NUMERICAL SIMULATION OF HEAT TRANSFER CHARACTERISTICS UNDER SEMI-CONFINED IMPINGING SLOT JETS

SHI YULING

NATIONAL UNIVERSITY OF SINGAPORE

2004
NUMERICAL SIMULATION OF HEAT TRANSFER CHARACTERISTICS UNDER SEMI-CONFINED IMPINGING SLOT JETS

SHI YULING
(M. Eng, Xi’an Jiaotong University, PRC)

A THESIS
FOR THE DEGREE OF DOCTOR OF PHILOSOPHY
DEPARTMENT OF CHEMICAL AND BIOMOLECULAR ENGINEERING
NATIONAL UNIVERSITY OF SINGAPORE
2004
ACKNOWLEDGEMENT

This research would not have been possible without the time, effort, and encouragement of a number of people. First and foremost, I would like to thank my supervisors, Dr. M.B. Ray and Prof. A.S. Mujumdar, for their guidance throughout the past three years. They have provided insight and expertise to overcome both large and minor problems during this research. Despite his busy schedule, Prof. Mujumdar tirelessly gave his precious time and knowledge to ensure the successful completion of this thesis. I also greatly appreciate the kindness, understanding, and care of Dr. Ray from the bottom of my heart.

Special thanks should also go to my husband, Huang Yueyuan, and my parents. They have had to endure with my ups and downs, but have supported and sustained me throughout. They are the persons to whom I have turned to for comfort and relief. Thank you for your patience, understanding and support.

Last but not least, I wish to thank my son, Ze An, for his persistence and intelligence. I am proud of you.
# TABLE OF CONTENTS

ACKNOWLEDGEMENT i  
TABLE OF CONTENTS ii  
SUMMARY vi  
NOTATION viii  
LIST OF FIGURES xi  
LIST OF TABLES xvi  

CHAPTER 1: INTRODUCTION 1  
1.1 Background information and industrial motivation  
1.2 Research objectives and scope  

CHAPTER 2: LITERATURE REVIEW 10  
2.1 Introduction  
2.2 Numerical studies of impinging jets heat transfer  
  2.2.1 Studies by various k-ε models and Reynolds Stress Model  
    2.2.1.1 Single slot impinging jet  
    2.2.1.2 Multiple slot impinging jets  
    2.2.1.3 Impinging round jets  
    2.2.1.4 Impinging jet with cross-flow  
  2.2.2 Studies with LES and DNS approaches  
  2.2.3 Studies using others models  
  2.3 Studies using both experimental and numerical methods  
  2.4 Heat transfer in turbulent gas-particle suspension flow  

CHAPTER 3: NUMERICAL SIMULATION 36  
3.1 Single phase laminar flow  
3.2 Single phase turbulent flow
3.2.1 The governing equations
3.2.2 Models of turbulence
  3.2.2.1 The standard k-ε model
  3.2.2.2 RSM model
3.3 Multiphase flow
  3.3.1 Governing equations
  3.3.2 Gas particle interaction
  3.3.3 Particle-wall conduction heat transfer
3.4 Near wall treatment
3.5 Numerical techniques of turbulent impinging jet simulations
  3.5.1 Boundary conditions
  3.5.2 Numerical parameters
    3.5.2.1 Relaxation factors
    3.5.2.2 Convergence criteria
    3.5.2.3 Grid independence tests

CHAPTER 4: HEAT TRANSFER UNDER A TURBULENT IMPINGING SLOT JET

4.1 Introduction
4.2 Results and discussion
  4.2.1 Comparison of results from various turbulence models
  4.2.2 Effect of turbulent Prandtl Number
  4.2.3 Effect of Reynolds Number
  4.2.4 Effect of turbulence level at the nozzle exit on heat transfer
  4.2.5 Effect of the near wall function on the predicted Nusselt number
  4.2.6 Effect of the magnitude of the heat flux
4.3 Conclusions

CHAPTER 5: EFFECT OF TEMPERATURE DIFFERENCE BETWEEN THE JET AND IMPINGEMENT SURFACE ON HEAT TRANSFER

5.1 Introduction
5.2 Results and discussion
5.2.1 Small temperature difference case
5.2.2 Large temperature difference case
5.2.3 Effect of jet Reynolds number on the $\text{Nu}_{\text{ave}}$ and $\text{Nu}_0$

5.3 Conclusions

CHAPTER 6: EFFECTS OF Pr ON IMPINGING JET HEAT TRANSFER UNDER A SLOT JET

6.1 Introduction
6.2 Results and discussion
   6.2.1 Laminar flow
      6.2.1.1 Effect of fluid Prandtl number on heat transfer rates
      6.2.1.2 Correlations between Prandtl number and Nusselt number
   6.2.2 Turbulent flow
      6.2.2.1 Effect of fluid Prandtl number on heat transfer rates
      6.2.2.2 Correlations between Prandtl number and Nusselt number
6.3 Conclusions

CHAPTER 7: EFFECT OF CROSS FLOW ON TURBULENT FLOW AND HEAT TRANSFER CHARACTERISTICS UNDER NORMAL AND OBLIQUE SEMI-CONFINED IMPINGING SLOT JETS

7.1 Introduction
7.2 Results and discussion
   7.2.1 Effect of cross flow and jet angles on flow pattern
   7.2.2 Effect of cross-flow on normal impinging jet heat transfer rate
   7.2.3 Effect of jet angle on local heat transfer rate distribution
   7.2.4 Effect of crossflow and jet angles on the average Nusselt number
   7.2.5 Effect of temperature difference between the jet and cross-flow
7.3 Conclusions

CHAPTER 8: HEAT TRANSFER UNDER TURBULENT MULTIPLE SLOT IMPINGING JETS OF GAS-PARTICLE SUSPENSION

iv
8.1 Introduction

8.2 Results and discussion

8.2.1 Comparison between experiment and simulation
8.2.2 Effect of the conduction heat transfer
8.2.3 Effects of wall factors
8.2.3.1 Effect of wall material
8.2.3.2 Effect of reflection coefficient on heat transfer
8.2.3.3 Effect of impingement wall temperature
8.2.4 Effect of particle factors on heat transfer
8.2.4.1 Effect of constant C and loading ratio
8.2.4.2 Effect of particle diameter
8.2.4.3 Effect of particle material
8.2.5 Effect of inlet conditions
8.2.5.1 Effect of inlet Reynolds number
8.2.5.2 Effect of gravity direction on heat transfer

8.3 Conclusions

CHAPTER 9: CONCLUSIONS

9.1 General conclusions
9.2 Major contributions to knowledge
9.3 Recommendations and for future work

REFERENCES

PUBLICATIONS
In this thesis, results of computational fluid dynamic simulation of impinging jet heat transfer under semi-confined slot jets with and without cross-flow are reported. While most of the results focus on both laminar and turbulent single jets, simulation results for heat transfer in gas-particle suspension flow for multiple jets are also presented. Initially, the simulation results for a single semi-confined turbulent slot jet impinging normally on a flat plate were compared with selected experimental data from the open literature. The standard $k\varepsilon$ and Reynolds stress turbulence models were used. Effects of turbulence models, near wall functions, turbulent Prandtl number, jet turbulence, jet Reynolds number, the type of thermal boundary condition at the target surface, as well as temperature differences between the jet and impingement surface are discussed in the light of available experimental data. Results indicate the advantages and shortcomings of the two turbulence models and the important parameters that affect the heat transfer characteristics of the impinging jet flow, specifically the jet Reynolds number, turbulent Prandtl number, jet turbulence, and near wall treatments. Further, for impinging jet heat transfer with large temperature difference between the jet and the target surface, an attempt is made to identify the optimal definitions of the Nusselt number.

While most of the numerical experiments were carried out for air jets, some simulations were performed for a variety of fluids including both liquids and gases. The results show that H$_2$ and He yield much higher heat transfer coefficients than air, Ar, N$_2$ and NH$_3$ under the same flow and boundary conditions. Also, the surface heat
transfer coefficient for the water jet is much higher than those for the other fluids studied here.

The simulation of the flow and heat transfer characteristics for an oblique single semi-confined turbulent slot jet impinging into an imposed cross-flow of air of the same or different temperature was also performed. Effects of the various flow and geometric parameters (e.g. jet-to-cross-flow mass ratio, nozzle-to-target spacing, jet angle and the temperature difference between the jet and the cross-flow) were evaluated.

The heat transfer rate in impinging jet flows has been observed to increase due to the presence of suspended inert particles in the air. Accurate predictions of the heat transfer characteristics of impinging jets of gas-particle suspension remain a major challenge. In the final phase of this work, heat transfer under multiple impinging jets of gas-particle suspension flow was numerically predicted by the Eulerian-Lagrangian model including the conductive heat transfer due to particle-wall collisions. The numerical results were compared with available experimental data. Finally, a parametric study characterizing the effect of geometric and particle parameters, and boundary conditions on impinging jet heat transfer in gas-particle flow was conducted.

The above studies indicate that CFD simulations provide a useful design tool for impinging slot jets under different conditions once an optimum simulation scheme is identified.
### NOTATION

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>contact area</td>
</tr>
<tr>
<td>C</td>
<td>conduction heat source due to particle wall collision</td>
</tr>
<tr>
<td>$C_p$</td>
<td>specific heat</td>
</tr>
<tr>
<td>D</td>
<td>hydraulic diameter</td>
</tr>
<tr>
<td>$D_p$</td>
<td>graphite powder diameter</td>
</tr>
<tr>
<td>e</td>
<td>energy</td>
</tr>
<tr>
<td>$E_{pw}$</td>
<td>equivalent elastic modulus of particle and impingement wall</td>
</tr>
<tr>
<td>$F_D$</td>
<td>drag force</td>
</tr>
<tr>
<td>$G_k$</td>
<td>production of kinetic energy</td>
</tr>
<tr>
<td>h</td>
<td>surface heat transfer coefficient</td>
</tr>
<tr>
<td>H</td>
<td>nozzle-to-plate spacing</td>
</tr>
<tr>
<td>I</td>
<td>turbulence intensity</td>
</tr>
<tr>
<td>k</td>
<td>turbulent kinetic energy</td>
</tr>
<tr>
<td>$k_B$</td>
<td>turbulent kinetic energy at point B</td>
</tr>
<tr>
<td>$k_{p,w}$</td>
<td>thermal conductivity of particle and wall</td>
</tr>
<tr>
<td>$k_v$</td>
<td>von Karman’s constant (=0.42)</td>
</tr>
<tr>
<td>l</td>
<td>turbulence length scale</td>
</tr>
<tr>
<td>L</td>
<td>length of the impingement surface</td>
</tr>
<tr>
<td>$L_0$</td>
<td>particle loading ratio</td>
</tr>
<tr>
<td>M</td>
<td>cross-flow parameter (cross-flow mass flow rate / jet mass flow rate)</td>
</tr>
<tr>
<td>$m_{pw}$</td>
<td>equivalent mass particle and impingement wall</td>
</tr>
<tr>
<td>$\bar{m}_p$</td>
<td>average mass of the particle in control volume</td>
</tr>
</tbody>
</table>
\* \( m_{p0} \) initial mass flow rate of the particle injection tracked

\( \text{Nu} \) Nusselt number, \( hW/\lambda \)

\( P \) pressure

\( \text{Pr} \) Prandtl number \( C_p\mu/\lambda \)

\( Q, q \) heat flux

\( R \) equivalent radius of particle and impingement wall

\( R^2 \) determination factor of curve-fitting line

\( \text{Re} \) Reynolds number, \( \rho UW/\mu \)

\( S_p \) surface area of the particle

\( t_c \) contact duration

\( T \) temperature

\( T_{ig}, T_{ip} \) jet inlet temperature of gas and particle

\( u_i, u_j \) fluctuating velocities in x, y direction, respectively

\( U_i, U_j \) velocity in x and y direction respectively

\( U \) average velocity

\( U_B \) mean velocity of the fluid at point B

\( v_p \) particle normal impact velocity

\( W \) nozzle width

\( \text{WS} \) with conduction heat transfer condition

\( \text{WOS} \) without conduction heat transfer condition

\( y^+ \) dimensionless distance

\( Y \) distance from wall

\( y_B \) distance from point B to wall

\( \varepsilon \) rate of dissipation of turbulence energy

\( \varepsilon_B \) production of dissipation rate
<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>µ</td>
<td>dynamic viscosity</td>
</tr>
<tr>
<td>ν</td>
<td>kinematic viscosity or poisson ratio</td>
</tr>
<tr>
<td>ρ</td>
<td>density</td>
</tr>
<tr>
<td>τ</td>
<td>shear stress</td>
</tr>
<tr>
<td>λ</td>
<td>thermal conductivity</td>
</tr>
<tr>
<td>σ</td>
<td>constant</td>
</tr>
<tr>
<td>ζ</td>
<td>reflection coefficient</td>
</tr>
<tr>
<td>θ</td>
<td>jet angle</td>
</tr>
</tbody>
</table>

**Subscripts**

<table>
<thead>
<tr>
<th>Subscript</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>stagnation point</td>
</tr>
<tr>
<td>avg</td>
<td>average</td>
</tr>
<tr>
<td>cf</td>
<td>cross-flow</td>
</tr>
<tr>
<td>f</td>
<td>film</td>
</tr>
<tr>
<td>g</td>
<td>gas</td>
</tr>
<tr>
<td>in</td>
<td>inlet</td>
</tr>
<tr>
<td>j</td>
<td>jet at inlet</td>
</tr>
<tr>
<td>out</td>
<td>outlet</td>
</tr>
<tr>
<td>p</td>
<td>particle</td>
</tr>
<tr>
<td>R</td>
<td>radiation</td>
</tr>
<tr>
<td>ref</td>
<td>reference</td>
</tr>
<tr>
<td>w</td>
<td>impingement surface</td>
</tr>
<tr>
<td>wi</td>
<td>impingement surface near the inlet area</td>
</tr>
<tr>
<td>wo</td>
<td>impingement surface near the exit</td>
</tr>
<tr>
<td>t</td>
<td>turbulent</td>
</tr>
</tbody>
</table>
**LIST OF FIGURES**

<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Figure 1.1</td>
<td>Flow regions of semi-confined impinging slot jet</td>
<td>2</td>
</tr>
<tr>
<td>Figure 1.2</td>
<td>Flow geometry of the slot impinging jet without cross flow</td>
<td>8</td>
</tr>
<tr>
<td>Figure 3.1</td>
<td>Near wall treatments</td>
<td>47</td>
</tr>
<tr>
<td>Figure 3.2</td>
<td>Effect of simulation region on results</td>
<td>53</td>
</tr>
<tr>
<td>Figure 3.3</td>
<td>Definition of different regions in the computational domain</td>
<td>54</td>
</tr>
<tr>
<td>Figure 3.4</td>
<td>Examples of typical grid structure</td>
<td>54</td>
</tr>
<tr>
<td>Figure 3.5</td>
<td>Effect of grid size on surface heat transfer coefficient</td>
<td>55</td>
</tr>
<tr>
<td>Figure 4.1</td>
<td>Comparison of simulated Nusselt number to the experimental data from literature for case 1</td>
<td>58</td>
</tr>
<tr>
<td>Figure 4.2</td>
<td>Comparison of simulated Nusselt number to the experimental data from van Heiningen for case 2</td>
<td>58</td>
</tr>
<tr>
<td>Figure 4.3</td>
<td>Comparison of Nusselt number predicted by standard k-ε model with increasing turbulent Pr no. to the experimental data</td>
<td>61</td>
</tr>
<tr>
<td>Figure 4.4</td>
<td>Comparison of Nusselt number predicted by the RSM model with increasing turbulent Pr no. to the experimental data</td>
<td>61</td>
</tr>
<tr>
<td>Figure 4.5</td>
<td>Effect of the jet Reynolds number on the surface heat transfer coefficients</td>
<td>63</td>
</tr>
<tr>
<td>Figure 4.6</td>
<td>Streamline contour predicted by the standard k-ε model for different turbulence intensities and length scales</td>
<td>65</td>
</tr>
<tr>
<td>Figure 4.7</td>
<td>Effects of turbulence intensity and length scale on Nusselt number and kinetic energy predicted by the standard k-ε model</td>
<td>65</td>
</tr>
<tr>
<td>Figure 4.8</td>
<td>Streamline contour predicted by RSM model for different turbulence intensities and length scales</td>
<td>66</td>
</tr>
<tr>
<td>Figure 4.9</td>
<td>Effect of turbulence intensity and turbulence length scales on turbulent kinetic energy predicted by RSM model</td>
<td>66</td>
</tr>
</tbody>
</table>
Figure 4.10: Effect of turbulence intensities and turbulent length scales on Nusselt number predicted by RSM model 67

Figure 4.11: Effect of wall functions on Nusselt number distributions 69

Figure 4.12: Comparison of Nusselt number at different thermal boundary conditions 70

Figure 5.1: Effect of small $\Delta T$ between the jet and the impingement surface on $\text{Nu}_j$ for cooling and heating. a: standard k-$\varepsilon$ model; b: RSM model 76

Figure 5.2: Effect of small $\Delta T$ between the jet and the impingement surface on $\text{Nu}_f$ for cooling and heating. a: standard k-$\varepsilon$ model; b: RSM model 77

Figure 5.3: Effect of small $\Delta T$ between the jet and the impingement surface on $\text{Nu}_w$ for cooling and heating. a: standard k-$\varepsilon$ model; b: RSM model 78

Figure 5.4: Effect of large $\Delta T$ between the jet and the impingement surface on $\text{Nu}_j$ for cooling and heating. a: standard k-$\varepsilon$ model; b: RSM model 80

Figure 5.5: Effect of large $\Delta T$ between the jet and the impingement surface on $\text{Nu}_f$ for cooling and heating. a: standard k-$\varepsilon$ model; b: RSM model 81

Figure 5.6: Effect of large $\Delta T$ between the jet and the impingement surface on $\text{Nu}_w$ for cooling and heating. a: standard k-$\varepsilon$ model; b: RSM model 82

Figure 5.7: Effect of temperature difference between the jet and impingement surface on jet Nusselt number for heating condition by the standard k-$\varepsilon$ model ($\text{Re}=1500$) 85

Figure 5.8: Effect of temperature difference between the jet and impingement surface on jet Nusselt number for heating condition by the standard k-$\varepsilon$ model ($\text{Re}=3000$) 86

Figure 5.9: Effect of temperature difference between the jet and impingement surface on jet Nusselt number for heating condition by the standard k-$\varepsilon$ model ($\text{Re}=6000$) 87

Figure 5.10: Effect of temperature difference between the jet and impingement surface on jet Nusselt number for heating condition by the standard k-$\varepsilon$ model ($\text{Re}=12000$) 88

Figure 5.11: Effect of temperature difference between the jet and impingement surface on film Nusselt number for heating condition by the standard k-$\varepsilon$ model ($\text{Re}=1500$) 89

Figure 5.12: Effect of temperature difference between the jet and impingement surface on film Nusselt number for heating condition by the standard k-$\varepsilon$ model ($\text{Re}=3000$) 90
Figure 5.13: Effect of temperature difference between the jet and impingement surface on film Nusselt number for heating condition by the standard k-ε model (Re=6000) 91

Figure 5.14: Effect of temperature difference between the jet and impingement surface on film Nusselt number for heating condition by the standard k-ε model (Re=12000) 92

Figure 5.15: Effect of temperature difference between the jet and impingement surface on wall Nusselt number for heating condition by the standard k-ε model (Re=1500) 93

Figure 5.16: Effect of temperature difference between the jet and impingement surface on wall Nusselt number for heating condition by the standard k-ε model (Re=3000) 94

Figure 5.17: Effect of temperature difference between the jet and impingement surface on wall Nusselt number for heating condition by the standard k-ε model (Re=6000) 95

Figure 5.18: Effect of temperature difference between the jet and impingement surface on wall Nusselt number for heating condition by the standard k-ε model (Re=12000) 96

Figure 5.19: Effect of jet Reynolds number on average jet Nusselt number for heating predicted by the standard k-ε model 98

Figure 5.20: Effect of jet Reynolds number on stagnation jet Nusselt number for heating predicted by the standard k-ε model 98

Figure 6.1: Effect of Pr number on the local Nusselt number distributions for laminar flow 103

Figure 6.2: Effect of Pr number on surface heat transfer coefficient distributions for laminar flow 104

Figure 6.3: Local Nu distributions for gases with similar fluid Pr number for laminar flow 105

Figure 6.4: Heat transfer coefficient distributions for gases with similar fluid Pr number for laminar flow 105

