow field resulting from a single hot air impinging wall. The simulation is carried out using computational fluid dynamics (CFD) commercial code FLUENT. Large Eddy Simulation (LES) approach with a subgrid-scale Smagorinsky-Lilly model is present. The classical Werner-Wengle wall model is used to compute the predicted results of velocity and temperature near walls. The Smagorinsky constant in the turbulence model is set to 0.1 and is kept constant throughout the investigation. The hot gas jet impingement on the flat plate with a constant surface temperature is chosen to validate the predicted heat flux results with experimental data. The jet Reynolds number is equal to 20,000 and a fixed jet-to-plate spacing of H/D = 2.0. Nusselt number on the impingement surface is calculated. As predicted by the wall model, the instantaneous computed Nusselt number agree fairly well with experimental data. The largest values of calculated Nusselt number are near the stagnation point and decrease monotonically in the wall jet region. Also, the contour plots of instantaneous values of wall heat flux on a flat plate are captured by LES simulation.

1. Introduction
In recent years, heat transfer of a hot jet impinging on a solid wall is widely applied in a cooling and heating process. Turbulent jets are initiated and discharged from the nozzle exit. High heat transfer rate at a surface is promoted. These rates are affected by many parameters for example Reynolds number, the distance from a stagnation point in radial direction, and a nozzle-plate spacing. There are many previous experimental studies on jet impingement reported in the literature. Wang et al. [1] investigated heat transfer characteristics during jet impingement on a flat surface while Zhou et al. [2] showed the numerical analysis of turbulent round jet impingement. Heat transfer by liquid jets impinging on a hot flat surface was proposed by Hosain et al. [3]. Moreover, Lyttle et al. [4] studied air jet impingement for small values of nozzle-plate spacing, while the circular air jet impingement on a flat plate with a uniform heat flux on the surface was reported by Baughn et al. [5], Singh et al. [6], and Mohanty et al. [7]. Physics of the jet impingement was explained by Gardon et al. [8]. Moreover, Zuckerman et al. [9] proposed the average heat transfer correlation from a plate and heat transfer rate was investigated in more details. The experiment of Cooper et al. [10] showed the mean velocity and turbulence statistics for a single jet impingement from a long pipe. Only a few papers for three dimensional numerical investigation of hot air jet impingement on a flat plate were published in the literature, thus the investigation of hot air jet impingement is needed. In the present study, CFD code is used to calculate
the flow of jet over the flat plate and to analyze heat transfer from the surface. Modeling and meshing are created in FLUENT.

2. Turbulence model
To simulate flow field, the Large Eddy Simulation (LES) is used for the research. The flow field is separated into large-scale (resolved or filtered) and small scale (sub-grid) motion by filtering (spatially averaging) local flow variables in LES. A subgrid-scale Smagorinsky-Lilly model is present and the eddy viscosity is modeled. The Smagorinsky constant is set to 0.1 and is kept constant throughout the investigation. More detail information for a particular turbulence model can be found in the literature.

ANSYS-FLuent usually has included the standard law of the wall in the code (LES-SL). However, the classical Werner-Wengle wall model (LES-WW) is focused in this research. Since the wall shear stress is strongly dependent on wall heat transfer, the effect of heat transfer is investigated. The air impingement jet is chosen to be a test case.

3. Werner-Wengle wall shear stress model formulation [11]

\[
\tau_w = \frac{2\rho v U}{\Delta y}, \quad U \leq \frac{v}{2\Delta y} A^{1-B}
\]

\[
\tau_w = \rho \left\{ \frac{1 - B}{2} A^{1-B} \left( \frac{v}{\Delta y} \right)^{1+B} + \frac{1 + B}{A} \left( \frac{v}{\Delta y} \right)^{B} U \right\}^{\frac{2}{1+B}}, \quad U > \frac{v}{2\Delta y} A^{1-B}
\]

where \(\tau_w\) is wall shear stress, \(U\) is the magnitude of gas velocity at the first grid point parallel to the wall, \(\rho\) is fluid density, \(\Delta y\) is the height of the first cell near walls, \(v\) is kinematic viscosity, \(A\) and \(B\) are equal to 8.3 and 1/7 respectively.

4. Test case description
The computational domain for impinging jets is divided into the two regions. First, in the top pipe region, the initial velocity profiles at the inlet are assumed by a fully developed flow. To generate turbulent motions, the initial condition for sub-grid scale turbulent kinetic energy is scaled with 5% of the jet bulk velocity in the pipe. The length of the pipe is 30D, which can provide a fully developed flow at the exit, and the adiabatic wall is applied at the pipe surfaces.