Figure 6.5: Effect of Pr number on stagnation and average Nusselt numbers for laminar flow 107

Figure 6.6: Effect of Pr number on the Nusselt number distributions for turbulent flow 109
Figure 6.7: Effect of Pr number on surface heat transfer coefficient distributions for turbulent flow 109

Figure 6.8: Local Nusselt number distributions of different gases with similar fluid Prandtl number predicted by the standard k-ε model 110

Figure 6.9: Surface heat transfer coefficient distributions of different gases with similar fluid Prandtl number for turbulent flow 111

Figure 6.10: Effect of Prandtl number on stagnation and average Nusselt numbers for turbulent flow 113

Figure 7.1: Flow geometry of the impinging slot jet with cross-flow 116

Figure 7.2: Streamline contours for $\theta = -30^\circ$ for two structures under small cross flow value, $M = 0.05$ 117

Figure 7.3: Streamline contours for $\theta = -30^\circ$ for two structures under large cross flow value, $M = 1$ 118

Figure 7.4: Streamline contours for $\theta = 30^\circ$ for two structures under small cross flow value, $M = 0.05$ 118

Figure 7.5: Streamline contours for $\theta = 30^\circ$ for two structures under large cross flow value, $M = 1$ 119

Figure 7.6: Effect of cross-flow on Nusselt number distributions predicted by the standard k-ε model 121

Figure 7.7: Effect of $\theta$ on Nusselt number predicted by the standard k-ε and RSM models for $M=0$ (The values of H/W, Re, $\theta$, $T_j$, $T_{cf}$, $T_w$ and W are same for both sides) 122

Figure 7.8: Effect of $\theta$ on Nusselt number predicted by the standard k-ε and RSM models for $M=0.05$ (The values of H/W, Re, $\theta$, $T_j$, $T_{cf}$, $T_w$ and W are same for both sides) 122

Figure 7.9: Effect of $\theta$ on Nusselt number predicted by the standard k-ε and RSM models for $M=0.25$ (The values of H/W, Re, $\theta$, $T_j$, $T_{cf}$, $T_w$ and W are same for both sides) 123

Figure 7.10: Effect of $\theta$ on Nusselt number predicted by the standard k-ε and RSM models for $M=1$ (The values of H/W, Re, $\theta$, $T_j$, $T_{cf}$, $T_w$ and W are same for both sides) 123

Figure 7.11: Effect of M on average Nusselt number predicted by the standard k-ε and RSM models for H/W=2.6 125
Figure 7.12: Effect of M on average Nusselt number predicted by the standard k-ε and RSM models for H/W=8

Figure 7.13: Effect of temperature difference between the jet and cross-flow on the Nusselt number predicted by the standard k-ε model

Figure 8.1: Flow configuration of the multiple impinging slot jets

Figure 8.2: Comparison between the experimental results and the numerical results for WOS condition

Figure 8.3: Effect of conduction heat transfer between particles and wall on heat transfer for both WOS and WS conditions

Figure 8.4: Effect of wall materials on heat transfer for both WOS and WS conditions

Figure 8.5: Effect of reflection coefficient on heat transfer for both WOS and WS conditions

Figure 8.6: Effect of impingement wall temperature on heat transfer

Figure 8.7: Effect of loading ratio on heat transfer for both WOS and WS conditions

Figure 8.8: Effect of constant C in heat source term on computed heat transfer for both WOS and WS conditions

Figure 8.9a: Effect of particle loading ratio on heat transfer for L/W=5, C=1.0

Figure 8.9b: Effect of particle loading ratio on heat transfer for L/W=5, C=1.5

Figure 8.10a: Effect of particle loading ratio on heat transfer for L/W=7, C=1.0

Figure 8.10b: Effect of particle loading ratio on heat transfer for L/W=7, C=1.5

Figure 8.11: Effect of loading ratio on outlet gas temperature distributions for WOS and WS conditions

Figure 8.12: Effect of particle diameter on heat transfer for L/W=5

Figure 8.13: Effect of particle diameter on heat transfer for L/W=7

Figure 8.14: Effect of particle materials on heat transfer

Figure 8.15: Effect of inlet Reynolds number on heat transfer

Figure 8.16: Effect of gravity direction on heat transfer
LIST OF TABLES

Table 1.1: Summery of parameters studied in this work 8
Table 2.1: Summary of the configurations studied in literature 10
Table 2.2: List of general reviews on jet impingement 11
Table 2.3: A literature summary on the experimental studies of impinging jet flow and heat transfer 27
Table 2.4: Studies on gas-particle flow 33
Table 4.1: Geometric parameters and boundary conditions for the two cases tested for turbulence models 57
Table 4.2: Geometric parameters, boundary conditions and maximum surface heat transfer coefficient for the test cases for different Reynolds numbers 63
Table 4.3: Geometric parameters and boundary conditions for the test cases 64
Table 4.4: Geometric parameters and boundary conditions for the test cases 70
Table 6.1: Thermal properties of fluids studied here at T = 300 K 102
Table 6.2: Stagnation and average Nusselt number correlations 112
Table 8.1: Structure and operating parameters 131
Table 8.2: Material properties of graphite, glass, copper, aluminum and steel 132
Table 8.3: Correlations between heat transfer and loading ratio 138
Chapter 1

Introduction

1.1 Background information and industrial motivation

Impinging jets of various configurations are used in numerous industrial processes because of their highly favorable heat and mass transfer characteristics. Impinging jets provide much higher convective heat and mass transfer rates than those with the same amount of gas flowing parallel to the target surface. The heat transfer coefficient for the typical application of impinging jets including many heating, cooling and drying processes is a few times (typically, 2-10 times) higher than that of a cross circulation dryer (Seyedein et al.) (1995). Moreover, impinging jets provide the potential of fine and fast control of local transfer rates by varying operating parameters such as the jet velocity and size of the nozzle opening. Depending upon the application, either slot or round jets, single or multiple jets and single phase or gas-particle two phases can be selected in impinging jets. The major applications of impinging jets include photographic films and paper, annealing of nonferrous metal sheet and glass, internal cooling of the leading edge of turbine blades, etc. Polat et al. (1989) provided detailed descriptions of the different flow regions in impinging jet. Figure 1.1 shows the flow regions of a single semi-confined impinging jet. In the potential core region, the axial velocity remains almost the same as the nozzle exit velocity. In the impingement region, the static pressure increases as a result of the sharp decrease in mean axial velocity. Upon impingement, the flow deflects and starts to accelerate along the impingement surface. In the wall jet region, the boundary layer grows along the
impingement surface. The free jet region is characterized by the associated processes of decrease of centerline velocity and spreading of the jet in the transverse direction.

![Diagram showing flow regions of semi-confined impinging slot jet](image)

**Figure 1.1** Flow regions of semi-confined impinging slot jet

Although the basic heat and mass transfer in impinging slot jets can be shaped by the flow field, the effects of numerous parameters, such as nozzle geometry and size, nozzle configuration, location of exhaust ports, nozzle-to-target spacing, surface motion, and operating variables such as cross flow and jet axis velocity, complicate the analysis.

At present, complete understanding of the influence of all the design and operating parameters is lacking. With the advent of high performance computing, numerical experimentation is gradually replacing expensive and tedious laboratory and pilot-scale studies wherever possible. Many researchers simulated the turbulent impinging jet using different turbulence models (Amano and Brandt (1984) and Seyedein et al. (1994)). However, with the recent development of the numerous powerful computational fluid dynamics (CFD) programs, use of the commercial CFD programs has been proven to be a useful tool in numerically experimenting with complex fluid flow problems. Morris et al. (1996, 1999) used the commercial CFD package FLUENT to calculate the flow-field and local heat transfer coefficient distribution in
submerged and confined liquid jet impingement. However, their studies show significant deviation from the experimental data. Both the turbulence and near wall models were suspected to be responsible for the difference.

Most experimental and computational studies of impingement heat transfer, which form the basis for design of impingement heat transfer equipment, have been made with small temperature differences between the jet and the target surface. Heat transfer rate can be further enhanced by increasing the temperature difference between the inlet and the impingement surface. Under small temperature differences, all fluid properties can be taken as constant (i.e. temperature-independent). However, many industrial applications for impinging jets involve processes at large temperature differences, for example, paper drying and turbine blade cooling. Under such large temperature difference conditions, the thermo-physical properties of the fluid change with local temperature. Few experimental studies exist for high impingement temperature difference between the jet and impingement surface. Das (1982) analyzed these studies and indicated that due to the shortcomings of the experimental techniques and large differences in their data, these studies did not provide a reliable basis for design and optimization of industrial processes using impinging jet. Das et al. (1985) also presented data on the effect of large temperature differences on the local and average heat transfer rates under a confined single slot jet by experiments over a range of temperature differences, from 50 to 300 °C. More recently Heikkila and Milosavljevic (2002) presented an overview of experimental investigation of impingement heat transfer rate at high air impingement temperatures from 100 to 700 °C, under arrays of round jets, a problem of considerable industrial interest in the design of Yankee dryers for tissue paper in particular. However, the studies on heat transfer under large
temperature differences are not adequate for the design and optimization of impinging jet used in industrial processes.

Generally, the dimensionless heat transfer rate, the Nusselt number is used to report the heat transfer behavior. The local Nusselt number for an isothermal impingement surface can be defined as $Nu_x = \frac{h_x W}{\lambda}$, where $\lambda$ is the fluid thermal conductivity, $h_x = \frac{q}{\Delta T}$ and $\Delta T$ is the temperature difference between the jet and impingement surface. The fluid conductivity can be calculated according to different reference temperatures. The most popular method is to calculate the $\lambda$ by film temperature, $T_f = \frac{(T_w + T_j)}{2}$. It is well known that the film Nusselt number changes significantly under large temperature difference. For a Nusselt number defined at a certain reference temperature by which the spread in Nusselt number under large $\Delta T$ is less, that will be able to furnish heat transfer data under large $\Delta T$.

Although jet impingement has been extensively studied and several excellent reviews of the contemporary research are available, the influence of fluid thermo-physical properties has received little attention. Only a few studies have reported the effect of fluid Prandtl number on heat transfer rate. Many researchers have reported experimental or numerical studies on impingement heat transfer for air. The heat transfer rates of some liquids, such as FC-77 have also been investigated. Garimella and Rice (1995) developed heat transfer correlations for the effect of fluid Prandtl number by employing a fixed Prandtl number exponent of 0.4. More recently, Li and Garimella (2001) experimentally determined the effect of Prandtl number on heat
transfer of round jet and accounted for fluid properties in their correlations of stagnation and average Nusselt number by deducing these exponents based on regression of their experimental data. Proper understanding of the effect of fluid properties on the flow and heat transfer characteristics is very important especially in the design of liquid impinging jet in industrial applications such as the cooling of combustion engines, and high-performance electrical circuits.

It is well-established that single jets provide the best convective heat transfer rate in the impingement and the adjoining wall jet region of the impinged surface. However, due to high heat load, in industrial practice it is necessary to use multiple jets. There, the interaction between jets can have crucial effects on their heat transfer performance. Saad (1981) has shown that if the jets interact with each other before impingement, the average heat transfer rate under the jets is reduced. If the spent flow from each jet is not allowed to exit the enclosure without interaction with the neighboring jets, it is forced to cross the normal jet flow and deflect it towards the exhaust, thereby reducing its thermal performance. The influence of cross flow together with the inclined jet on impingement heat transfer has received little attention in literature. This cross-flow effect is obviously more significant for two dimensional slot jets and less so for arrays of round jets which provide opportunity for the spent flow from upstream jets to negotiate their way around the downstream jet towards the exhaust port without significant interaction. Cross-flow in industrial equipment is therefore undesirable but also unavoidable since it is practically not feasible to design the equipment with a large number of jets that exhaust individually. Cross-flow due to neighboring jets is often termed “induced cross-flow” in the literature.
Induced cross-flow can also result from the motion of the target surface as in the case of paper dryers where the high-speed sheet entrains significant amount of air as a boundary layer upon which the jets impinge in the drying chamber. Mujumdar et al. (1985) investigated turbulent jets impinging on a moving surface in the presence of a cross-flowing stream using the high-Reynolds-number k-ε model. The results showed that the cross-flow and wall motion had significant effects on Nusselt number and wall shear stress distributions. Saad (1981) investigated heat transfer under a row of slot jets and showed the negative effects of the induced cross-flow from upstream jets on heat transfer. Oblique impinging jet plays an important role in electronic chip cooling because inclined jets not only offer localized cooling but also serve to guide the spent air away from the hot spots. Goldstein and Franchett (1988) studied the local heat transfer for a jet issuing from a square-edge orifice and impinging at different angles (40-90°) onto a flat surface using naphthalene sublimation method. The results showed that the displacement of the peak heat transfer location was mainly due to the angle of inclination. Ward et al. (1991) investigated the heat transfer between a circular air jet impinging onto a uniform cross-flow of air over a flat surface coated with naphthalene. The maximum value of heat transfer was found to depend on the impinging jet angle and the velocity ratio between the impinging jet and the cross flow. Despite their obvious practical implications, the normal and oblique impinging jets in cross flow have received very little attention both experimentally and numerically. Few researchers have studied the effect of cross-flow as well as jet inclination angles on slot jet impingement flow and heat transfer behavior.

Heat transfer in turbulent gas-particle flow is an essential part of many industrial processes. Addition of inert particles in impinging jet turbulent flow is expected to
have an important influence on the dynamic flow characteristics and on the heat
transfer between the wall and the suspended particles (Mujumdar and Huang (1995)).
Numerous numerical studies on the influence of particles on turbulent flow and heat
transfer characteristics have been performed for channel flows. However, very little
research has been reported on impingement flows of gas-particle suspensions. Yoshida
et al. (1990) investigated the turbulence structure and heat transfer mechanism for a
two-dimensional impinging jet with gas-particle suspensions using laser-Doppler
anemometry. The results showed that with increase of loading ratio, the Nusselt
number markedly increased in the vicinity of the stagnation point. Hosseinalipour and
Mujumdar (1995) numerically studied the flow and heat transfer in confined opposing
jets of particle suspensions using Eulerian-Lagrangian models. More recently,
Yokomine et al. (2002) experimentally and numerically investigated the heat transfer
mechanism of multiple impinging jets of gas-particle suspensions and evaluated the
effects of nozzle Reynolds number, solid loading ratio, distance from jet exit to
impingement surface, spacing between jets and solid particle characteristics on the heat
transfer coefficient. However, the predicted Nusselt number did not agree well with
their experimental data; the results matched only qualitatively. Optimum design and
scale-up of such systems need a thorough understanding of impinging jet gas-solid
flow and heat transfer behavior.

1.2 Research objectives and scope

In light of the above studies, comprehensive numerical experiments on impinging jet
heat transfer including all of the above aspects have been conducted in this study. The
objectives of this work are to predict the flows and heat transfer rates between the
semi-confined impinging slot jets for both single phase and gas-particle two phases
using the commercial CFD software, FLUENT. The numerical results were compared with selected experimental data, followed by detailed parametric studies. Numerical simulations were conducted to characterize impinging jet heat transfer varying the following parameters:

- Geometric parameters (H/W and W) and boundary conditions (Re, turbulence intensity of inlet, \(T_w\), and \(q_w\)).
- Fluid properties under various thermal boundary conditions.
- Thermal properties of fluid and the Prandtl number
- Jet-to-cross-flow mass ratio and jet angles
- Inclusion of inert particles

Figure 1.2 illustrates the basic flow geometry of the slot impinging jet simulated in this work. The commercial CFD software, FLUENT is used to solve the governing equations involved in impinging jet heat transfer. The matrices of parameters studied in this work are summarized in Table 1.1.

![Flow geometry of the slot impinging jet without cross flow](image)

**Figure 1.2** Flow geometry of the slot impinging jet without cross flow

<table>
<thead>
<tr>
<th>Boundary condition</th>
<th>Jet Configuration</th>
<th>Fluids</th>
</tr>
</thead>
<tbody>
<tr>
<td>Re</td>
<td>Single jet</td>
<td>Gas: Air, Ar, NH(_3), H(_2), He, C(_2)H(_2)</td>
</tr>
<tr>
<td>(\Delta T)</td>
<td>Cross-flow</td>
<td>High Prandtl fluids (Liquid)</td>
</tr>
<tr>
<td>(q_w)</td>
<td>Inclined jet</td>
<td></td>
</tr>
<tr>
<td>Turbulence intensity</td>
<td>Multiple jets</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Nozzle to target spacing</td>
<td>Inert particles</td>
</tr>
</tbody>
</table>

Table 1.1 Summary of parameters studied in this work
In what follows is a brief overview of the chapters included in this thesis. Chapter 2 presents a short literature review on impinging jet heat transfer. The model and the numerical simulation scheme adopted in this work are presented in Chapter 3. Chapters 4-6 deal with single impinging slot jets, while Chapter 7 investigates single slot impinging jet with cross-flow and Chapter 8 presents the results on multiple impinging slot jets flow with gas particle suspensions. Finally, conclusions of this work are presented in Chapter 9.
Chapter 2

Literature review

2.1 Introduction

Numerous studies on the hydrodynamics and the heat and mass transfer characteristics of jet impingement in various configurations have been reported in the literature. A review of the flow, heat and mass transfer characteristics under impinging jets is presented in this chapter. A summary of the different configurations of impinging jet factors studied in literature is presented in Table 2.1, while several available review articles are summarized in Table 2.2. Initially, a review of the numerical studies on impinging jet flow and heat transfer is presented in section 2.2, followed by the studies which cover both the experimental and numerical analysis of flow and heat transfer behavior of impinging jet in section 2.3. The literature on the heat transfer characteristics of the gas-particle suspension flow is discussed in section 2.4.

Table 2.1 Summary of the configurations studied in literature (see pages 32-33 for nomenclature)

<table>
<thead>
<tr>
<th>Types of Study</th>
<th>Numerical</th>
<th>Experimental</th>
</tr>
</thead>
<tbody>
<tr>
<td>Configuration</td>
<td>[Images of configurations]</td>
<td>[Images of configurations]</td>
</tr>
<tr>
<td>Jets</td>
<td>Steady or pulse jets; Newtonian fluids (low and high Prandtl number); non-Newtonian fluid; Single phase or multiphase</td>
<td></td>
</tr>
<tr>
<td>Data collected</td>
<td>V; Nu; Sh; turbulence; pressure; k; Recovery factor</td>
<td></td>
</tr>
</tbody>
</table>
Table 2.2 List of general reviews on jet impingement

<table>
<thead>
<tr>
<th>Year</th>
<th>Authors</th>
<th>General Review on the Topic</th>
</tr>
</thead>
<tbody>
<tr>
<td>1987</td>
<td>Downs, S.J. and E.H. James</td>
<td>Heat transfer under round jet impingement</td>
</tr>
<tr>
<td></td>
<td>Polat, S. et al.</td>
<td>Numerical flow and heat transfer under impinging jets</td>
</tr>
<tr>
<td>1993</td>
<td>Viskanta, R.</td>
<td>Heat transfer under single and multiple isothermal turbulent air and flame jets impinging on surfaces</td>
</tr>
<tr>
<td>1995</td>
<td>Webb, B.W. and C.F. Ma</td>
<td>Single phase liquid jet impingement heat transfer</td>
</tr>
<tr>
<td>1995</td>
<td>Mujumdar, A.S. and B. Huang</td>
<td>Impingement drying</td>
</tr>
</tbody>
</table>

2.2 Numerical studies of impinging jets heat transfer

Many researchers simulated the turbulent impinging jet using different turbulent models, e.g. one-equation models, two-equation models, Reynolds Stress Model (RSM), Large Eddy Simulation (LES) and Direct Numerical Simulation (DNS). However, these studies show significant deviations among themselves and also from the relevant experiments.

2.2.1 Studies by various k-ε models and Reynolds Stress Model

The k-ε model is most widely used in various flow and heat transfer problems. The standard k-ε model is not applicable in the vicinity of the solid walls in which the viscous effect is neglected. To handle the near wall flows, the following two procedures have been used.

1. Use the low-Re model instead of the high-Re k-ε model;

2. Use high-Re k-ε model together with a separate model to treat the near-wall boundary.
In the low-Re $k$-$\varepsilon$ model, some additional terms are included in the $k$ and $\varepsilon$ equations to account for the effects of viscous diffusion of $k$ and $\varepsilon$ and nonisotropy due to the wall. Patel et al. (1985) gave a review of the comparison of 8 low-Re $k$-$\varepsilon$ models. The frequently used low-Re $k$-$\varepsilon$ models in impinging jet flow problems are Launder-Sharma (referred to as LS)(1974); Lam-Bremhorst (referred to as LB)(1981), Abid (referred to as AB) (1993), Fan et al. (referred to as FLB) (1993) and Abe et al. (referred to as AKN)(1994).

2.2.1.1 Single slot impinging jet

Seyedein et al. (1994) carried out a numerical investigation of single confined impinging slot jets using various low-Re and high-Re turbulence $k$-$\varepsilon$ models. The parameters studied were jet Reynolds number and nozzle-to-target spacing, the values of which ranged from $5000 < \text{Re} < 20,000$ and $2.5 < H/W < 7.5$, respectively. The results of simulation using low-Re number turbulence models presented by LB (1981) and LS (1974) exhibited very good agreement with the available experimental data. However, the standard $k$-$\varepsilon$ model underestimated the Nusselt number. The accuracy of the results of the standard $k$-$\varepsilon$ model depended on both the model parameters and near-wall treatment. Hosseinalipour and Mujumdar (1995) numerically investigated the flow and heat transfer characteristics of two-dimensional turbulent confined impinging slot jet flow. Five low-Reynolds $k$-$\varepsilon$ models and standard high-Reynolds numbers $k$-$\varepsilon$ model were used. A proposed Yap correction with low-Re model was tested and it was found that this correction improved the heat transfer predictions in some models.
2.2.1.2 Multiple slot impinging jets

The flow and heat transfer characteristics of multiple slot impinging jets were also investigated by various low Reynolds number k-\(\varepsilon\) models and high Reynolds numbers k-\(\varepsilon\) model (Shiravi et al. (1995), Seyedein et al. (1995) and Tzeng et al. (1999)). Shiravi et al. (1995) found that high Reynolds number turbulence models failed to predict heat transfer accurately although they predicted flow field reasonably well. On the other hand, the low Reynolds number models predicted considerably better results for both fluid flow and heat transfer. While Seyedein et al. (1995) found that the LB model over-predicted the normalized heat transfer coefficient \(N_{\text{u},x}\), the standard high Reynolds number model under-predicted it. The ranges of Re and H/W used in this work were the same as those in their previous work on single slot impinging jet (see 2.2.1.1) (Seyedein et al. (1994)). However, it was noted that there was a difference in performance of the LB low Re k-\(\varepsilon\) for single slot jet and multiple slot jets. More recently, Tzeng et al. (1999) pointed out that the prediction by each turbulence model depended on grid distribution and numerical scheme adopted in the work.

2.2.1.3 Impinging round jets

Knowles (1996) numerically studied the single and multiple impinging round jets flow by the k-\(\varepsilon\) turbulence model with Rodi and Malin corrections. It was found that both the Rodi and Malin corrections tended to improve the prediction of the hydrodynamic field of free and impinging jets but there were still significant errors in the predicted wall jet growth. Dianat et al. (1996) predicted the axisymmetric impinging jet flow using the standard k-\(\varepsilon\) turbulence model and RSM turbulence model, which included the effect of pressure reflections from a solid surface. Comparison of the predictions with the experimental data demonstrated the superiority of the RSM model where
observed mean velocities and shear and normal stresses were reproduced accurately. On the contrary, the standard k-ε model did not predict the experimental observations accurately.

Craft et al. (1993) studied an axisymmetric turbulent impinging jet flow and heat transfer by a k-ε eddy viscosity model and three second-moment closures (RSM). It was found that the k-ε model and one of the Reynolds stress models predicted large levels of turbulence near the stagnation point. This excessive energy in turn resulted in much too high heat transfer coefficients and turbulent mixing with the ambient fluid. The other two second-moment closures accounted for the wall’s effect on pressure fluctuations and performed better. But none of the methods was entirely successful in predicting the effects of Reynolds number, which varied from 23,000 to 70,000. More recently, Shuja et al. (1999) investigated an axisymmetric impinging jet by four turbulence models, which were low Reynolds number k-ε model, high Reynolds number k-ε model and two Reynolds stress models. The agreement between the temperature profiles predicted by both the low Reynolds number k-ε model and RSM turbulence models was better than that obtained from the standard k-ε model. The reason was that the standard k-ε model predicted excessive kinetic energy generation in the vicinity of the stagnation region, which in turn, resulted in excessive heat transfer and the lowering of the temperature in this region.

Park and Sung (2001) developed a near-wall turbulence model called k-ε-\(f_\mu\) to study the fluid flow and heat transfer for an axisymmetric impinging jet flow. The \(f_\mu^2\) function was newly formulated to derive a realizable eddy viscosity. The model performance was validated by available experimental data and compared with those by
the k-\(\varepsilon\) and k-\(\varepsilon\)-v\(^2\) models. It was found that the predicted results from k-\(\varepsilon\)-f\(_\mu\) model agreed well with the available experimental data, while the k-\(\varepsilon\) model over-predicted heat transfer and the k-\(\varepsilon\)-v\(^2\) model slightly over-predicted the Nusselt number. Behnia et al. (1997 and 1999) numerically investigated heat transfer in an axisymmetric turbulent jet impinging on a flat plate by the normal-velocity relaxation turbulence model (V2F model). Local heat transfer coefficient predictions were compared to the available experimental data and also to the predicted results by the standard k-\(\varepsilon\) model. It was found that the V2F heat transfer predictions were in excellent agreement with the experiments while the k-\(\varepsilon\) model greatly over-predicted the heat transfer rate. Results also indicate the effect of confinement is limited to very low nozzle-to-plate distances and confinement leads to a decrease in the average heat transfer rates, but the local stagnation heat transfer coefficient is unchanged.

Morris et al. (1996) numerically predicted the local heat transfer coefficient distribution on a square heat source due to an axisymmetric, confined and submerged liquid impinging jet using the standard k-\(\varepsilon\) model with different turbulent Prandtl number functions. The predicted heat transfer coefficients were compared with available experimental data and it was found that the predicted stagnation and average heat transfer coefficients agreed well with the experimental data within a deviation of 16 to 20%, respectively. Morris et al. (1999) investigated the flow field of a normally impinging, axisymmetric, confined and submerged liquid jet by Reynolds stress model (RSM) using FLUENT. The computed flow patterns were in good agreement with the experimental measurements. The predictions of the flow field using the standard k-\(\varepsilon\) and RNG k-\(\varepsilon\) models were shown to be inferior to the predictions of Reynolds stress model.
2.2.1.4 Impinging jet with cross-flow

Mujumdar et al. (1985) predicted heat transfer under an axisymmetric plane turbulent impinging jet including the effect of cross-flow and wall motion by $k$-$\varepsilon$ model. It was found that the effects of cross-flow and wall motion on the local Nusselt number and wall shear stress distributions were significant. The local wall shear stress and Nusselt number in impingement region decreased with the increase of the cross flow. Chuang et al. (1992) simulated twin-jet impingement with cross flow by Jones-Launer (1972) $k$-$\varepsilon$ model. The computed velocity without cross-flow was compared with the published data and good agreement was observed. The simulations showed that the strength of cross-flow had a strong influence on the pressure distributions of the lower and upper plates and on the lift of the flow. Kim and Benson (1992) calculated a three-dimensional turbulent flow of a jet in a cross flow using a multiple-time-scale turbulence model (MS turbulence model). It was found that the turbulent transport of mass, concentration and momentum was strongly governed by the non-equilibrium turbulence in which the turbulent transport of mass and momentum was described using the time scale of large eddies and the dissipation rate was described using the time scale of fine-scale eddies. The calculated flow and concentration fields were in good agreement with the measurement.

Sarkar and Bose (1995) presented a comparison of performance of different turbulence models for the predictions of flow and temperature fields created by the interactions of jet and cross flow for film cooling. The low Reynolds number $k$-$\varepsilon$ model and $k$-$\omega$ models, the algebraic Baldwin-Lomax (1978) model and also a relaxation eddy viscosity model were used to simulate the fine-scale turbulence. Low Reynolds number $k$-$\varepsilon$ model seemed to perform better compared to others in view of both the
predicted surface temperature distribution and the relaxation behavior of the velocity. He et al. (1999) discussed the effect of Schmidt number on turbulent scalar mixing in a jet-in-cross-flow using the standard k-ε turbulence model. It was noted that the turbulent Schmidt number had a significant effect on the spreading rate of the species in jet-in-cross-flow, especially for the cases where the jet-to-cross-flow momentum flux ratios were relatively small. A turbulent Schmidt number of 0.2 was recommended for best agreement with the experimental data. More recently Kalita et al. (2002) numerically predicted the flow field of turbulent plane jets discharged normal to a weak or moderate stream by the standard k-ε model. The agreement between the predicted results and available experimental data was found to be satisfactory.

2.2.2 Studies with LES and DNS approaches

The large-eddy simulation (LES) method integrates the three-dimensional (3-D) time-dependent Navier-Stokes equations directly to resolve the large eddies, while the small eddies that cannot be resolved on the grid are represented by a sub-grid model. Direct numerical simulation (DNS) does not rely on any empirical model and is able to solve all the physical scales of flows. As both direct and large-eddy simulations contain time-dependent information, they can give a much more realistic picture of the turbulence than traditional methods. However both LES and DNS are limited by the computational resource because both of them are time-consuming methods. Recently, with the development of high speed computer, a growing number of researchers are using LES and DNS models.
Gao and Voke (1995) presented the results of large-eddy simulation (LES) of thermally inhomogeneous jets issuing into an enclosed pool and impinging on a plate. The LES predictions of mean quantities and low-order statistics were in sufficient agreement with the experimental data. Voke and Gao (1998) numerically studied the heat transfer from an impinging jet using LES. It was found that the lateral heat conduction within the plate did not have any significant effect on the transmission of thermal fluctuations from the fluid in the plate and by this means a simple one-dimensional model of the thermal, interaction between the media can be justified. Chattopadhyay et al. (1999, 2000 and 2001) reported turbulent flow field and heat transfer from an array of impinging slot jets on a moving surface by large-eddy simulation (LES). The performance of a horizontal knife jet with an exit angle of 60° was compared with standard jet (Chattopadhyay et al. (2000 and 2001)). Nusselt number distributions for varying surface velocity were presented and it was found that increasing velocity of the impingement plate reduced the heat transfer for both types of jets and distribution of Nusselt number over the impingement surface became more uniform with the increasing velocity of the impingement surface. Cziesla et al. (2001) also simulated the flow field due to an impinging jet from a rectangular slot nozzle using LES technique. The computed results compared favorably with the experimental observation, especially in the stagnation zone.

To understand the detail flow feature of the impinging round jet, Satake and Kunugi (1998) carried out numerical simulation on a round turbulent impinging jet using the DNS approach for Reynolds number of 10,000. The results in the downstream region were in fairly good agreement with the available experimental data. It was also found that the wall-layer streaks were extended in the radial direction. More recently, Chung
et al. (2002) used direct numerical simulations (DNS) to study momentum and heat transfer characteristics in an unsteady impinging jet. Detailed analysis of the instantaneous flow and temperature fields were performed and showed that the impingement heat transfer was very unsteady, and the unsteadiness was caused by the primary vortices emanating from the jet nozzle. The correlation between the local heat transfer and the flow field was examined.

Some studies on the impinging jet flow and heat transfer with high Prandtl number fluids were conducted numerically. Generally, the flow with high Prandtl number is in the laminar region, thus, the laminar model is always applied. Rahman et al. (1999) presented the numerical simulation results of a free impinging jet with high Prandtl number fluids. The solid and fluid regions were solved as a conjugate problem. The influence of different operating parameters such as jet velocity, heat flux, plate thickness, nozzle height, and plate material was investigated. Similarly, Bula et al. (2000) numerically investigated the conjugate heat transfer from discrete heat sources to a two-dimensional jet of a high Prandtl number fluid discharging from a slot nozzle. The effects of the heat flux, jet Reynolds, physical properties of the substrate material, location and power of the discrete heat sources on the maximum substrate temperature, temperature variation at the solid-fluid interface, local and average heat transfer coefficients and local and average Nusselt numbers were studied. It was found that besides jet Reynolds number, plate thickness and its thermal conductivity had significant effect on temperature distribution and average Nusselt number. Tan (2001) developed the heat transfer coefficient correlations for low viscosity fluids based on his simulated data by CFD software, FLUENT. Chatterjee et al. (2002) numerically studied a confined axisymmetric impinging flow heat transfer with a purely viscous
inelastic fluid. Important features of the non-Newtonian developing flow field were described and contrasted with the Newtonian flow field. The effects of nozzle-to-target spacing, the rheological parameters, jet Reynolds number and Prandtl number on the off-stagnation point peak heat transfer rate were discussed.

2.2.3 Studies using other models

Gibson and Harper (1997) studied an axisymmetric turbulent impinging jet heat transfer with the low-Reynolds-number $q$-$\xi$ turbulence model. The new variables were the square root of the temperature variance and its dissipation rate and these variables were attractive for eddy-diffusivity calculations. It was found that the results by the $q$-$\xi$ low-Reynolds number eddy-viscosity model were better than those by the corresponding $k$-$\varepsilon$ model. However, $q$-$\xi$ low-Reynolds number eddy-viscosity model still has some deficiencies. This model predicted high levels of kinetic energy and square root of the temperature $q$. It was shown that the use of the equivalent $q_\infty$-$\xi_\infty$ model for heat transfer produced dramatic improvements in the stagnation Nusselt number. Fujumots et al. (1999) investigated the convective heat transfer between a circular free surface impinging jet and a solid surface numerically. The effects of surface tension, viscosity, gravity and heat transfer between the film flow and the solid surface were taken into account, but the turbulence was neglected. The steady-state flow on non-heated surface was examined first with experimental data and it was found that the predicted flow structure agreed reasonably well with the experimental data. Then the simultaneous flow and heat transfer were studied. It was shown that although the local Nusselt numbers were over-predicted in the stagnation region, the calculated Nusselt number agreed well with the experimental data in the downstream region.
Careful analysis of boundary layer at impingement surface was necessary for accurate determination of wall shear stress and heat and mass transfer rates under an impinging jet due to the large pressure gradients near the stagnation point. In order to provide the boundary conditions, Phares et al. (2000) modeled the inviscid impingement of a jet with arbitrary velocity profile. Expressions for the stream function were derived in terms of the vorticity function distribution. The method was applied to flow calculations for various two-dimensional and axisymmetric impinging jet configurations. The calculations showed excellent agreement with previous experimental and numerical results. Abdon and Sunden (2001) numerically investigated heat transfer of a single round unconfined impinging air jet under different flow and geometrical conditions using linear and nonlinear two-equation turbulence models, which are the various $k$-$\omega$ and $k$-$\varepsilon$ turbulence models. The results by different linear and nonlinear two-equation turbulence models were compared and discussed.

2.3 Studies using both experimental and numerical methods

Catalano et al. (1989) investigated the flow of a turbulent impinging round jet with cross flow by both measurement using Laser-Doppler anemometry and simulation employing two-equation $k$-$\varepsilon$ turbulence model. Good agreement between experimental data and simulated results was obtained in the downstream region, but only fair agreement occurred in the initial region. It was found that the jet trajectory and the existence of impingement were strongly dependent on the velocity ratio. Liu et al. (1991) analytically and experimentally investigated the impingement cooling of unsubmerged, circular liquid jet. The predictions were found to agree well with the measurements for both laminar and turbulent flows. The effects of Prandtl number
were investigated and predictive correlations for Nu were developed for the entire range of the radii of the jet.

Barata et al. (1992) characterized the effect of jet-to-crossflow velocity ratio on the mean and turbulent velocities by Laser-Doppler anemometry for single round impinging jet. The experimental data were used to examine the predictive capability of standard k-ε model, particularly in the immediate vicinity of the stagnation point. It was found that the shear stress was not predicted correctly in the impingement zone. Ashforth-Frost and Jambunathan (1996) numerically investigated the turbulent flow and heat transfer behavior under an axisymmetric impinging jet using standard k-ε model in conjunction with logarithmic law of the wall as well as by experimental method using Laser-Doppler anemometry and liquid crystal thermography. It was found that the stagnation point heat transfer was over-predicted by about 300%, which was due to the inapplicability of the wall function.

Chen and Chalupa’s research group (2000, 2001) experimentally and numerically investigated high Schmidt-number mass transfer in impinging slot jet for laminar and turbulent flows, respectively. Both the experimental and theoretical results showed that the peak values in Nusselt number occurred at 1-1.5 times and 1 time of the nozzle width (W), for laminar and turbulent flow, respectively. Prakash et al. (2001, part I, II) reported the studies on impinging round jet in a cylindrical enclosure with and without a porous layer. The effect of a porous layer on flow patterns was investigated. The part I of their work presented the flow visualization experiments and comparisons with CFD simulations. Part II presented laser Doppler velocimetry measurements for the same system and comparisons of these measurements with the CFD simulations to
evaluate the mathematical model. Two low-Reynolds number k-ε turbulence models and a laminar model for the porous medium were used. The turbulent kinetic energy profiles in the fluid layer showed good agreement with predictions by all three models for the porous medium.

Roy et al. (2002) experimentally and numerically studied the heat transfer between an impinging jet and an inclined surface. 3D numerical simulations by RNG k-ε turbulence model for heat transfer were validated by the experimental results. The effect of different turbulence levels on Nusselt number in the numerical solution was documented on six specific lines on heat pad.

2.4 Heat transfer in turbulent gas-particle suspension flow

Avila et al. (1995) numerically predicted an isothermal particulate pipe flow by a Eulerian-Lagrangian mathematical model. The turbulence of air flow was calculated by the standard k-ε model. The predicted average heat transfer coefficients at the inner wall of a vertical pipe with spherical glass particles were compared with the published experimental data. It was found that they over-predicted the average heat transfer coefficients for small particle size. Sharma (1997) experimentally investigated the relative contributions from particle conduction and gas convection to the heat transfer coefficient between gas-solid fluidized beds and surfaces. 18 different fluidization systems were studied including five different materials, twelve different particle sizes and two distributor plates. The proportions of the particle conductive contribution to surface heat transfer coefficient were found to be over a wide range.
Sato et al. (1998) used Eulerian-Lagrangian method to investigate the mechanism of two-phase heat and turbulent transport by small solid particle suspended in a gas flow. The DNS model was used to solve the continuous phase. The simulation of the dispersed phase was achieved using a Lagrangian model. They tested the effect of fluid mean temperature gradient on heat transfer between dispersed and gas phases. It was found that DNS model was able to simulate the heat transfer between the gas and the solid particles and also the heat and turbulent transport in the dispersed phase. Nguyen et al. (1999) numerically investigated the heat transfer process between a dilute gas-particle suspension flow and an obstruction using their novel Eulerian formulation by taking into consideration the particle incidence and reflection explicitly. It was noted that the particle size and the concentration affected the heat transfer and there was an optimum particle size for maximum enhancement of the heat transfer.

Murray (1994) studied heat transfer characteristics in a cross-flow situation. His study indicates that the main mechanisms by which the particles affected heat transfer in gas-particle flow are the increased effective thermal capacity of the suspension, conduction between impacting particles and the tube wall, thermal energy transport by rebounding particles and changes in the boundary-layer characteristics and turbulent structure of the gas flow. Li (2000) numerically studied heat transfer in gas-particle flows through pipes using a two-dimensional coupled CFD and discrete element method (DEM) model. A quantitative estimate of the proportions contributed to the total heat transfer rate by each of the above four heat transfer mechanisms was presented. It was reported that conduction heat transfer due to particle-wall collisions had negligible effect on heat transfer. Its proportion was less than 0.05% in the case of a horizontal pipe flow.
Boulet et al. (2000) used the Eulerian-Lagrangian model to numerically predict the heat transfer between a vertical pipe wall and a turbulent gas-particle suspension. A k-ε model coupled with a particle tracking method was used. Heat transfer by conduction due to particle-particle interactions and particle-wall interactions was neglected. The results were compared with available experimental data, and they found that although the tests indicated encouraging results, the model became unsatisfactory as loading ratio increased. Bourloutski et al. (2002) compared two theoretical approaches (viz. Eulerian-Eulerian modeling and Eulerian-Lagrangian approach) for the numerical investigation of turbulent gas-solid flows with heat transfer in a pipe. The comparison of the theoretical approaches showed good predictive properties of the theoretical tool for research activities to study new models of turbulent gas-particle suspension flow. In very recent studies Mansoori et al. (2002a, 2002b) studied the turbulent heat transfer in vertical gas-particle flows and proposed new models allowing for the two-way and four-way interactions of two-phase flows following the Eulerian-Lagrangian approach. Their simulation results were in good agreement with the available experimental data. The simulations indicated that the levels of thermal turbulence intensity and heat transfer were strongly affected by particle collisions while the inter-particle contact conduction heat transfer during collisions had no significant effect in the range of Reynolds number and particle diameter studied. Rozenblit et al. (2000) investigated the heat transfer coefficient of solid-liquid pipe flow using an electro-resistance sensor and infra-red imaging. It was found that the local value of the heat transfer coefficient was influenced by the distribution of the solid phase in the pipe cross section and the average heat transfer increased with particle concentration.
Although there are numerous numerical studies on the influence of particles on the heat transfer for channel flows, very little research has been reported on impingement flows of gas-particle suspensions. Hosseinalipour and Mujumdar (1986) numerically studied the flow and heat transfer in confined opposing jets of particle suspensions using Eulerian-Lagrangian models. Yoshida et al. (1990) investigated the turbulence structure and heat transfer mechanism for a two-dimensional impinging jet with gas-particle suspensions using laser-Doppler anemometry. The results of Yoshida et al. (1990) showed that with increase of loading ratio, the Nusselt number markedly increased in the vicinity of the stagnation point. More recently, Yokomine et al. (2002) experimentally and numerically investigated the heat transfer mechanism of multiple impinging jets of gas-particle suspensions. They carried out heat transfer experiments and evaluated the effects of nozzle Reynolds number, solid loading ratio, distance from jet exit to impingement surface, spacing between jets, and solid particle characteristics on the heat transfer coefficient. Their experimental results showed that in the case of graphite powder, the heat transfer coefficient increased greatly compared with that obtained for a single phase gas flow. Numerical simulations were also performed using the Eulerian-Lagrangian approach; conduction heat transfer due to the particle-wall collisions was neglected. The turbulence of gas was calculated by the low-Reynolds number k-ε model. The predicted Nusselt numbers did not agree well with their experimental data; the results matched only qualitatively.