In the base region, the distance between the pipe exit and the wall surfaces is \(H=2D\) and the turbulent jet impinges normally on the surface with size of 25D. The inlet and initial condition of air in the pipe and the base region is set to 700 K. The numerical study of impinging jets is tested with a computational grid, as shown in figures 1 and 2. In addition, figure 3 shows the CFD mesh at inlet of the computational domain. In order to be consistent with available experimental data, constant wall temperature boundary conditions are used on the top wall and the bottom, or impingement, wall, while the outlet pressure boundary conditions are applied at the ambient exits. The constant wall temperature is chosen to be 450 K, which matched with experimental data. The boundary conditions of the sub-grid scale turbulent kinetic energy are the Neumann boundary conditions.
Figure 1. CFD mesh of the impingement jet on the flat plate

Figure 2. CFD mesh of the computational domain

Figure 3. CFD mesh at inlet of the computational domain
Cylindrical coordinate is used for the computational domain of the impinging jet. The original point (r=0) is located at the center line. The velocity components in the axial (z), radial (r) and azimuthal (θ) directions are denoted by W, U and V. The research also focuses heat transfer on the impingement wall, when Werner-Wengle wall model is applied in the code. The non-dimensional heat transfer coefficient, i.e., Nusselt number on the impingement surface are calculated and compared with existing experimental data.

5. Results and discussion
The comparisons of computed and measured time averaged Nusselt number with three different mesh sizes can be seen in figure 4. The experiment work from Baughn et al. [5] are used for comparison. First, the numerical study of impinging jets is tested with three different computational grids i.e., LES-grid I, LES-grid II and LES-grid III. There is no significant difference between LES-grid II and LES-grid III. To obtain a better result, grid II is chosen for the simulation in this research. The axial and radial mean velocity components at the stagnation point (r/D=0) and r/D=0.5 as predicted by LES-SL and LES-WW models are shown in figures 5 and 6, respectively. Also, the root mean square (rms) of axial and radial velocities as predicted by LES-SL and LES-WW models are presented in figures 7 and 8. These axial mean velocity and rms values are normalized by the jet bulk velocity, Wb. The experiment data are selected from works of Geers et al. [12] to compare with calculated values. The mean axial velocity, W, and the mean radial velocity, U, at r/D=0 (centerline) are shown in figure 5. On the centerline, the axial mean velocity remains nearly constant from the pipe exit (Z/D=2.0) up to Z/D=0.4, while, below the position Z/D=0.4, the flow decelerates while the jet flows approach to the wall. As can be seen in figure 6, at the position r/D =0.5, the strong flow deflection due to the impingement wall makes the mean radial velocity (U/Wb) start to increase and reaches the maximum value near the wall whereas the mean axial velocity (W/Wb) approaches to zero.

![Figure 4.](image_url)
Figure 5. Comparisons of measured and computed mean axial and radial velocity profiles as predicted by LES-SL and LES-WW models at a stagnation point \((r/D = 0)\) with \(Re = 20,000\) and \(H/D = 2.0\).

Figure 6. Comparisons of measured and computed mean axial and radial velocity profiles as predicted by LES-SL and LES-WW models at \(r/D = 0.5\) with \(Re = 20,000\) and \(H/D = 2.0\).
Figure 7. Comparisons of measured and computed root mean square (rms) of axial and radial velocities as predicted by LES-SL and LES-WW models at a stagnation point (r/D = 0) with Re =20,000 and H/D =2.0

For the rms values as shown in figure 7 and 8, the mean axial and radial velocity fluctuations decrease down to zero due to wall effect. At the centerline (r/D=0) in figure 7, the rms values of the axial and radial velocity fluctuation are over-predicted in the region Z/D < 0.3 while the results agree fairly well when Z/D > 0.3. At r/D = 0.5, the velocity fluctuations are higher than those of values at the centerline. The rms values at the point r/D = 0.5 agree fairly well with the selected experiment data until Z/D < 0.4. The discrepancies between the rms values of the velocity fluctuations predicted by LES-SL, LES-WW and experiment data can be observed in figure 8 in the region Z/D > 0.4.