Besides the numerical studies of impinging jet flow and heat transfer, many researchers investigated the impinging jet flow and heat transfer by experimental methods. Table 2.3 is a summary of experimental studies on impinging jet flow and heat transfer limited to the recent 15 years. The studies on flow behavior of gas-
particle suspension flow are fundamental to the studies of heat transfer characteristics in gas-particle suspension flow. Thus, Table 2.4 summarizes the studies on flow behavior of gas-particle suspension flow.

Table 2.3 A summary of the experimental studies of impinging jet flow and heat transfer

<table>
<thead>
<tr>
<th>Author</th>
<th>Configurations and Range of Operation</th>
<th>Variables Reported</th>
<th>Objectives</th>
</tr>
</thead>
<tbody>
<tr>
<td>Das et al. (1985)</td>
<td>SSJ; W = 6mm; H/W = 5, 8, 10, 12; Re = 1000-20,000; T_{i} - T_{w} = 50-300K</td>
<td>Nu_{x}, Nu_{ave}</td>
<td>Heat transfer under large ΔT; correlation of Nu_{ave}</td>
</tr>
<tr>
<td>Polat et al. Part I (1991)</td>
<td>SSJ; H/W = 2.5; 16,400 &lt; Re &lt; 57,700</td>
<td>Nu_{ave}, Nu_{x}</td>
<td>Effect of throughflow on impingement heat transfer</td>
</tr>
<tr>
<td>Polat et al. Part II (1991)</td>
<td>SSJ; H/W = 2.5; Re = 18,100-35,400; Mvs = 0.03-0.35</td>
<td>Nu_{x}; Nu_{ave}; position of off-stagnation maxima</td>
<td>Effects of surface motion and throughflow on heat transfer</td>
</tr>
<tr>
<td>Stevens and Webb (1991)</td>
<td>Single obliquely round free liquid jet; D = 4.6 and 9.3mm; θ = 40-90° Re = 6600-52,000</td>
<td>Nu_{x}; correlation of Nu and Nu_{max}; Effect of θ on upstream shift of the point of Nu_{max}</td>
<td>Effect of jet inclination on heat transfer for various Re</td>
</tr>
<tr>
<td>Gau and Chung (1991)</td>
<td>SWJ on concave and convex surfaces; W = 0.35, 0.6, 1, 1.5 &amp; 2 cm; D_{c}/W = 8-45.7; H/W = 2-12; Re = 6000-35,000</td>
<td>Photos of flow field; Nu_{x}; Nu_{0}; correlation of Nu_{0} and Nu_{ave}</td>
<td>Effects of surface curvature on impingement cooling flow and heat transfer</td>
</tr>
<tr>
<td>Saad et al. (1992)</td>
<td>SSJ&amp;MSJ; W = 2.5-13.3 mm; H/W = 4-30; S/W ≤ 24; Re ≤ 21,720</td>
<td>Fluctuating velocity; mean velocity; static pressure; stagnation pressure; Nu_{0}; Nu_{x}</td>
<td>Comparison turbulence, mean flow and heat transfer characteristics of SSJ and MSJ</td>
</tr>
<tr>
<td>Copper et al. (1993)</td>
<td>SRJ, D = 26, 106 mm; H/D = 2-10; Re = 23,000-70,000</td>
<td>Mean and turbulent velocity; Nu_{x}; turbulent shear stress</td>
<td>Data suitable for assessing the engineering models</td>
</tr>
<tr>
<td>Hansen &amp; Webb (1993)</td>
<td>SRJ on modified surfaces with arrays of fin-type extensions. H/D = 0.25-14; D = 6.91-13.3 mm; Re = 4700-33,000</td>
<td>Nu_{ave} vs Re for various surfaces; Nu_{ave} vs H/D for various surfaces</td>
<td>Evaluation of the heat transfer enhancement for various surfaces</td>
</tr>
<tr>
<td>Mohanty et al. (1993)</td>
<td>SRJ; H/D = 4-58;</td>
<td>Nu_{x}; Nu_{0}; Nu_{x}/Nu_{0};</td>
<td>Heat transfer</td>
</tr>
<tr>
<td>Authors</td>
<td>Nozzle Type</td>
<td>Parameters</td>
<td>Heat Transfer Characteristics</td>
</tr>
<tr>
<td>---------</td>
<td>-------------</td>
<td>------------</td>
<td>--------------------------------</td>
</tr>
<tr>
<td>Tawfek (1993)</td>
<td>SRJ</td>
<td>D = 3, 5 and 7mm; Re = 4860-34,500</td>
<td>$h_x$</td>
</tr>
<tr>
<td>Lytle and Webb (1994)</td>
<td>SRJ</td>
<td>H/D = 0.1, 0.15, 0.2, 0.25, 0.5, 0.75, 1.0 and 6.0; Re = 3600-27,600</td>
<td>Mean velocity; Static pressure; $Nu_0$; $Nu_x$</td>
</tr>
<tr>
<td>Huber et al. (1994)</td>
<td>MRJ</td>
<td>S/D = 4, 6 and 8; H/D = 0.25, 1.0 and 6.0; Re = 3500-20,400</td>
<td>$Nu_x$; $Nu_{ave}$</td>
</tr>
<tr>
<td>Lee et al. (1994)</td>
<td>An elliptical nozzle on a flat plate; AR = 2.14; H/D = 2, 4, 6 and 10; Re = 5000, 10,000 and 20,000</td>
<td>$Nu_x$; $Nu_0$; Isothermal contour</td>
<td>Nu for an elliptical jet. Found that Nu for elliptical jet in impingement region was larger than that for a circular jet.</td>
</tr>
<tr>
<td>Gaimella et al. (1995)</td>
<td>SRJ; FC-77; 0.79 mm &lt; D &lt; 6.35 mm; 1 ≤ H/D ≤ 14; Re = 4000 to 23,000; $q_w$ = constant;</td>
<td>$h_x$; $Nu_0$; $Nu_{ave}$; sketches of pathline; correlations of $Nu_{0f}$ and $Nu_{avef}$</td>
<td>Local heat transfer rate distributions and correlations of $Nu_{0f}$ and $Nu_{avef}$</td>
</tr>
<tr>
<td>Liu and Sullivan (1996)</td>
<td>SRJ</td>
<td>D = 12.7mm; H/D ≤ 2; $q_w$ = 2756W/m²; Re = 12000-15000</td>
<td>$Nu_0$, $Nu_x$(2D&amp;3D); $\eta$; photos of flow field</td>
</tr>
<tr>
<td>Colucci and Viskanta (1996)</td>
<td>SRJ; Free jet &amp; IJ; D = 0.25&lt;H/D&lt;6.0; Two hyperbolic nozzles; Re = 10,000, 50,000</td>
<td>$\Delta P$; $Nu_x$ for various H/D, Re and nozzle geometries</td>
<td>Effects of hyperbolic nozzle geometry on local heat transfer coefficients</td>
</tr>
<tr>
<td>Garimella and Nenaydykh (1996)</td>
<td>SRJ; D = 0.79-6.35mm; H/D = 1-14; Re = 4000-23,000</td>
<td>$h_x$; $h_0$; $h_{ave}$; $Nu_x$; correlations of $Nu_{0f}$</td>
<td>Influence of nozzle geometry on local heat transfer on a small, square heat source</td>
</tr>
<tr>
<td>Nishino et al. (1996)</td>
<td>SRJ, D = 40 mm; H/D = 5.86 and 5.63; Re = 13,000 and 10,400;</td>
<td>Flow axisymmetry mean velocity; axial mean momentum; $\bar{I}$; turbulent shear stresses</td>
<td>Various turbulence statistics properties.</td>
</tr>
<tr>
<td>Lee et al. (1997)</td>
<td>SRJ, D = 1.3, 2.15, 3.4cm H/D = 2-10; Re = 11,000 to 50,000</td>
<td>Mean velocity; $\bar{I}$; pressure coefficient; $Nu_x$; Correlations of $Nu_0$ and $Nu_{ave}$</td>
<td>Effects of geometric parameters and Re on heat transfer; Correlations of $Nu_0$ and $Nu_{ave}$</td>
</tr>
<tr>
<td>Author(s)</td>
<td>Experiment Description</td>
<td>Key Parameters</td>
<td>Heat Transfer Parameters</td>
</tr>
<tr>
<td>----------------------------</td>
<td>----------------------------------------------------------------------------------------</td>
<td>----------------</td>
<td>--------------------------</td>
</tr>
<tr>
<td>Lin et al. (1997)</td>
<td>SSJ; H/W = 1-8; W = 5mm; Re =190-1537</td>
<td>Mean velocity; Nu&lt;sub&gt;x&lt;/sub&gt;; correlations of Nu&lt;sub&gt;0&lt;/sub&gt; and Nu&lt;sub&gt;ave&lt;/sub&gt;</td>
<td>Parametric effects of Re, H/W on heat transfer behaviors</td>
</tr>
<tr>
<td>Yan and Saniei (1997)</td>
<td>Obliquely impinging round jet; θ = 90°-45°; H/D = 2, 4, 7 and 10; Re = 10,000-23,000</td>
<td>Velocity; I; shift of Nu&lt;sub&gt;max&lt;/sub&gt;, Nu&lt;sub&gt;x&lt;/sub&gt;</td>
<td>Heat transfer of obliquely impinging jet. Effect of θ on heat transfer</td>
</tr>
<tr>
<td>San et al. (1997)</td>
<td>SRJ, D = 3, 4, 6 &amp; 9mm H/D = 2; Re =30,000-67,000; qw = 500-2000W/m²</td>
<td>Nu&lt;sub&gt;x&lt;/sub&gt;; Nu&lt;sub&gt;0&lt;/sub&gt;; T&lt;sub&gt;j&lt;/sub&gt;-T&lt;sub&gt;w&lt;/sub&gt;</td>
<td>Effect of Re, D, qw on heat transfer</td>
</tr>
<tr>
<td>Ashforth-Frost et al. (1997)</td>
<td>SSJ; H/W = 4 and 9.2; W = 30mm Re = 20,000</td>
<td>Nu&lt;sub&gt;x&lt;/sub&gt;/Nu&lt;sub&gt;0&lt;/sub&gt;; mean velocity; surface pressure; spanwise velocity; spanwise turbulence</td>
<td>Velocity and turbulence characteristics under above conditions</td>
</tr>
<tr>
<td>Ma et al. Part I (1997)</td>
<td>SRJ; Liquid jet with high Pr number; Pr = 263-270; H/D = 2-20; Re = 220-1500</td>
<td>T&lt;sub&gt;w&lt;/sub&gt;; r; Nu&lt;sub&gt;0&lt;/sub&gt;; Nu&lt;sub&gt;x&lt;/sub&gt;; Nu&lt;sub&gt;x&lt;/sub&gt;/Nu&lt;sub&gt;0&lt;/sub&gt;; Nu&lt;sub&gt;ave&lt;/sub&gt;/Nu&lt;sub&gt;0&lt;/sub&gt;</td>
<td>Heat transfer for submerged impinging jet with large Pr</td>
</tr>
<tr>
<td>Ma et al. Part II (1997)</td>
<td>SSJ; Liquid jet with high Pr number; Pr = 200-270; W = 0.091, 0.146 and 0.234 mm; Re = 55-415</td>
<td>Nu&lt;sub&gt;0&lt;/sub&gt;; r; Nu&lt;sub&gt;x&lt;/sub&gt;/Nu&lt;sub&gt;0&lt;/sub&gt;; Nu&lt;sub&gt;x&lt;/sub&gt;/Pr&lt;sup&gt;1/3&lt;/sup&gt;; Nu&lt;sub&gt;ave&lt;/sub&gt;/Nu&lt;sub&gt;0&lt;/sub&gt;</td>
<td>Heat transfer for submerged impinging jet with large Pr for tiny slot nozzles</td>
</tr>
<tr>
<td>Ma et al. (1997)</td>
<td>Oblique round free surface jets with large Pr number liquid; H/D = 4-12; θ = 90°-45°; Re = 235-1745</td>
<td>Nu&lt;sub&gt;max&lt;/sub&gt;/Pr&lt;sup&gt;1/3&lt;/sup&gt;; r; shift of Nu&lt;sub&gt;max&lt;/sub&gt;; Nu/Nu&lt;sub&gt;max&lt;/sub&gt;</td>
<td>Heat transfer for submerged impinging jet with large Pr for oblique round free jet</td>
</tr>
<tr>
<td>Haneda et al. (1998)</td>
<td>SSJ, H/W = 3 and 5; W =15mm; A cylinder was set between the plane and jet; Dv = 2, 3, 4, 6 and 8 mm; Re = 10000</td>
<td>Frequent velocity; Mean pressure; Nu&lt;sub&gt;x&lt;/sub&gt;; flow field photo</td>
<td>Effects of suspended cylinder; Effectively enhanced heat transfer outside the stagnation point</td>
</tr>
<tr>
<td>Fitzgerald and Garimella (1998)</td>
<td>SRJ confined and submerged turbulent jet FC-77, D=3.18 and 6.35 mm; H/D =2, 3, 4</td>
<td>Streamlines centerline velocity; radial velocity; I;</td>
<td>Flow field of a confined and submerged impinging jet</td>
</tr>
<tr>
<td>Study</td>
<td>Description</td>
<td>Methodology</td>
<td>Results</td>
</tr>
<tr>
<td>------------------</td>
<td>------------------------------------------------------------------------------</td>
<td>----------------------------------------------------------------------------</td>
<td>------------------------------------------------------------------------</td>
</tr>
<tr>
<td>Chen et al.</td>
<td>SRJ on a rotating disk; ( \omega = 500-4000 \text{rpm} ) ( H/D = 5 ), ( \text{Re} = 2000-100000 )</td>
<td>Shr</td>
<td>Heat (mass) transfer of a rotating disk with an impinging round jet</td>
</tr>
<tr>
<td>Cornaro et al.</td>
<td>SRJ on concave, convex and flat surfaces; ( D = 47.2, 72.6 ) &amp; 98.6 mm; Free jet and impinging jet; ( H/D = 1, 2, 3, 4 ); ( I = 0.15-16% ) ( \text{Re} = 6000-20000 )</td>
<td>Axial velocity; ( I ); photos of flow visualization</td>
<td>Flow behavior; Effects of relative curvature, ( D ), ( H/D ) and ( \text{Re} ) on flow structure</td>
</tr>
<tr>
<td>Yang et al.</td>
<td>Impingement cooling on a concave surface; 3 slot nozzles, 1 round nozzle, 1 rectangular nozzle and 1 2D contoured nozzle; ( 5920 \leq \text{Re} \leq 25500 )</td>
<td>Mean velocity; ( \text{Nu}<em>0 ); ( \text{Nu}</em>{ave} ); comparison of ( \text{Nu}_x ), ( \text{Nu}<em>0 ) and ( \text{Nu}</em>{ave} ) with other literature data</td>
<td>Jet impingement cooling on concave surface for different nozzle geometries. Effects of nozzle shapes, curvature, ( \text{Re} ) and ( H/D ) on heat transfer</td>
</tr>
<tr>
<td>Lee et al.</td>
<td>Round jet on concave surface; ( D = 1.3, 2.15 ) and 3.4cm; ( D/D_s = 0.034, 0.056 ) and 0.089; ( \text{Re} = 11000, 23000 ) and 50000</td>
<td>( \text{Nu}_0 ); ( \text{Nu}_x )</td>
<td>Effect of concave surface curvature on heat transfer</td>
</tr>
<tr>
<td>Failla et al.</td>
<td>MRJ on finned surface with controlled crossflow; ( H = 1, 3, 5, 7 ); ( W = 3.23 \text{mm} ); ( \text{Re} \leq 4700 )</td>
<td>Various dimensionless heat transfer rate vs various parameters (flow power; ( \text{Re} ); pressure drop) ( \text{Nu} ) correlations</td>
<td>Overall heat transfer under conditions: only IJ, only crossflow and combined jet with crossflow for several geometries</td>
</tr>
<tr>
<td>Sailor et al.</td>
<td>Pulsed SRJ on a heated surface; ( D = 14 \text{mm} ); ( L = 51 \text{mm} ); ( DC = 0.25, 0.33, 0.5 ); ( 0.009 &lt; St(f) &lt; 0.042 ); ( \text{Re} = 21000, 26000, 31000 )</td>
<td>Radial velocity; Centerline velocity ( \text{Nu}_x ) vs frequency (Hz)</td>
<td>Effects of ( H ), ( \text{Re} ), pulse frequency and duty cycle on ( \text{Nu}_x )</td>
</tr>
<tr>
<td>Lee et al.</td>
<td>A air round jet impinging perpendicularly on convex hemispherical surface; ( D = 2.87 \text{cm} ); ( H/D = 2-10 ), ( D/D_s = 10.6 ); ( \text{Re} = 11000-87000 )</td>
<td>( \text{Nu}_x ); ( \text{Nu}_0 ) vs ( H/D ); ( \text{Nu}_0 ) vs ( \text{Re} ); ( \text{Nu}_0 ) correlations</td>
<td>Heat transfer from a convex hemispherical surface to a round IJ</td>
</tr>
<tr>
<td>Yapici et al.</td>
<td>Single submerged round jet; ( H/D = 2-10 ); ( D = 8 \text{mm} ); ( \text{Re} = 9200-73500 )</td>
<td>Surface shear stresses</td>
<td>Effect of experimental parameters on shear stresses.</td>
</tr>
<tr>
<td>Lee and Lee</td>
<td>An elliptic impinging jet</td>
<td>Flow visualization;</td>
<td>Heat transfer</td>
</tr>
<tr>
<td>Year</td>
<td>Study</td>
<td>Description</td>
<td>Parameters</td>
</tr>
<tr>
<td>------</td>
<td>-------</td>
<td>-------------</td>
<td>------------</td>
</tr>
<tr>
<td>2000</td>
<td>Inoue et al.</td>
<td>Planar jet to concave surface with high heat flux</td>
<td>$Rc=24.8, 62.1, \alpha , mm$; $V_j=5, 8, 12 , m/s$; Subcooling $\Delta T_{sub}=30, 60$ and $80 , K$</td>
</tr>
<tr>
<td>2000</td>
<td>Choi et al.</td>
<td>SSJ on a semi-circular concave surface</td>
<td>$Re=150 , mm$; $H/W=0.2-14; W=5 , mm$; $Re=1780-7100$</td>
</tr>
<tr>
<td>2000</td>
<td>Beitelmal et al.</td>
<td>Single inclined 2D slot impinging air jet</td>
<td>$H/2W=4-12; W=5.5 , mm$; $\theta=90^\circ-40^\circ; Re=4000-12000$</td>
</tr>
<tr>
<td>2001</td>
<td>Li et al.</td>
<td>Single confined submerged impinging jet; air, FC-77 and water; $Pr=0.7-25.2$; $D=1.59-12.7 , mm$; $H/D=1-5$; $Re=4000-23000$</td>
<td>Correlations of $Nu_0$ and $Nu_{ave}$ for water, FC-77, air, liquids and all fluids; $h_x$ distributions; Influence of thermophysical properties on heat transfer</td>
</tr>
<tr>
<td>2001</td>
<td>Zhe and Modi</td>
<td>SSJ; $H/W=2-9.2$; $W=40 , mm$; $Re=10000-30000$</td>
<td>Mean velocity and root mean square velocity; Normal stresses; skin friction coefficient</td>
</tr>
<tr>
<td>2002</td>
<td>Tawfek</td>
<td>Single oblique round jet on a curved surface</td>
<td>$D=3, 5, 7 , mm$; $H/D=7-30$; $Re=3800-40000$; $\theta=90^\circ-20^\circ$</td>
</tr>
<tr>
<td>2002</td>
<td>Lee et al.</td>
<td>A swirling round turbulent jet impinging on a flat-plate</td>
<td>$H/D=2-10$; $D=3.4 , cm$; swirl number=0-0.77; $Re=23000$</td>
</tr>
<tr>
<td>2002</td>
<td>Lee et al.</td>
<td>SRJ with perforated plate installed between the jet</td>
<td></td>
</tr>
</tbody>
</table>
### Chapter 2

<table>
<thead>
<tr>
<th>Study</th>
<th>Setup</th>
<th>Variables</th>
<th>Findings</th>
</tr>
</thead>
<tbody>
<tr>
<td>Heikkila and Milosavljevic</td>
<td>Round jet; nozzle geometry resembling those used in Yankee hoods</td>
<td>Various temperature distributions; Heat transfer coefficients</td>
<td>Impingement heat transfer under large temperature difference</td>
</tr>
<tr>
<td>(2002)</td>
<td>$T_{studied}=100^\circ C-700^\circ C$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Guo and Wood (2002)</td>
<td>SRJ, H/D=2, 4, 5</td>
<td>Mean velocity; stresses; pressure-containing terms; spectral densities;</td>
<td>Measurements in the vicinity of stagnation point</td>
</tr>
<tr>
<td></td>
<td></td>
<td>plate pressure; shear stress</td>
<td></td>
</tr>
<tr>
<td>Ekkad and Kontrovitz (2002)</td>
<td>MRJ; 48 impingement holes of 0.635cm diameter $H/D=3$; $S/D=4$; $Re=4800-14800$</td>
<td>$Nu_x/Nu_0; Nu_{ave}/Nu_0$</td>
<td>Effect of dimple location, underneath the jets or between the jets</td>
</tr>
<tr>
<td>Bart et al. (2002)</td>
<td>Multiple jet; $H/D=4$ and 6; $Re=10000$</td>
<td>$Nu_x$</td>
<td>Effects of mechanical modulation of a jet assembly</td>
</tr>
</tbody>
</table>

| D:                           | jet diameter.                                                                                   |                                                                           |                                                                          |
| Dc:                          | outer diameter of the hemisphere.                                                               |                                                                           |                                                                          |
| DC:                          | duty cycle of flow field (ratio of on time to total cycle time).                                |                                                                           |                                                                          |
| dh:                          | hole diameter on perforated plate; $p_h/d_h$: pith-to-hole diameter.                           |                                                                           |                                                                          |
| Dv:                          | the diameter of cylinder.                                                                     |                                                                           |                                                                          |
| I:                           | turbulent intensity.                                                                           |                                                                           |                                                                          |
| MSJ:                         | multiple slot impinging jets.                                                                  |                                                                           |                                                                          |
| MRJ:                         | multiple round impinging jets.                                                                 |                                                                           |                                                                          |
| $Nu_{st}$ and $Nu_{ave}$:    | stagnation and average Nusselt number with properties evaluated at film temperature.       |                                                                           |                                                                          |
| $r$:                         | recovery factor, $r=\frac{(T_w - T_f)}{V_j^2/2C_p}$.                                           |                                                                           |                                                                          |
| Rec:                         | surface curvature $R$; AR-elliptic nozzles ratio.                                              |                                                                           |                                                                          |
SSJ: single slot impinging jet.
SRJ: single round impinging jet.
S/D or S/W: dimensionless spacing between jets.

$W_f$: fin width.
$\eta$: heat transfer enhancement factor $\log(Q/Q_0)$.
$\omega$: rotational speed of disk.

### Table 2.4 Studies on gas-particle flow

<table>
<thead>
<tr>
<th>Author</th>
<th>Configuration and Operation Parameters</th>
<th>Methods</th>
<th>Abstract</th>
</tr>
</thead>
<tbody>
<tr>
<td>Arastoopour (2001)</td>
<td>gas/solid flow system</td>
<td>Numerical and experimental methods</td>
<td>A review</td>
</tr>
<tr>
<td>Sato et al. (1994)</td>
<td>Vertical channel flow with soft magnetic particles; Lo=0.13, 0.7 (downward flow) and 0.18 (upward)</td>
<td>Experimental and simulation by high-Re k-ε model and Lagrangian simulation</td>
<td>Turbulence modification was investigated. The turbulence attenuation or augmentation level in 2-phase dilute flows is correlated with the modified particle Re.</td>
</tr>
<tr>
<td>Arastoopour and Hea (1994)</td>
<td>Steady-state 2D dilute gas-particle flow</td>
<td>Numerical method based on kinetic theory using conservative variables, the orthogonal collocation method and the non-linear minimization technique</td>
<td>1. A computer code based on the kinetic theory to simulate steady state 2D gas-particle flow was developed. 2. Predicted gas and particle V, mass flux and concentration agreed reasonably well with available experimental data</td>
</tr>
<tr>
<td>Cao and Ahmadi (1995)</td>
<td>Gas-particle turbulent flow in a vertical duct</td>
<td>Simulation by a thermodynamically consistent turbulent 2-phase flow model using two-equation k-ε model for turbulence analysis</td>
<td>1. Gas-particle flows at various Lo between two vertical parallel plates. 2. The predicted V and I were compared with available experimental data. 3. The phase fluctuation energy, and its production and dissipation and interaction momentum and energy supply terms</td>
</tr>
<tr>
<td>Study</td>
<td>Description</td>
<td>Methods</td>
<td>Findings</td>
</tr>
<tr>
<td>-------------------------------</td>
<td>-----------------------------------------------------------------------------</td>
<td>----------------------------------------------------------------------------------------------</td>
<td>-----------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>Sato et al. (1996)</td>
<td>Vertical down-flow turbulent wall jet embedded in a uniform stream with</td>
<td>Both experimental and numerical methods: Laser-Doppler; Eulerian-Lagrangian (with k-ε model)</td>
<td>1. Motion of small particles was influenced by strong shear in the developing region; 2. Streamwise $I$ was strongly attenuated by particles in the free shear layer region; 3. Transverse $I$ was suppressed in the fully-developed region of both the free and wall shear regions</td>
</tr>
<tr>
<td>Levy, et al. (1997)</td>
<td>Gas-particle flow through an inclined section of pipe</td>
<td>Analytical approach and numerical model by EE simulation: k-ε model was used</td>
<td>1. Flow characteristics by both analytical and numerical models were predicted. 2. The agreement between the experimental data and models was satisfactory.</td>
</tr>
<tr>
<td>Zhao and Brodkey (1998)</td>
<td>Particle paths in 3D opposing jet mixing system</td>
<td>Experimental method: PTV technique</td>
<td>1. Mixing process was Analyzed. 2. Experimental velocity field is determined by using neutrally buoyant particles. 3. The effect of gravity can be easily studied.</td>
</tr>
<tr>
<td>Nguyen and Fletcher (1999)</td>
<td>Gas-particle flow</td>
<td>Numerical simulation by Eulerian-Lagrangian model using FLUENT</td>
<td>1. Incidence and reflection of small particles. 2. The boundary layer theory is employed to predict the key model parameters. 3. The analytical prediction agrees well with the computational results.</td>
</tr>
<tr>
<td>Schiewe et al. (1999)</td>
<td>Downward vertical gas-solid flow; $D=15$ cm; $D_p=125\mu m$, glass bead; $V_g=0$-$6.6m/s$</td>
<td>Experiment by two measurement techniques: gamma-absorption tomography and capacitance sensing</td>
<td>1. Solid concentration. 2. Good agreement was obtained between the solid concentration measurements from both measurement techniques.</td>
</tr>
<tr>
<td>Sommerfeld and Huber (1999)</td>
<td>Particle-laden horizontal channel flow; $D_p=100$, $500\mu m$; glass bead; quartz particles;</td>
<td>Experimental and numerical methods</td>
<td>1. Particle-wall collisions were studied in detail. 2. The effect of wall material on the collision process was analyzed. 3. All the parameters for the wall collision model were assumed as function of impact angle and were obtained from experiments</td>
</tr>
<tr>
<td>Tanlbee et al.</td>
<td>Homogeneous turbulent shear</td>
<td>Eulerian- Eulerian simulation:</td>
<td>1. New RSM model for the simulation of gas-particle flow</td>
</tr>
<tr>
<td>Year</td>
<td>Study Description</td>
<td>Methods</td>
<td>Findings</td>
</tr>
<tr>
<td>----------</td>
<td>-----------------------------------------------------------------------------------</td>
<td>-------------------------------------------------------------------------</td>
<td>----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>1999</td>
<td>DNS and RSM; Lagrangian frame;</td>
<td>was developed; DNS was used to determine the magnitudes of the empirical constants in RSM. Final model predictions were compared with DNS data.</td>
<td></td>
</tr>
<tr>
<td>Cao and Ahmadi (2000)</td>
<td>Gas-particle turbulent flow in horizontal and inclined ducts</td>
<td>Simulation by two-phase flow model closed by low-Re turbulence closure</td>
<td>Predicted V and solid volume fraction were compared with experimental data. Fluctuation energy; fluctuation energy production; dissipation and shear stress were discussed; effect of Dp on flow properties was presented.</td>
</tr>
<tr>
<td>Triesch and Bohnet (2001)</td>
<td>An upstream gas solids flow in pipes and diffusers</td>
<td>Measurement by PDA technique and simulation by standard k-ε model and Lagrangian particle tracking using FLUENT.</td>
<td>Good agreement was obtained between Exp. &amp; Numerical data if the additional models available in FLUENT. Individual influence of added model was discussed. The improved version of FLUENT was applied to various Lo.</td>
</tr>
<tr>
<td>Mashayek and Taulbee (2002)</td>
<td>Dilute gas-particle turbulent flow</td>
<td>Simulation by a four-equation model</td>
<td>The model was based on explicit algebraic relations for Reynolds stress and turbulence fluxes of the void fraction. Encouraging agreements with available experimental data</td>
</tr>
<tr>
<td>Lain et al. (2002)</td>
<td>Particle-laden horizontal channel flow</td>
<td>Measurement using phase-Doppler anemometry; numerical calculations based on Eulerian-Lagrange approach</td>
<td>Comparison of experiment and numerical data was presented for different Dp and Lo; The agreement was found to be reasonable for both mean and fluctuating velocities.</td>
</tr>
</tbody>
</table>
Chapter 3

Numerical simulation

The focus of this study is the investigation of the flow and heat transfer characteristics of two-dimensional, steady state semi-confined turbulent impinging slot jets. The commercial CFD software, FLUENT, was used in conjunction with its user defined function option to solve the governing conservation equations and turbulence models as required. In most of the simulations, the default options of FLUENT have been used, except for the cases with gas-solid heat transfer where a user-defined subroutine was deployed to introduce a new heat transfer model to account for conductive heat transfer due to particle collisions with the impingement wall. The governing equations describing the flow and heat transfer along with the appropriate boundary conditions in such systems are described below.

3.1 Single phase laminar flow

The two-dimensional, laminar incompressible steady flow and heat transfer in semi-confined impinging slot jets can be described by the following continuity, momentum and energy equations. Under the assumptions that all properties of the working fluids remain constant, viscous dissipation and buoyancy are neglected; the continuity, momentum and energy equations for a Newtonian fluid in the dimensionless form can be written as follows:

\[
\frac{\partial U}{\partial X} + \frac{\partial V}{\partial Y} = 0
\]  
(3.1)
Chapter 3

\[ U \frac{\partial U}{\partial X} + V \frac{\partial U}{\partial Y} = -\frac{\partial P}{\partial X} + \frac{1}{\text{Re}} \nabla^2 U \] (3.2a)

\[ U \frac{\partial V}{\partial X} + V \frac{\partial V}{\partial Y} = -\frac{\partial P}{\partial Y} + \frac{1}{\text{Re}} \nabla^2 V \] (3.2b)

\[ U \frac{\partial T}{\partial X} + V \frac{\partial T}{\partial Y} = -\frac{1}{\text{Re Pr}} \nabla^2 T \] (3.3)

In these equations, for a slot jet, \( \text{Re} = \frac{U W}{v} \) and \( \text{Pr} = \frac{\mu C_p}{\lambda} \), where \( v \) is the kinematic viscosity and \( \mu \) is the dynamic viscosity.

### 3.2 Single phase turbulent flow

#### 3.2.1 The governing equations

The partial differential equations, including conservation of mass, momentum, and energy for a time-dependent three dimensional fluid flow and heat transfer of a compressible Newtonian fluid are expressed in the following form.

**Mass**

\[ \frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i} (\rho u_i) = 0 \] (3.4)

**Momentum**

\[ \frac{\partial}{\partial t} (\rho u_i) + \frac{\partial}{\partial x_j} (\rho u_i u_j) = -\frac{\partial P}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} + \rho g_i + F_i \] (3.5)

**Internal energy**

\[ \frac{\partial (\rho \cdot e)}{\partial t} + u_i \frac{\partial (\rho \cdot e)}{\partial x_i} = -p \frac{\partial u_i}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \lambda \frac{\partial T}{\partial x_j} \right) + \mu \cdot \Phi \] (3.6)

Modeling of turbulent flows requires describing the effect of turbulent fluctuations of velocity and scalar quantities on mean flow and energy quantities. In turbulent flows, the velocity, pressure and temperature at a point are considered as a sum of the mean and fluctuating components.
Substituting equation (3.7) into equations (3.4), (3.5) and (3.6) and taking the assembly averages of resulting equations for an incompressible Newtonian fluid, the governing equations become:

Continuity:
\[
\frac{\partial U_i}{\partial x_i} = 0
\]  

Momentum balance:
\[
0 = \frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \rho u_i u_j \right]
\]  

Energy balance:
\[
0 = \frac{\partial}{\partial x_j} \left( \frac{\mu}{Pr} \frac{\partial T}{\partial x_j} - \rho u_j T' \right)
\]

In the averaged equations, Reynolds stresses, \(\overline{u_i u_j}\), and the turbulent heat fluxes, \(\overline{u_i T'}\), are required to be determined to address the effects of them on mean flow and energy quantities.

3.2.2 Models of turbulence

In order to characterize \(\overline{u_i u_j}\) and \(\overline{u_i T'}\), many turbulence models have been proposed in the literature. In this work, we use the standard k-\(\varepsilon\) model and Reynolds stress model to close the above governing equations (3.8 to 3.10). These models are selected mainly due to their frequent successful applications in this field, and also in the light of the past experience of our research group.
3.2.2.1 The standard k-ε model

In the k-ε model, the turbulence kinetic energy, $k$, and the dissipation rate of the turbulence kinetic energy, $\varepsilon$, can be determined by using the Boussinesq approximation, which states that the Reynolds stresses are directly proportional to the velocity gradients with an eddy turbulent viscosity, $\mu_t$, which is correlated with $k$ and $\varepsilon$ via the expression $\mu_t = \rho C_\mu k^2/\varepsilon$, where $C_\mu$ is the constant of the k-ε model. The turbulent kinetic energy, $k$, and its dissipation rate, $\varepsilon$, are obtained from the following transport equations:

$$\rho \frac{Dk}{Dt} = \frac{\partial}{\partial x_i} \left[ \left( \mu + \frac{\mu_t}{\mu_k} \right) \frac{\partial k}{\partial x_i} \right] + G_k + G_b - \rho \varepsilon - Y_M \quad (3.11)$$

and

$$\rho \frac{D\varepsilon}{Dt} = \frac{\partial}{\partial x_i} \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_i} \right] + C_1 \varepsilon \frac{\varepsilon}{k} (G_k + C_3 G_b) - C_2 \rho \frac{\varepsilon^2}{k} \quad (3.12)$$

In these equations, $G_k$ and $G_b$ are the generations of turbulent kinetic energy due to the mean velocity gradients and buoyancy, respectively. $Y_M$ 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 and equal to 1.44, 1.92 and 0.09, respectively. $\sigma_k = 1.0$ and $\sigma_\varepsilon = 1.3$ are the turbulent Prandtl numbers for $k$ and $\varepsilon$, respectively.

3.2.2.2 RSM model

The Reynolds stress model uses the exact transport equations for the transport of the individual Reynolds stresses and turbulent fluxes. A particular advantage of deriving the exact equations is that these terms overcome some of the limitations of the k-ε model and account for buoyancy, rotation and other effects inherently present in turbulent flows. The RSM model may be written as follows:
The detailed descriptions of the transport equations for individual Reynolds stresses and turbulent fluxes appearing in eq. 3.13 are provided following the models of Gibson and Launder (1978), Launder (1989) and Launder et al. (1975).

Both of the above two turbulence models are only valid at high local turbulence Reynolds number; therefore they cannot be used in the viscous sub-layer. Hence this region has to be bridged by so-called wall functions to the main flow, as described in section 3.4.

3.3 Multiphase flow

The multiphase (gas-solid) flow can be described by two possible theoretical approaches. One is called Eulerian-Eulerian modeling, which is based on two-fluid assumptions. The other is Eulerian-Lagrangian approach, in which the particles are regarded as the discrete phase and the fluid as the continuous one. In this work, we use the Eulerian-Lagrangian approach to solve the multiphase flow. The mathematical model for the two-way interaction of gas-particle flow in a Eulerian-Lagrangian approach is presented in the following section. The basic assumptions used in our mathematical formulation for the gas-solid flow are the following:

1. The flow is steady
2. The gas is incompressible
3. Each particle has uniform temperature and constant characteristics
4. Particles are solid, spherical, with fixed diameter
5. The heat transfer by conduction due to particle-particle interactions is negligible

Under above assumptions, the conservation equations for the flow and heat transfer in gas and suspended particles can be modeled as follows:

### 3.3.1 Governing equations

The turbulent gas flow is governed by the mass, momentum and energy equations, which are described in the previous section by equations (3.8), (3.9) and (3.10). In the simulation of gas-solid flow, we used the standard $k$-$\varepsilon$ model (equations (3.11) and (3.12)) to close the governing equations.

The solid phase is described by a Lagrangian particle tracking approach in the simulation. The trajectory of the suspended particle can be determined by integrating the differential force balance equation for the particle. The suspended particle in a gas flow is subjected to aerodynamic, gravitational, buoyancy, contact, thermophoretic, Brownian as well as the Saffman’s lift forces. These forces can be calculated simultaneously with the particle motion using the local parameters of the gas and the particles. In this work, the thermophoretic force, Brownian force and Saffman’s lift force are assumed to be negligible. The force balance on single particle can be written as,

$$\frac{dU_p}{dt} = F_D (U_g - U_p) + g(\rho_p - \rho_g) / \rho_p$$  \hspace{1cm} (3.14)
where $F_D (U_g - U_p)$ is the drag force per unit particle mass, and

$$F_D = \frac{18 \mu_g C_D \text{Re}}{\rho_p D_p^2} 24$$

(3.15)

Re is the relative Reynolds number, which is defined as

$$\text{Re} = \frac{\rho_g D_p |U_p - U_g|}{\mu_g}$$

(3.16)

where $C_D$ is the drag coefficient. When a particle strikes a wall, it is assumed that it will bounce from the surface. The rebound velocity of the particle from the wall is evaluated using the classical inelastic collision equations. The thermal energy equation of the particles is given as:

$$m_p c_p \frac{dT_p}{dt} = h A_p \left(T_g - T_p\right) + \varepsilon_p S_p \sigma \left(T^4_p - T^4_{ip}\right)$$

(3.17)

3.3.2 Gas-particle interaction

The influence of particles on the flow and heat transfer field of the continuous phase is considered by the momentum and heat transfer for each particle. The momentum transfer is computed by examining the change in momentum of a particle as it passes through each control volume. Similarly, the heat transfer rate is also computed by examining the change in thermal energy of a particle as it passes through each control volume. The momentum change and heat exchange are computed by equations 3.18 and 3.19, respectively.

$$F = \sum \left(\frac{18 \mu C_D \text{Re}}{\rho_p D_p^2} 24 \left(U_p - U_g\right)\right) m_p \Delta t$$

(3.18)

$$Q = \left[ \frac{m_p}{m_{p0}} c_p \Delta T_p + \frac{\Delta m_p}{m_{p0}} \left(\int_{t_{ip}}^{t} c_p dT\right) \right] m_{p0} + S$$

(3.19)
The source term $S$, which represents the conductive heat transfer when particle and impingement wall impact, will be discussed in the following section. These momentum and heat exchange terms appear as source or sink in the continuous phase momentum balance and energy balance during any subsequent calculations of the continuous phase flow field.