Figure 8. Comparisons of measured and computed root mean square (rms) of axial and radial velocities as predicted by LES-SL and LES-WW models at r/D = 0.5 with Re =20,000 and H/D =2.0
Figure 9. Instantaneous velocity vector field

Figure 10. Contours of the instantaneous jet temperature as predicted by LES-WW model

Figure 11. Contours of the instantaneous heat flux as predicted by LES-WW model
The instantaneous velocity vector field of wall jet are illustrated in figure 9. The development of wall jet can be seen. The instantaneous vectors field reveals the absence of the small eddy motion within the stagnation zone. The thin boundary layer near walls are formed in the radial direction after jet deflection, while the flow patterns of acceration wall jet are observed. Figures 10 and 11 demonstrate the contour plots of the instantaneous jet temperature and heat flux as predicted by LES-WW model. The plate surface temperature is chosen to be 400 K in the research. The transfer of heat is from the higher wall jet temperature to the lower temperature of the plate. Figure 11 shows the contour plots of the instantaneous heat flux as predicted by LES-WW model. The magnitude of plotted heat fluxes are varied from 723 to 47,132 W/m², while the peak heat transfer rate is near the stagnation zone. The contour of heat fluxes is non-uniform and the heat flux results from LES-WW model can provide more flow structure and more local information on the impingement wall.

Figure 12. Time averaged Nusselt number as predicted by LES-SL and LES-WW models with Re = 20,000 and H/D = 2.0

During the post-processing step, the predicted Nusselt number is stored in each wall surface of the impingement wall. The plots of the time-averaged values of these surface properties with the centroid of those surfaces are shown in Figure 12. In this research, Nusselt number defined as

\[
Nu = \frac{hD_h}{k} = \frac{q_w D_h}{(T_{jet} - T_w)k}
\]

(3)

where \( q_w \) is the convective heat transfer from hot gas to wall, \( T_{jet} \) is gas temperature of wall cells and \( T_w \) is wall temperature. The physical properties are evaluated at the film temperature which is defined as

\[
T_f = \frac{1}{2}(T_{jet} + T_w)
\]

(4)

Figure 12 presents the time averaged Nusselt number as predicted by LES-SL and LES-WW models with Reynolds number is 20,000 and H/D is about 2.0. It is clear that the values of Nusselt number from the measurement are largest at the stagnation point and decrease monotonically in the wall jet region.
The Nusselt number predictions from the LES-SL and LES-WW models agree fairly well with the experimental data within the stagnation zone, while both models show under-prediction when r/D > 2.0.

6. Conclusions
Heat transfer and flow field resulting from a single hot air impinging wall is investigated in this study. The simulation is carried out using computational fluid dynamics (CFD) commercial code FLUENT. To simulate fluid flows, Large Eddy Simulation (LES) approach with a subgrid-scale Smagorinsky-Lilly model is employed. The classical Werner-Wengle wall model is used to compute the predicted results of velocity near walls. The Smagorinsky constant in the turbulence model is set to 0.1 and is kept constant throughout the investigation. The hot gas jet impingement on the flat plate with a constant surface temperature is chosen to validate the predicted heat flux results with experimental data. The jet Reynolds number is equal to 20,000 and a fixed jet-to-plate spacing of H/D = 2.0. Nusselt number on the impingement surface are calculated. As predicted by the wall model, the instantaneous computed Nusselt number agree fairly well with experimental data. The largest values of calculated Nusselt number are near the stagnation point and decrease monotonically in the wall jet region. The local information of instantaneous wall heat flux on a flat plate are captured by LES simulation.

Acknowledgements
This material is based upon work supported by Faculty of Engineering Kasetsart University. In addition, the authors are grateful for the intellectual support from PhD. candidate Mongkol Kaewbumrung from Applied Combustion Lab, Mechanical Engineering Kasetsart University.

References
[1] Wang B, Lin D, et al. 2016 Applied Thermal Engineering 100 pp 902-910
[2] Zhou T, Xu D, et al. 2016 Applied Thermal Engineering 100 pp 55-61
[3] Hosan M L, Fdhila R B, et al. 2015 Applied Energy 164 pp 934-943
[4] Lytle D and Webb B W 1991 In Experimental Heat Transfer, Fluid Mechanics and Thermodynamics ed. Keffler J F, Shah R K and Ganic E N (Elsevier)
[5] Baughn, J and Shimizu S 1989 ASME J. Heat Transfer 111 pp 1096-1098
[6] Singh D, Premachandran B, et al. 2013 Numer. Heat Transfer Part A 64(2) pp 153-185
[7] Mohanty A K and Tawfek A 1992 Int. J. Heat Mass Transfer 36(6) pp 1639-1647
[8] Gardon R and Cobonpue, J 1962 Int. Developments in Heat Transfer Int. Heat Transfer Conf. ASME pp 454-460
[9] Zuckerman N and Lior N 2005 ASME J. Heat Transfer 127(5) pp 544-552
[10] Cooper D, Jackson D C, et al. 1993 Intl J. Heat Fluid Flow 36 pp 2675-2684
[11] Werner H and Wengle H 1989 In Proc. of 7th Symposium on Turbulent Shear Flows (Stanford University, California) pp 10.21-10.26
[12] Geers L, Hanjalic K, et al. 2004 Exp. Fluids 36 pp 946-958