### 3.3.3 Particle-wall conduction heat transfer

When moving particles impact on the impingement surface, heat conduction occurs. Sun and Chen (1988) theoretically analyzed the heat transfer due to particle-wall impact. They modelled the conduction heat transfer due to particle-wall collisions from the elastic collision equation. From the elastic collision theory, the rate of the area change for the compression process is

$$
\frac{dA}{dt} = \left[ \left( \pi \cdot v_p \cdot R_{pw} \right)^2 - \frac{4}{5\sqrt{\pi}} \frac{E_{pw}}{m_{pw}} A^{5/2} \right]^{1/2}
$$

(3.20)

where

$$
R_{pw} = \frac{R_p R_w}{R_p + R_w}
$$

(3.21)

$$
E_{pw} = \frac{4/3}{1 - \varphi_p^2 + 1 - \varphi_w^2} \frac{E_p}{E_w}
$$

(3.22)

and

$$
m_{pw} = \frac{m_p m_w}{m_p + m_w}
$$

(3.23)

The decompression process is exactly the inverse of the compression process. The maximum contact area $A_c$ and its radius $r_c$ are found by setting the derivative to zero.

$$
A_c = \pi \cdot r_c^2 = \pi \left( \frac{5m_{pw} R_{pw}^2}{4E_{pw}} \right)^{2/5} v_p^{4/5}
$$

(3.24)
The dependence of contact area $A$ on time can best be seen with the following dimensionless variables:

$$A^* = \frac{A}{A_c} = \frac{r^2}{r_c^2} \quad (3.25)$$

$$\tau \equiv \left( \frac{4E_{pw}}{5m_{pw}} \right)^{2/5} \left( R_{pw} v_p \right)^{2/5} t \quad (3.26)$$

where $r$ and $r_c$ are radii of $A$ and $A_c$, respectively. Equation (3.20) can then be integrated for the compression to yield

$$\tau = \int_0^{r_c} \frac{dx}{\left(1 - x^{2/5}\right)^{1/2}} \quad (3.27)$$

The total contact duration is then found to be

$$t_c = 2.94 \left( \frac{5m_{pw}}{4E_{pw}} \right)^{2/5} \left( R_{pw} v_p \right)^{2/5} \quad (3.28)$$

The heat conduction between the two particles is treated as that between two semi-infinite media. Within the contact area, perfect thermal contact is assumed, that is, there is no thermal resistance between the contact surfaces. Therefore, the problem is governed by two axi-symmetric heat conduction equations with appropriate boundary and initial conditions.

When the contact duration $t_c$ is very small, the heat conduction in the two media will not penetrate too deeply from the surfaces. This suggests that one-dimensional heat transfer between the two media may be a good approximation for this situation. Under the above condition, the solution of above equation is
\[ q_w = \frac{(T_w - T_p)(\pi \cdot t)^{1/2}}{(\rho_p c_{pp} k_p)^{1/2} + (\rho_w c_{pw} k_w)^{1/2}} \]

where \( t \) is elapsed time measured from the moment of initial contact, which is a function of \( r \). The total energy exchange per unit contact area is thus a function of the local duration of contact, which is also a function of \( r \). Hence the total energy exchange per impact is

\[ e = 2\pi \int_0^r \int_0^{t_{1c}(r)} q_w \, dt \, dr \]

where the local contact duration \( t_{1c}(r) \) can be computed from

\[ t_{1c}(r) = t - 2t(r) \]

where \( t(r) \) is determined from equations (3.25)-(3.27). Integrating (3.30), the following was obtained

\[ e_{pw} = \frac{0.87(T_w - T_p)A_t t_{1c}^{1/2}}{(\rho_p c_{pp} k_p)^{1/2} + (\rho_w c_{pw} k_w)^{1/2}} \]

In each trajectory, the heat transferred per second is:

\[ S = Cn \frac{0.87(T_w - T_p)A_t t_{1c}^{1/2}}{(\rho_p c_{pp} k_p)^{1/2} + (\rho_w c_{pw} k_w)^{1/2}} \]

where \( n \) is the number of the particles impacted on the impingement surface per second and \( C \) accounts for inelastic collisions and particle shape effect and was introduced first in this work. The equations described in this section were incorporated in the user-defined subroutine of FLUENT 6 to incorporate the heat transfer due to conduction between the particles and the impingement surface.
3.4 Near wall treatment

When the flow to be computed involves walls, the wall affects the turbulent flows in the regions close to the walls. First, the presence of the walls changes the turbulence and the mean velocity field is affected by the no-slip condition. Because the near-wall modeling significantly impacts the reliability of the numerical solutions, accurate representation of flow in near-wall region determines successful prediction of wall-bounded turbulent flows.

To handle the near-wall flows two procedures have been suggested. These are:

1. For near wall modeling, use the low-Re k-ε model instead of the high Re k-ε model.

2. Use a separate model to treat the near-wall boundary

The first method was proposed by Jones and Launder and was successfully used by many researchers, for example, Seyedein et al. (1994). When low-Reynolds number versions of turbulence model are used, the iterations can be carried up to the wall. Some different models of turbulence also had been applied in literature. Patel et al. (1985) compared eight models in their review. In the second method, semi-empirical formula called ‘wall functions” are used to bridge the viscosity-affected region between the wall and the fully turbulent region. It accounts for the shortcomings of the high-Reynolds number k-ε model in the near wall region, where molecular viscosity affects the generation, destruction and transport of the turbulence energy. Since viscous effects are confined to the first grid point close to the wall, near wall functions have the advantage of reducing the grid density required in the near wall region. Figure 3.1 shows the near wall treatments. The wall function approach is popular because it is economical, robust and reasonably accurate. It is a practical option for the near-wall
treatments for industrial flow simulation. The second method using special function is available in FLUENT, which offers two choices of wall function approaches: a. Standard wall function; b. Non-equilibrium wall function.

\[
\begin{align*}
\rho \tau & \equiv \mu \frac{C_k^{1/4} k^{1/2}}{\tau_w / \rho} \\
\end{align*}
\]

*Figure 3.1 Near wall treatments*

The standard wall functions in FLUENT are based on the proposal of Launder and Spalding (1972), which are provided as a default option in FLUENT. The standard wall function introduces the logarithmic law of the wall to describe the velocity profile in absence of pressure gradient and mass transfer. Near the solid wall surfaces, no-slip conditions are imposed for flow analysis. They considered the first grid point close to the wall and assumed that the thin layer of the fluid close to the wall is in local equilibrium.

When the mesh is such that \(y^*<11.225\), at the wall-adjacent cells, the laminar stress-strain relationship is used as \(U^*=y^*\) \hspace{1cm} (3.34)
and
\[ y^* = \frac{\rho C_{\mu}^{1/4} k_B^{1/2} y_B}{\mu} \] (3.36)

That is
\[ \tau_w = \frac{U_B \mu}{y_B} \] (3.37)

The log-law is employed when \( y^* > 11.225 \),
\[ U^* = \frac{1}{k_v} \ln(9.81 y^*) \] (3.38)
\[ \tau_w = \frac{\rho U_B C_{\mu}^{1/4} k_B^{1/2}}{U^*} \] (3.39)

To analyze heat transfer, linear law is used for the thermal conduction sublayer where conduction is important and logarithmic law is used for the turbulent region where the turbulence effects dominate conduction. Generally, the thickness of the thermal conduction layer is different from the thickness of the viscous sublayer and changes with fluid. Once the physical properties of the fluid are specified, its molecular Prandtl number is computed. Then the thermal sublayer thickness, \( y_T^* \), is calculated from the intersection of the linear and logarithmic profiles. During the iteration, depending on the value of \( y^* \), either the linear or the logarithmic profile is applied to compute the wall temperature \( T_w \) or heat flux \( q \).

The production of kinetic energy; \( G_k \), and its dissipation rate, \( \varepsilon \), at wall-adjacent cell are computed from
\[ G_k \approx \tau_w \frac{\partial U}{\partial y} = \tau_w \frac{\tau_w}{k_v \rho C_{\mu}^{1/4} k_B^{1/2} y_B} \] (3.40)
and
\[ \varepsilon = \frac{C_{\mu}^{1/4} k_B^{1/2}}{k_v y_B} \] (3.41)
In addition to the standard wall function, a two-layer-based, nonequilibrium wall functions of Chieng and Launder (1980) are additional options available in FLUENT for the strong nonequilibrium flows. The nonequilibrium wall functions are described as follows:

a. Launder and Spalding’s log-law for mean velocity is sensitized to pressure gradient effects.

b. The two-layer-based concept is adopted to compute the budget of turbulent kinetic energy in the wall neighboring cells.

Details of the log-law for mean velocity sensitized to pressure gradients and the turbulent kinetic energy \((G_k, \varepsilon)\) are available in the manual of FLUENT. The law of the wall for mean temperature remains the same as in the standard wall function described above.

In conclusion, the standard wall functions give reasonably accurate predictions for most high-Reynolds number, wall-bound flows, and the non-equilibrium wall functions further extend the applicability of the wall function approach by considering the effects of pressure gradient and strong non-equilibrium.

3.5 Numerical techniques for turbulent impinging jet simulations

3.5.1 Boundary conditions

In this part, the boundary conditions needed for both laminar and turbulent jets are considered. Unless stated otherwise, the following boundary conditions were applied for the continuous phase in all cases simulated in this work:
• Impingement surface: the impingement surface was specified as an isothermal or a constant heat flux wall. The velocities satisfy the no-slip condition and temperature value is fixed at $T_w$ or the heat flux value is fixed at $q_w$.

• Confinement surface: the confinement surface is considered to be an adiabatic wall. The velocities also satisfy the no-slip condition and the heat flux value is fixed to be zero.

• Inlet: the inlet is considered as velocity inlet boundary conditions, at which uniform velocity and temperature profiles are prescribed. Unless otherwise stated, turbulence intensity and length scale at the inlet for all the calculations are chosen to be 2 % and 0.07 D (hydraulic diameter), respectively.

• Outlet: the outlet is considered as fully developed outflow. Thus, $V=0$ and the gradients of other variables are set to zero. Pressure outlet is another choice where the pressure in outlet is specified to be atmospheric and the temperature is equal to the ambient fluid temperature.

• Symmetry line: at the symmetry line, the axial velocity is set to zero and gradients of all other variables are equal to zero.

For multiphase cases, the particles were introduced in the inlet with the same velocity and temperature as the continuous phase. It should be emphasized that all particles were introduced to the flow field at 10 discrete positions. The ten different trajectory positions were determined by numerical experiments, which had shown that 10 trajectory positions were large enough and appropriate for the simulation. A larger number of locations (e.g. 20, 30 and 40) yielded the same results. The dispersion of particles due to gas turbulence is also considered using the statistical method. (FLUENT 6.0)
3.5.2 Numerical parameters

To solve the main variables $u$, $v$, $p$ and $T$, a control volume based on the finite difference method (CVPDM) was used to discretize the governing equations by integration over the control volume. The general form of the algebraic governing equations obtained from discretization can be written as follows:

$$
\alpha_p \Phi_p = \alpha_E \Phi_E + \alpha_W \Phi_W + \alpha_N \Phi_N + \alpha_S \Phi_S + b_p
$$

(3.42)

Where $\alpha$ is the coefficient of the convection and diffusion fluxes; $E$, $W$, $N$ and $S$ stand for the grid locations at the east, west, north and south sides of point $P$, and $b$ is the discretized source term. In order to ensure stable simulation, the momentum equations, energy equation, turbulence kinetic energy equation, turbulence dissipation rate equation and Reynolds stress equations were discretized in FLUENT using the first-order upwind interpolation scheme. The discretized equations were solved using the SIMPLEC algorithm of Patankar (1980). Simulations using higher order interpolation scheme produced the same results as the first-order upwind interpolation.

3.5.2.1 Relaxation factors

To promote smooth convergence of the discretized equations, proper and successive under-relaxation parameters for the variables should be used. For single phase flow, the typical relaxation factors were 0.3, 0.7 and 0.5 for pressure, momentum and Reynolds stresses. For energy, the relaxation factor was 1. For the turbulence kinetic energy, and turbulence dissipation rate, the relaxation factor was 0.8 in each case. For gas-particle suspension flow, to make the cases converge, lower relaxation factors were selected, and the typical relaxation factors used were 0.2 and 0.3 for pressure and discrete phase sources, respectively. For others, the relaxation factor was set at 0.5.
3.5.2.2 Convergence criteria

The convergence criteria commonly require the following aspects. Firstly, the overall mass balance must be satisfied. Secondly, each of the sums of the local residuals must be less than the prescribed level.

In this study, the solution was considered to be converged when the normalized energy residual was less than $10^{-6}$ and the normalized residuals of all other variables were less than $10^{-4}$ for the single phase flow, while for gas-particle suspension flow, the solution was considered converged when the normalized energy residual was less than $10^{-5}$ and the normalized residuals of all the other variables were less than $10^{-4}$.

3.5.2.3 Grid independence tests

The choice of grid layouts and density in the simulation domain is a very important aspect in numerical simulations. Generally, in the regions where the velocity or temperature gradients are large, the grid density is needed to be high. In this work, the flow and heat transfer behavior of single semi-confined impinging slot jet with and without cross flow and multiple impinging slot jets are studied. Here we use single semi-confined impinging slot jet without cross flow as an example to illustrate grid definition. This impinging slot jet flow configuration is symmetric, as shown in figure 1.1, therefore simulation of only half-domain is adequate for complete characterization of the flow. Several simulations involving the full domain were also conducted, which produced results in full agreement with the results of the half-domain as shown in Figure 3.2 and hence were not pursued any further. The impingement region is the most difficult domain to simulate accurately due to large gradients at streamline curvature. The grid layout in this region was specified to be much finer than that in the
downstream region. Figure 3.3 is simulation domain and definition of different regions. A non-uniform grid was used in all the simulations and a typical grid distribution is shown in Figure 3.4. Grid-independence tests were conducted for each new case. Figure 3.5 shows the results of one such test. The grid densities, $20 \times 80$, $26 \times 90$, $32 \times 90$ and $40 \times 100$ in Figure 3.5 refer only to the impingement region (Region A), and not to the whole region. Typically, a grid density of $20 \times 80$ in Region A yields grid-independent results for the example shown.

![Figure 3.2 Effect of simulation region on results](image.png)
**Figure 3.3** Definition of different regions in the computational domain

a: Typical grid structure

b: Second type of grid at the inlet area

**Figure 3.4** Examples of typical grid structure
Figure 3.5 Effect of grid size on surface heat transfer coefficient

Re = 12000, W = 7.5 mm, H/W = 12, T_j = 305 K, q_{in} = 3490 W/m^2
Chapter 4

Heat transfer under a turbulent impinging slot jet

4.1 Introduction

As mentioned in Chapter 1, impinging jets of various configurations are commonly used in numerous industrial applications such as drying of tissue, paper, textiles, photographic films, and cooling of high power density electronic components, due to their highly favorable heat and mass transfer characteristics. Despite an extensive body of research literature on the subject over the past two decades, impinging jet heat transfer remains an active area of research covering both the experimental and computational fluid dynamics aspects due to its inherently complex fluid and heat flow characteristics. This chapter presents simulation results for a single semi-confined turbulent slot jet impinging normally on a flat plate using FLUENT 5. Effects of various turbulence models, near wall functions, jet turbulence, jet Reynolds number as well as the type of thermal boundary condition at the target surface on heat transfer are discussed in the light of experimental data. The numerical simulation scheme adopted in this work was validated by comparing predicted Nusselt number distributions with experimental data from literature. Four sets of experimental results were selected from the numerous data available in the literature on the basis of how detailed the authors were in describing their experimental setup and procedures, and also the boundary conditions needed for simulation. The geometric parameters and boundary conditions of the selected experimental results are described in the following section in some detail.
For all of the cases studied here, besides the boundary conditions mentioned in Chapter 3, the fully developed outflow boundary condition was used at the outlet and the confinement surface was specified as an adiabatic wall.

### 4.2 Results and discussion

#### 4.2.1 Comparison of results from various turbulence models

The geometric parameters and the corresponding boundary conditions of the simulated cases for different turbulent models are presented in Table 4.1. The results are presented as the distribution of local Nusselt number in the stream-wise direction. The heat transfer coefficient over the impingement surface was normalized in the form of a local Nusselt number for an isothermal impingement surface as: \( \text{Nu}_x = \frac{h_x W}{\lambda} \), where

\[
h_x = \frac{q}{(T_w - T_j)}
\]

and \( \lambda \) is fluid thermal conductivity.

<p>| Table 4.1 Geometric parameters and boundary conditions for the two cases tested for turbulence models |
|---------------------------------|-----------------|-----------------|-----------------|-----------------|</p>
<table>
<thead>
<tr>
<th>W, (mm)</th>
<th>H/W</th>
<th>Re_j</th>
<th>T_j, (K)</th>
<th>Boundary Condition at Impingement Surface</th>
<th>Turbulence Intensity on Nozzle Exit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Case 1</td>
<td>6.2</td>
<td>6.0</td>
<td>11000</td>
<td>T_w=338 K</td>
<td>2 %</td>
</tr>
<tr>
<td>Case 2</td>
<td>14.1</td>
<td>2.6</td>
<td>10200</td>
<td>T_w=348 K</td>
<td>2 %</td>
</tr>
</tbody>
</table>
Figure 4.1 Comparison of simulated Nusselt number to the experimental data from literature for case 1

Figure 4.2 Comparison of simulated Nusselt number to the experimental data from van Heiningen for case 2

The computed local Nusselt numbers for two cases using the standard k-ε model and the Reynolds stress model are compared with the experimental data in Figures 4.1 and 4.2. Figure 4.1 compares the simulation results predicted by the two models with the
experimental data of Gardon and Akfirat (1966) from literature for case 1. Whereas Figure 4.2 compares simulation results predicted by the two models with the experimental data from van Heinigen (1982) for case 2. As can be seen from Figures 4.1 and 4.2, both standard k-ε and the Reynolds stress models slightly over-predict the Nusselt number in the stagnation region for both the cases, while greater difference (maximum of 20%) occurs for the standard k-ε model. The over-prediction of the Nusselt number at the stagnation zone is due to the high level of turbulence kinetic energy generation. The Nusselt number predicted by the Reynolds stress model reaches a maximum at X/W equal to 2 and 1.7 for case 1 and case 2, respectively, while the experimental maximum occurs at the stagnation point for the flow and geometric parameters shown in these figures.

For large nozzle-to-target spacing, H/W = 6, prediction by the standard k-ε model was better than the RSM model as shown in Figure 4.1, and both the models depict the qualitative trend of the Nusselt numbers very well. For small nozzle-to-target spacing, H/W = 2.6, van Heiningen measured two peak values in the local Nusselt number distribution (Figure 4.2). The first peak occurred at the stagnation point, while the second peak was located at X/W of about 6. In semi-confined impinging jet flows, a recirculation bubble develops in the off-stagnation region attached to the confinement surface, which forms a constricted channel with minimum area at the thickest section of the bubble, reducing the channel area and accelerating the fluid in this region. This causes an increase in the local heat transfer coefficient. This effect becomes appreciable only at smaller nozzle-to-target spacings as the experimental data show a clear second peak for H/W=2.6. Gardon and Akfirat (1966) and Polat et al. (1985) reported similar results for small H/W values. The RSM model shows a small
secondary hump but at a distance greater than the distance reported in the experimental work of van Heiningen (1982). Although generally the standard k-ε model gives a better prediction with respect to the shape of the Nusselt number distribution curve at least over a range of H/W and Re, it does not predict the secondary peak. The RSM model, on the other hand, predicts the numerical value of the secondary peak of Nusselt numbers well, but it does not predict the position of the secondary peak accurately.

4.2.2 Effect of turbulent Prandtl Number

Thakre and Joshi (2000) contended that even for simple flows such as the turbulent pipe flow, turbulent Prandtl number increases steeply and far exceeds unity as the wall is approached. The turbulent Prandtl number is defined as the ratio of the momentum to heat eddy diffusivities: \( D_t(T) = \nabla \cdot \left( \frac{\nu}{Pr} + \frac{\nu_l}{Pr_l} \nabla T \right) \) and hence must influence the predictions of heat transfer rates using both the standard k-ε and the RSM models. A systematic parametric study was carried out to examine the influence of Pr_l on the Nusselt number distributions. For Case 2, the turbulent Pr value was varied from 0.85 (default) to 1.5 based on Lai & So (1990b) and Kim & Moin (1987).
Figures 4.3 and 4.4 show the effect of turbulent Prandtl number on the computed Nusselt number along with the experimental data for case 2 as predicted by the standard k-ε model and the Reynolds Stress Model, respectively. Although, the standard k-ε model over-predicted the Nusselt number, the deviations from the
experimental data were reduced by increasing the turbulent Pr number and the fit with experimental data was much better away from the impingement zone. It seems that the turbulent Pr number affected Nusselt number to a greater extent in the downstream (wall jet) region than in the stagnation region for both the models tested. The Nusselt number decreased with the increase of turbulent Prandtl number for both standard k-ε and RSM models, but the magnitude of the turbulent Prandtl number did not change the shape of the Nusselt number profile and the position of the secondary peak of the Nusselt number.

4.2.3 Effect of Reynolds Number

Simulations were carried out for five Reynolds numbers for an impinging jet under constant heat flux conditions. The Reynolds number for the impinging jet flow is defined as, \[ \text{Re} = \frac{UL_j}{\mu}, \] where \( L \) is the width of the slot jet. The geometric parameters, boundary conditions, and the maximum surface heat transfer coefficients of the cases are presented in Table 4.2. Figure 4.5 shows the effect of the jet Reynolds number on the distribution of the surface heat transfer coefficients. The surface heat transfer coefficients predicted by both the standard k-ε model and the Reynolds stress model were compared with the experimental data of Bi (2001). As expected, the impingement surface heat transfer coefficients predicted by both the standard k-ε and the RSM model increased with the increase of jet Reynolds number. The agreement with the experimental data for both the models was much better for low \( \text{Re}_j \) (1500) values. However, interestingly, Bi’s data are unlike the results shown in Figures 4.1 and 4.2, as the simulated heat transfer coefficients are lower than the experimental values as shown in Figure 4.5. This is probably due to the fact that a
uniform velocity profile at the end of the jet was used in simulations whereas that was probably not the case in the experiments of Bi, where a nozzle with high level of turbulence was used.

Table 4.2 Geometric parameters, boundary conditions and maximum surface heat transfer coefficient for the test cases for different Reynolds numbers

<table>
<thead>
<tr>
<th>Case</th>
<th>W (mm)</th>
<th>H/W</th>
<th>Rej</th>
<th>Tj (K)</th>
<th>Boundary Condition at Impingement Surface (W/m²)</th>
<th>Turbulence Intensity at Nozzle Exit</th>
<th>Maximum Heat Transfer Rate (W/m²-K)</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>7.5</td>
<td>12</td>
<td>12000</td>
<td>305</td>
<td>3490</td>
<td>2 %</td>
<td>200</td>
</tr>
<tr>
<td>4</td>
<td>7.5</td>
<td>12</td>
<td>9000</td>
<td>305</td>
<td>2830</td>
<td>2 %</td>
<td>158</td>
</tr>
<tr>
<td>5</td>
<td>7.5</td>
<td>12</td>
<td>6000</td>
<td>305</td>
<td>1960</td>
<td>2 %</td>
<td>113</td>
</tr>
<tr>
<td>6</td>
<td>7.5</td>
<td>12</td>
<td>3000</td>
<td>305</td>
<td>1260</td>
<td>2 %</td>
<td>80</td>
</tr>
<tr>
<td>7</td>
<td>7.5</td>
<td>12</td>
<td>1500</td>
<td>305</td>
<td>870</td>
<td>2 %</td>
<td>61</td>
</tr>
</tbody>
</table>

Figure 4.5 Effect of the jet Reynolds number on the surface heat transfer coefficients

Usually, the dependence of an average Nusselt number on Reynolds number is expressed as $Nu_{ave} = ARe^b$. Based on the simulations, relationships between Nu and
Re over a range of X/W = 0 ~ 24 such as $Nu_{ave} = 0.1023 \text{Re}^{0.603}$ and $Nu_{ave} = 0.126 \text{Re}^{0.576}$ were developed by standard k-ε model and RSM model, respectively, with the correlation coefficient, $R^2 = 0.996$ for both the models. The majority of the studies reported “b” values ranging from 0.5 to 0.8. For example, Saad (1981) found that the exponent “b” was constant at a value of 0.65 for all practical purposes and Martin (1977) reported it to be 0.67.

4.2.4 Effect of turbulence level at the nozzle exit on heat transfer

The turbulence intensity and length scale at the nozzle exit are important factors to represent the turbulence. The turbulence intensity, I, is defined as the ratio of root-mean-square of the velocity fluctuations, $u$, to the mean flow velocity, $u_{avg}$, while the turbulence length scale, $l$, represents the size of the large eddies that contain the energy in turbulent flows. Higher turbulence levels are commonly associated with high heat transfer coefficients. However, exploratory computations by Morris et al. (1996) showed that the turbulence intensity at the inlet (nozzle exit) had a negligible effect on the exit profile by using the standard k-ε model in FLUENT. In this work, effects of turbulence intensity and turbulence length scales on the flow characteristics, heat transfer and turbulent kinetic energy distributions were evaluated. The geometric parameters and boundary conditions for the cases studied are listed in Table 4.3.

<table>
<thead>
<tr>
<th>Case</th>
<th>W (mm)</th>
<th>H/W</th>
<th>Re_j</th>
<th>T_j (K)</th>
<th>Boundary Condition at Impingement Surface</th>
<th>Turbulence Length Scales</th>
<th>Turbulence Intensity on Nozzle Exit %</th>
</tr>
</thead>
<tbody>
<tr>
<td>Case 8</td>
<td>14.1</td>
<td>2.6</td>
<td>10200</td>
<td>310</td>
<td>$T_w=348$ K</td>
<td>l=D</td>
<td>2, 4, 6, 8 and 10</td>
</tr>
<tr>
<td>Case 9</td>
<td>14.1</td>
<td>2.6</td>
<td>10200</td>
<td>310</td>
<td>$T_w=348$ K</td>
<td>l=0.07 D</td>
<td>2, 4, 6, 8 and 10</td>
</tr>
</tbody>
</table>
Figure 4.6 Streamline contour predicted by the standard k-\(\epsilon\) model for different turbulence intensities and length scales

a: turbulence intensity = 2\% (0 \sim 0.1448 kg/s), turbulence length scale = D;
b: turbulence intensity = 10\% (0 \sim 0.1446 kg/s), turbulence length scale = D;
c: turbulence intensity = 2\% (0 \sim 0.129 kg/s), turbulence length scale = 0.07D;
d: turbulence intensity = 10\% (0 \sim 0.1446 kg/s), turbulence length scale = 0.07 D

Figure 4.7 Effect of turbulence intensities and turbulence length scales on Nusselt number and turbulent kinetic energy predicted by the standard k-\(\epsilon\) model
Figure 4.8 Streamline contour predicted by RSM model for different turbulence intensities and length scales
a: turbulence intensity = 2% (0~0.1364 kg/s), turbulence length scale = D;
b: turbulence intensity = 10% (0~0.1397 kg/s), turbulence length scale = D;
c: turbulence intensity = 2% (0 ~ 0.121 kg/s), turbulence length scale = 0.07D;
d: turbulence intensity = 10% (0 ~ 0.125 kg/s), turbulence length scale = 0.07D

Figure 4.9 Effect of turbulence intensities and turbulent length scales on turbulent kinetic energy predicted by RSM model
The streamline contours for different turbulence intensities and length scales predicted by the standard k-ε model in the whole domain (Figure 4.6) show small effect of the turbulence intensity and large effect of the turbulence length scales at the nozzle exit. When the turbulence intensity increased, the size of the main vortex decreased slightly, while it decreased significantly when the turbulence length scales increased from 0.07D to D (D is the hydraulic diameter). Modest changes can be seen in the Nusselt number and turbulent kinetic energy distributions in the stagnation region when the turbulence intensity increased from 2% ~ 10% predicted by the standard k-ε model in Figure 4.7. For example, when the turbulence intensity increased from 2% to 10%, the turbulent kinetic energy and Nusselt number at the stagnation zone increased by 32.9% and 10.7%, respectively (for Re=10,200, H/W=2.6). Away from the stagnation zone, turbulent kinetic energy and Nusselt number distributions showed no influence of turbulence intensity. These results are similar to those of Morris et al. (1996) where turbulence intensity did not influence the exit flow profiles. Greater impact in turbulent kinetic energy (126% increase) and the Nusselt number (53% increase) at the...
stagnation zone was seen when the turbulence length scale was increased from 0.07D to D. Thus, turbulence intensity and length scales at the nozzle exit affected the Nusselt number distributions more in the impingement region than in the downstream region for the standard k-ε model.

Greater difference in streamline contours can be seen when RSM was used to characterize the effect of turbulence intensity and length scales at the nozzle exit (Figure 4.8). The recirculation zone shifted upstream and the size of the recirculation decreased when the turbulence intensity and length scale increased. Similarly, much greater effects of both the turbulence intensity and length scale were observed on the maximum turbulent kinetic energy and Nusselt number as shown in Figures 4.9 and 4.10, respectively. In addition to the impingement region, the Nusselt number and turbulence kinetic energy in the downstream zone were also affected by the turbulence intensity and the magnitude of turbulence length scale at the nozzle exit, which is quite different from the standard k-ε model.

4.2.5 Effect of the near wall function on the predicted Nusselt number

As mentioned in chapter 3, the nonequilibrium wall function partly accounts for the effect of pressure gradients and the departure from equilibrium. The nonequilibrium wall function is commonly claimed to be suitable for complex and separated flows. Morris et al. (1999) in their work on jet impingement heat transfer used the standard wall function over the Rej range of 2000-13,000, and H/D = 2~4 and the nonequilibrium wall function for the cases with Rej ≥ 13,000. However, no comparison of the effects of chosen wall functions on the heat transfer rate distributions was presented. In this work, we present a case for H/W = 2.6 and Rej = 71,300. It is noted
that the standard wall function predicts slightly higher Nusselt numbers than the nonequilibrium wall function at the stagnation and downstream zones for both the turbulence models used (Figure 4.11). Since the simulated Nusselt number values are generally higher than the experimental data, use of nonequilibrium wall function can be considered to be marginally better than the standard wall function. However, generally the standard wall function is more stable and converges faster than the nonequilibrium wall function. Thus, for low Re$_j$, the standard wall function was chosen and nonequilibrium wall function was used for Re$_j$ greater than 20,000. On passing, it was observed that at high Reynolds number, k-ε model produces much greater stagnation Nusselt number than the Reynolds Stress Model.

![Figure 4.11 Effect of wall functions on Nusselt number distributions](image)

4.2.6 Effect of the magnitude of the heat flux

Four cases were selected to evaluate the effect of heat fluxes applied at the impingement surface on heat transfer. The geometric parameters and boundary
conditions for these four cases are listed in Table 4.4. The properties of the materials were treated as constant for cases 10-12 as a small range of temperature difference was used in these computations, while the properties of the materials were defined as temperature-dependent for case 13 where large range of temperature difference (>100K) was used in calculations.

Table 4.4 Geometric parameters and boundary conditions for the test cases

<table>
<thead>
<tr>
<th>Case</th>
<th>W (mm)</th>
<th>H/W</th>
<th>Re</th>
<th>Tj</th>
<th>Boundary Condition at Impingement Surface (W/m²)</th>
<th>Boundary Condition at Confinement Surface (W/m²)</th>
<th>Turbulence Intensity on Nozzle Exit</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
<td>5</td>
<td>6</td>
<td>12000</td>
<td>302</td>
<td>1000</td>
<td>0</td>
<td>2 %</td>
</tr>
<tr>
<td>11</td>
<td>5</td>
<td>6</td>
<td>12000</td>
<td>302</td>
<td>3000</td>
<td>0</td>
<td>2 %</td>
</tr>
<tr>
<td>12</td>
<td>5</td>
<td>6</td>
<td>12000</td>
<td>302</td>
<td>5450</td>
<td>0</td>
<td>2 %</td>
</tr>
<tr>
<td>13</td>
<td>5</td>
<td>6</td>
<td>12000</td>
<td>302</td>
<td>50000</td>
<td>0</td>
<td>2 %</td>
</tr>
<tr>
<td>14</td>
<td>5</td>
<td>6</td>
<td>12000</td>
<td>302</td>
<td>310 (K)</td>
<td>0</td>
<td>2 %</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Figure 4.12 Comparison of Nusselt number at different thermal boundary conditions</th>
</tr>
</thead>
<tbody>
<tr>
<td>Nu predicted by standard k-ε model</td>
</tr>
<tr>
<td>Nu predicted by RSM model</td>
</tr>
</tbody>
</table>

Re = 12000, W = 5 mm, H/W = 6, Tj = 302 K
The effect of heat flux on the impinging surface as predicted by the standard k-ε and RSM models on the Nusselt number is shown in Figure 4.12. Both models showed that the variable heat fluxes did not influence the numerical values of Nusselt number at low heat flux values (Figure 4.12). When the heat flux at the impingement surface was increased from 1000 W/m$^2$ to 5450 W/m$^2$, the greater difference of Nusselt number occurred at X/W about 2.5. With the increase of the heat flux of the impingement surface from 1000 W/m$^2$ to 5450 W/m$^2$, the Nusselt number (at X/W at about 2.5) increased only by 11% by the standard k-ε model and 14% by the RSM model, respectively. Beyond X/W=13, the Nusselt number changed slightly for different heat flux values. However, when the heat flux at the impingement surface was increased to a very high value, 50,000 W/m$^2$, the Nusselt numbers were decreased significantly (as shown in Figure 4.12). Interestingly, the difference in Nusselt number with increased heat flux was higher with the increasing X/W.

The effect of heat flux on Nusselt number and wall temperature distributions has important significance in practice since it permits one to use data and empirical design correlations carried out using isothermal, constant heat flux or variable temperature impingement surface. For example, the two dotted lines in Figure 4.12 are the Nusselt number distributions with the constant temperature boundary condition (the constant temperature is the average temperature of the inlet and outlet temperatures on the impingement surface for case 12). By comparing the Nusselt numbers under the constant temperature boundary conditions with those under the constant heat flux boundary conditions, it can be seen that for low heat flux values, the Nusselt number distributions with the constant temperature boundary condition were quite similar to those of constant heat flux boundary conditions (constant temperature was the average
temperature of the impingement surface with the constant heat flux boundary conditions).

4.3 Conclusions

Two turbulence models, namely the standard k-ε model and the RSM, were used to calculate the heat transfer under a semi-confined turbulent impinging slot jet for various flow and geometric parameters. It was found that the qualitative trends predicted by both turbulence models compared very well with the experimental trend in most cases. The simulated results compare favorably with the experimental results for large nozzle-to-target spacing, but the predictions for small nozzle-to-target spacing call for further improvement in the simulation. Considering that all experimental results are subject to an uncertainty level, which is rarely quantitatively reported by the authors, and inadequate description about the flow conditions at the nozzle exit, the agreement between simulation and experimental data is quite encouraging. Some specific conclusions can be drawn from the study conducted in this chapter as follows:

i) The standard k-ε model was not successful in predicting the secondary flow patterns, which occur for small H/W ratios, whereas RSM could predict a secondary peak for such cases, albeit at a different distance than the experimental value.

ii) The turbulent Pr number brought about noticeable effects on the computed Nusselt number in the downstream direction, and a better match between the simulated and experimental data was observed when the turbulent Pr was increased from its default value of 0.85 to 1.5.
iii) Differences between the simulated and experimental heat transfer coefficients in the stagnation zone increased with increasing Reynolds number from 1500 to 12,000.

iv) The standard wall function predicted slightly higher Nusselt number distributions than the nonequilibrium wall function for both the standard k-\(\varepsilon\) and the Reynolds Stress models.

v) The turbulence length scale at the nozzle exit brings forth significant changes in the Nusselt number distribution by both the models, whereas the effect of turbulence intensity at the nozzle exit is more remarkable in case of only RSM model.

vi) For low heat flux values (1000–6000 W/m\(^2\)), the physical properties of air are temperature-independent and the heat flux has negligible effect on the Nusselt number distribution. However, for large heat flux (50,000 W/m\(^2\)), the physical properties of air become temperature-dependent and the heat flux has significant effect on the Nusselt number distribution.

In this chapter, we demonstrate the use of CFD as a successful prediction tool for flow problems involving impinging jet by comparing experimental data with the simulations, although further improvement in prediction is necessary for small nozzle-to-target spacing, especially in the impingement region.
Chapter 5

Effect of temperature difference between the jet and impingement surface on heat transfer

5.1 Introduction

In heating, cooling or drying applications involving large temperature differences between the jet and the target surface, it is necessary to incorporate the temperature-dependence of fluid properties on the flow and temperature fields. Despite their frequent occurrence in industrial practice, there is little research reported in the literature on this subject. It is also necessary to distinguish between heating and cooling applications since the thermo-physical properties of the fluid in the vicinity of the target surface vary in different directions for the two cases. In this chapter, CFD simulation results for heat transfer in a semi-confined slot turbulent jet under thermal boundary conditions where temperature-dependence of the fluid properties affect the flow and thermal fields are presented. A comparative analysis of the performances of standard k-ε and RSM models at constant target surface temperature in turbulent flow regimes is presented.

This chapter examines the effect of small and large temperature differences, $\Delta T$, ranging from 12 K to 272 K, between the jet and the impingement surface on the local Nusselt number distributions under isothermal boundary conditions. The local Nusselt number for an isothermal impingement surface can be defined as $Nu_x = \frac{h_x W}{\lambda}$, where $\lambda$ is the fluid thermal conductivity, $h_x = \frac{q}{\Delta T}$ and $\Delta T$ is the temperature difference between the jet and impingement surface. The fluid thermal conductivity can be
calculated according to different reference temperatures. The most popular method is to calculate the \( \lambda \) by film temperature, \( T_f = \frac{T_w + T_j}{2} \).

In this work, \( \lambda \) is calculated at jet, film and impingement wall temperatures and the corresponding Nusselt numbers are denoted by \( \text{Nu}_j \), \( \text{Nu}_f \) and \( \text{Nu}_w \), respectively. A comparison is made between the Nusselt number distributions under these three definitions of Nu. The choice of flow configuration and geometric parameters for this aspect of study is same as presented before in Chapter 4.

5.2 Results and discussion

Initially, the effects of temperature difference between the jet and impingement surface on Nusselt number were carried out by studying four temperature difference values (\( \Delta T = 12 \text{ K}, 52 \text{ K}, 172 \text{ K} \) and 272 K) for both heating and cooling conditions with the nozzle-to-target spacing, \( H/W = 6 \), by both the standard \( k-\varepsilon \) and RSM turbulence models.

5.2.1 Small temperature difference case

Figures 5.1 - 5.3 show the effects of small temperature differences between the jet and impingement surface on the variously defined Nusselt number distributions (\( \text{Nu}_j \), \( \text{Nu}_f \) and \( \text{Nu}_w \)) predicted by both the standard \( k-\varepsilon \) as well as the RSM models for both cooling and heating conditions.
Figure 5.1 Effect of small $\Delta T$ between the jet and the impingement surface on $\text{Nu}_j$ for cooling and heating. a: standard $k-\varepsilon$ model; b: RSM model
Figure 5.2 Effect of small $\Delta T$ between the jet and the impingement surface on $\text{Nu}_f$ for cooling and heating. a: standard k-$\epsilon$ model; b: RSM model.

Re = 12000, $W = 5$ mm, $H/W = 6$
Figure 5.3 Effect of small $\Delta T$ between the jet and the impingement surface on $N_u_w$ for cooling and heating. a: standard k-$\varepsilon$ model; b: RSM model

In these three figures, $N_u_j$, $N_u_f$ and $N_u_w$ predicted by RSM all reach a maximum at $X/W = 2.5$, while the standard k-$\varepsilon$ model predicts the maximum Nusselt number at the stagnation point. Both the standard k-$\varepsilon$ and RSM models showed that the Nusselt number changed in opposite directions with increase of the temperature difference for cooling and heating conditions since the thermo-physical properties of the fluid in the
vicinity of the target surface vary in different directions for heating and cooling conditions.

These figures also show that the three definitions of Nusselt number, $\text{Nu}_j$, $\text{Nu}_f$ and $\text{Nu}_w$, yield comparable results, i.e., $\text{Nu}_j$, $\text{Nu}_f$ and $\text{Nu}_w$ have nearly the same values at small $\Delta T$s. When the temperature difference between the jet and impingement surface increased from 12 K to 52 K, the difference in the wall Nusselt number ($\lambda$ is calculated at impingement wall temperature) was larger than that in the film Nusselt number ($\lambda$ is calculated at film temperature) and the jet Nusselt number ($\lambda$ is calculated at jet temperature). The largest difference in wall Nusselt number was about 10%, while the jet and film Nusselt number changed little when temperature difference increased from 12 K to 52 K. These results suggest that, under small temperature differences between the jet and impingement surface, the physical properties of the fluid do not change considerably and the fluid properties can be treated as temperature-independent for small temperature difference.

5.2.2 Large temperature difference case

Figures 5.4 – 5.6 present the results for large temperature differences between the jet and impingement surface. Similarly as for small temperature differences, the shape of the Nusselt number distributions ($\text{Nu}_j$, $\text{Nu}_f$ and $\text{Nu}_w$) predicted by RSM model are different from those predicted by the standard k-ε model; the peak values of the three Nusselt numbers predicted by the standard k-ε model and RSM model are at stagnation point and at $X/W = 2.5$, respectively.
Figure 5.4 Effect of large ΔT between the jet and impingement surface on Nu_j for cooling and heating. a: standard k-ε model; b: RSM model
Figure 5.5 Effect of large $\Delta T$ between the jet and impingement surface on $N_u$ for cooling and heating. a: standard k-$\varepsilon$ model; b: RSM model
In Figure 5.4, the spread in jet Nusselt number distribution was the least when the temperature difference between the jet and impingement wall increased from 12 K to 272 K for both cooling and heating conditions predicted by the standard k-ε as well as RSM turbulence models. Despite the large variation in T, the jet Nusselt numbers were almost unchanged at the stagnation point for both cooling and heating conditions. At
the exit, the jet Nusselt number varied the largest, but by only 7.5% according to the standard k-ε model and only 10% according to the RSM model for the cooling condition. For heating, the jet Nusselt number varied by only 2% and 4.5% as computed using the standard k-ε and RSM models, respectively.

When the temperature difference increased from 12 K to 272 K, the difference in the film Nusselt number and wall Nusselt number increased greatly according to both models. The Nusselt number changed with increase of ΔT in opposite directions for the cooling and heating conditions, which is also because the thermo physical properties of the fluid in the vicinity of the target surface vary in opposite directions for the two cases.

For the same variation in T as above, for cooling, the peak film Nusselt number varied by 29% and 27% as predicted by the standard k-ε and RSM models, respectively. For the heating condition, the peak film Nusselt number varied by about 20% as predicted by the two turbulence models tested. Also for the same range of ΔT for cooling, at the exit, the film Nusselt number varied by 30% and 36%; for heating, by 27.5% and 30% as predicted by the standard k-ε model and RSM model, respectively (Figure 5.5).

From Figure 5.6, we notice that, with the increase of ΔT, the difference in the wall Nusselt number was the largest among the three definitions. When ΔT increased from 12 K to 272 K, the peak Nusselt number decreased by 41% and 39% for cooling condition by the standard k-ε model and RSM model, respectively and increased 59% and 56% for heating condition by the standard k-ε model and RSM model, respectively. At the exit, the wall Nusselt number decreased by 42% and 48% for
cooling by the standard k-\(\varepsilon\) model and RSM model, respectively and increased by 75% and 115% for heating by the standard k-\(\varepsilon\) model and RSM model, respectively.

Based on the above results, we extended the numerical experiments for variable H/W values (H/W = 2.6, 6 and 12), and different jet Reynolds number values (Re = 1500, 3000, 6000 and 12,000). Only the standard k-\(\varepsilon\) model was used because although the standard k-\(\varepsilon\) model and RSM model produce different numerical values for Nusselt number distributions, both models produce the same conclusion about the effect of temperature difference on Nusselt number. Figures 5.7-5.10 show the effects of temperature differences between the jet and impingement surface on \(N_u_j\) predicted by the standard k-\(\varepsilon\) model for three nozzle-to-target spacing, H/W = 2.6, 6 and 12, for four jet Reynolds numbers, Re = 1500, 3000, 6000 and 12,000, respectively. Figures 5.11-5.14 and Figures 5.15-5.18 are similar to the Figures 5.7-5.10, however, presenting the results for \(N_u_f\) and \(N_u_w\), respectively.

From the above figures, we observe that for both small and large nozzle-to-target spacings, and for the range of jet Reynolds number from 1500 to 12,000, the \(N_u_j\) shows the least spread among the three Nusselt number definitions under the range of temperature differences between the jet and impingement surface from 12 K to 272 K and \(N_u_w\) shows the largest spread under the same range of \(\Delta T\). We also found that the difference in \(N_u_f\) and \(N_u_w\) increases with the increase of jet Reynolds numbers. For example, impingement region, for H/W = 12, the difference of \(N_u_f\) increased from 10% for Re = 1500 to 25% for Re = 12,000, similarly, also for H/W = 12, \(N_u_w\) increased from 40% for Re = 1500 to 70% for Re = 12,000.
Figure 5.7 Effect of temperature difference between the jet and impingement surface on jet Nusselt number for heating condition by the standard k-ε model
Figure 5.8 Effect of temperature difference between the jet and impingement surface on jet Nusselt number for heating condition by the standard $k$-$\varepsilon$ model.
Figure 5.9 Effect of temperature difference between the jet and impingement surface on jet Nusselt number for heating condition by the standard k-ε model.
Figure 5.10 Effect of temperature difference between the jet and impingement surface on jet Nusselt number for heating condition by the standard k-ε model
Figure 5.11 Effect of temperature difference between the jet and impingement surface on film Nusselt number for heating condition by the standard $k$-$\varepsilon$ model.
Figure 5.12 Effect of temperature difference between the jet and impingement surface on film Nusselt number for heating condition by the standard k-ε model.
Figure 5.13 Effect of temperature difference between the jet and impingement surface on film Nusselt number for heating condition by the standard $k$-$\varepsilon$ model.
Figure 5.14 Effect of temperature difference between the jet and impingement surface on film Nusselt number for heating condition by the standard k-ε model.
Figure 5.15 Effect of temperature difference between the jet and impingement surface on wall Nusselt number for heating condition by the standard k-ε model.
Figure 5.16 Effect of temperature difference between the jet and impingement surface on wall Nusselt number for heating condition by the standard $k$-$\varepsilon$ model.
Figure 5.17 Effect of temperature difference between the jet and impingement surface on wall Nusselt number for heating condition by the standard k-ε model.
All of the above discussion relating to the effect of large temperature differences between the jet and impingement surface on Nusselt number distributions indicates that, under large temperature differences between the jet and impingement surface, the physical properties of the fluid should be defined as temperature-dependent as the large temperature difference between the jet and impingement surface has great effect on film and wall Nusselt number distributions. The $\text{Nu}_j$ is the most suitable definition for the Nusselt number distributions in impinging jet heat transfer since it exhibited the least variation under large $\Delta T$ compared to those of $\text{Nu}_f$ and $\text{Nu}_w$. 

Figure 5.18 Effect of temperature difference between the jet and impingement surface on wall Nusselt number for heating condition by the standard k-ε model.
5.2.3 Effect of jet Reynolds number on the Nu_{ave} and Nu_{j0}

From the above discussion, we note that the jet Nusselt number is the best among the three definitions (Nu_{j}, Nu_{r} and Nu_{w}) to represent the impinging jet heat transfer, especially for large temperature differences between the jet and the impingement surface due to its least spread with the increase of ΔT. Usually, the dependence of the average Nusselt number and Reynolds number can be expressed as $Nu_{ave} = A \cdot Re^B$.

Based on the above study, we investigated the effect of jet Reynolds number on the average jet Nusselt number, Nu_{ave} and the stagnation jet Nusselt number, Nu_{j0}.

Figures 5.19 and 5.20 show the effect of jet Reynolds number on Nu_{ave} and Nu_{j0}, respectively, for the nozzle-to-target spacing equals 2.6, 6 and 12 by the standard k-ε model. The relationships between the jet Nusselt number and Re over a range of X/W = 0 - 20 were $Nu_{ave} = 1.502 \cdot Re^{0.654}$, $Nu_{jave} = 0.943 \cdot Re^{0.697}$ and $Nu_{jave} = 1.382 \cdot Re^{0.649}$ with the correlation coefficients, $R^2 = 0.999$, 0.993 and 0.998 for H/W = 2.6, 6 and 12, respectively (Figure 20). The majority of the studies reported “b” values ranging from 0.5 to 0.8. For example, Saad (1981) found that the exponent “b” was constant at a value 0.65 for all practical purposes and Martin (1977) reported it to be 0.67.
Finally, an empirical correlation of form $Nu_{j0} = A \cdot Re^a$ was developed based on the CFD predictions. The exponent values of the correlations were 0.3872 for $H/W = 2.6$, 0.6055 for $H/W = 6$ and 0.6088 for $H/W = 12$ predicted by the standard $k$-$\varepsilon$ model, as shown in Figure 5.21. In Chapter 4 we had seen that for small nozzle-to-target spacing...
Chapter 5

H/W = 2.6, the standard k-ε model over predicted the stagnation Nusselt number, while for large nozzle-to-target spacing, H/W = 12, the standard k-ε model under predicted the heat transfer coefficient at the stagnation point and when H/W = 6, the deviation of the numerical stagnation Nusselt number from the experimental stagnation Nusselt number was quite small. Hence, the correlation $Nu_{j0} = 0.1708 \text{Re}^{0.6055}$, for H/W = 6, could be more accurate than the other two correlations. That means that the exponent value of 0.6 should be quite reasonable. But these exponents have been evaluated only with 4 points, so they are subject to uncertainty. The slope of 0.5 in stagnation point had been widely cited in the literature (Lytle & Webb (1994) and Lee et al. (1995)).

5.3 Conclusions

Results of the CFD modelling of turbulent impinging jets show that large temperature differences between the jet and the impingement surface lead to significant differences in the heat transfer coefficients. Small temperature differences (e.g. up to 50 K) show only minor differences between the local Nusselt numbers calculated using thermal conductivity values at the jet, film or wall temperatures. However, large temperature differences (over 100 K) can result in significant differences between the three possible definitions of the Nusselt numbers. This is true for both heating and cooling conditions with impinging jets. It is shown that use of the jet temperature as the reference temperature for the calculation of the Nusselt numbers shows the least spread. This result allows the designer to use previously published correlations, which are obtained at small temperature differences between the jet and the impingement surface. This conclusion is valid regardless of the jet Reynolds number and nozzle-to-surface spacing. Also, the same conclusion is drawn from the use of the standard k-ε as
well as the Reynolds stress models of turbulence. Finally, empirical correlations are
provided for the average and stagnation jet Nusselt numbers under a semi-confined
turbulent slot jet based on the numerical results. The exponent values in the above two
correlations based on CFD models are comparable with the values reported in
literature which are based on experimental data.
Chapter 6

Effects of Pr on impinging jet heat transfer under a slot jet

6.1 Introduction

Despite the extensive literature dealing with impinging jet heat transfer, little attention has been directed at the effect of fluid Prandtl number. Almost without exception, earlier studies, both experimental and numerical, deal with air or water as the jet fluids. In this chapter, we carry out numerical experiments to examine the effect of different thermal properties of the fluid (gas or liquid phase) on heat transfer and to correlate the effect of the fluid Prandtl number on heat transfer under a semi-confined impinging slot jet. Both laminar and turbulent flows were studied. New results are presented for different gases viz. air, argon, nitrogen, hydrogen, helium, ammonia and ethylene and several liquids viz. water, benzene, ethanol, turpentine and iso-butyl alcohol. Local, stagnation and average values of the impingement Nusselt number as well as the heat transfer coefficient were reported by FLUENT 5. The configuration of the impinging slot jet was that shown in Chapter 1. Empirical correlations are derived for the stagnation and average Nusselt numbers as a function of the fluid Prandtl number on the basis of extensive numerical experiments. The average Nusselt number is given by

$$Nu_{ave} = \frac{1}{L} \int_0^L Nu(x)dx,$$

where L is the length of the impingement surface.
6.2 Results and Discussion

The fluids Prandtl number studied in this chapter varied from 0.67-71. Table 6.1 lists the properties of all the fluids investigated here. The results are presented as surface heat transfer coefficient as well as the corresponding Nusselt number distribution in the stream-wise direction. For liquids, the temperature of the impinged surface was well below the fluid boiling point.

Table 6.1 Thermal properties of fluids studied here at T = 300 K

<table>
<thead>
<tr>
<th>No.</th>
<th>Materials</th>
<th>λ (W/m K)</th>
<th>Cp (J/kg K)</th>
<th>Pr</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Argon (Ar), gas</td>
<td>0.0176</td>
<td>521</td>
<td>0.68</td>
</tr>
<tr>
<td>2</td>
<td>Nitrogen (N₂), gas</td>
<td>0.0267</td>
<td>1043</td>
<td>0.69</td>
</tr>
<tr>
<td>3</td>
<td>Ammonia (NH₃), gas</td>
<td>0.0246</td>
<td>2200</td>
<td>0.90</td>
</tr>
<tr>
<td>4</td>
<td>Hydrogen (H₂), gas</td>
<td>0.1980</td>
<td>14780</td>
<td>0.67</td>
</tr>
<tr>
<td>5</td>
<td>Helium (He), gas</td>
<td>0.1490</td>
<td>5200</td>
<td>0.70</td>
</tr>
<tr>
<td>6</td>
<td>Air, gas</td>
<td>0.0267</td>
<td>1005</td>
<td>0.69</td>
</tr>
<tr>
<td>7</td>
<td>Ethylene (C₂H₂), gas</td>
<td>0.0209</td>
<td>2229</td>
<td>1.10</td>
</tr>
<tr>
<td>8</td>
<td>Water (H₂O), liquid</td>
<td>0.6098</td>
<td>4181</td>
<td>6.00</td>
</tr>
<tr>
<td>9</td>
<td>Benzene (C₆H₆), liquid</td>
<td>0.1439</td>
<td>1735</td>
<td>7.00</td>
</tr>
<tr>
<td>10</td>
<td>Ethanol (C₂H₅OH), liquid</td>
<td>0.1676</td>
<td>2474</td>
<td>15.00</td>
</tr>
<tr>
<td>11</td>
<td>Turpentine (C₁₀H₁₆), liquid</td>
<td>0.1260</td>
<td>1800</td>
<td>21.20</td>
</tr>
<tr>
<td>12</td>
<td>Iso-butyl alcohol (C₄H₁₀O), liquid</td>
<td>0.1260</td>
<td>2313</td>
<td>71.10</td>
</tr>
</tbody>
</table>

6.2.1 Laminar flow

6.2.1.1 Effect of fluid Prandtl number on heat transfer rates

The local and average Nusselt numbers and surface heat transfer coefficients were investigated numerically as a function of fluid Prandtl number over the range of 0.7-71 for H/W = 6 and Re = 100. The effect of the Prandtl number on local Nusselt number distributions can be seen from Figure 6.1. As expected, the Nusselt number increases
with increase of Pr from 0.7 to 71, with the notable exception of air. Although the Prandtl number of air (Pr = 0.69) is lower than that of ethylene (gas, Pr = 1.1), the Nusselt number for air is found to be consistently greater than that for ethylene. One possible reason for this is that the fluid Prandtl number is not sufficient to account fully for the heat transfer rate distributions. A second peak value was found for laminar flow when fluid Prandtl number was greater than 1.1. Moreover, with the increase of the fluid Prandtl number, the second peak value of Nusselt number becomes predominant.

![Figure 6.1](image.png)

**Figure 6.1** Effect of Pr number on the local Nusselt number distributions for laminar flow

The distributions of the computed heat transfer coefficients are shown in Figure 6.2. For all fluids the maximum values of the heat transfer coefficients are located in the impingement region, the minimum values occur at the exit and the second peak value of heat transfer rate occurs only when fluid Prandtl number is greater than 1.1. However, the surface heat transfer coefficients do not increase with increase of the fluid Prandtl number. In Figure 6.2, we find that the surface convective heat transfer coefficient for the liquid water jet is much higher than those of other fluids studied.
here. The stagnation point heat transfer coefficient for the water jet is 50 times greater than that for air. Thermal conductivity of water is much higher than those of other fluids, as shown in Table 6.1. Higher thermal conductivity improves heat transfer in the boundary layer and therefore results in a higher heat transfer rate.

![Figure 6.2](image-url)  
**Figure 6.2** Effect of Pr number on surface heat transfer coefficient distributions for laminar flow

Figures 6.3 and 6.4 show the distributions of the local Nusselt number and surface heat transfer coefficient of 5 different gases with similar fluid Prandtl number (0.9 for NH₃ and about 0.7 for others), respectively. As expected, the local Nusselt number distributions of the gases with similar Prandtl number are quite similar, as shown in Figure 6.3. However, much greater differences in the predicted surface heat transfer coefficients were noticed among the gases. From the Figure 6.4, it can be seen that the surface heat transfer coefficient distributions for air, argon and ammonia are similar, while the surface heat transfer coefficients for hydrogen and helium are much greater than those for other gases. The stagnation point heat transfer coefficient for the hydrogen jet is about 7 times greater than those for air, argon, nitrogen and ammonia.
As seen in the earlier section, thermal conductivity values of hydrogen and helium are much higher than those of the other gases tested here, Table 6.1.

**Figure 6.3** Local Nu distributions for gases with similar fluid Pr number for laminar flow

**Figure 6.4** Heat transfer coefficient distributions for gases with similar fluid Pr number for laminar flow
6.2.1.2 Correlations between Prandtl number and Nusselt number

Based on the numerical results obtained over a wide range of fluid physical thermo properties, empirical correlations were obtained for the stagnation and average Nusselt numbers in a semi-confined laminar slot impinging jet. Heat transfer correlations by regression analysis of numerical results of 11 test fluids (except the N₂) (Pr = 0.67 – 71.1) were developed.

The left and right axes of Figure 6.5 show the best-fit correlations for the stagnation Nusselt number and the average Nusselt number, respectively. The relations between stagnation and average Nusselt numbers and the fluid Prandtl number can be expressed as \( Nu_0 = A \Pr^b \) and \( Nu_{\text{ave}} = A' \Pr^{b'} \), respectively. The correlations between \( Nu_{\text{ave}} \) and \( \Pr \) were conducted over a range of \( X/W = 0 - 60 \). Our simulation results showed that the values of \( R^2 \) for the correlations of \( Nu_0 \) and \( Nu_{\text{ave}} \) were 0.81 and 0.97, respectively. The exponent values of \( Nu_0 \) and \( Nu_{\text{ave}} \) were 0.2512 and 0.4095, respectively. Bejan’s scaling analysis (Bejan (1995)) shows that, for laminar boundary layers flow \( Nu \propto \Pr^{\frac{1}{2}} \) for fluids \( \Pr \ll 1 \) and \( Nu \propto \Pr^{\frac{1}{3}} \) for fluids \( \Pr \gg 1 \). Figure 6.5 also indicated that the fluid Prandtl number alone does not fully correlate the Nu data, perhaps the Prandtl number exponent should not be taken as a constant. Additional studies are needed to resolve this issue.
6.2.2 Turbulent flow

6.2.2.1 Effect of fluid Prandtl number on heat transfer rates

The effect of Prandtl number on local Nusselt number distributions for turbulent flow predicted using the standard k-ε model is shown in Figure 6.6. Similar to the corresponding results for laminar flow, the Nusselt number increases with the increase of Pr from 0.7 to 71, with the noticeable exception of air. Although the Prandtl number of air is lower than that of ethylene, the Nusselt number for air is found to be consistently greater than that for ethylene. Moreover, with the increase of fluid Prandtl number, the location of the maximum Nusselt number shifted from the stagnation point (e.g. air) in the downstream direction; the Nusselt number then decreases monotonically from the peak towards the exit. For example, for the iso-butyl alcohol jet, the maximum Nusselt number is located at X/W = 2.3. That is a notable difference from the laminar flow case, where two peak values of Nusselt number occurred. The differences in Nusselt number distributions among the different fluids are the largest in
the impingement region and are the smallest in the exit region of the wall jet of the channel.

The distributions of the computed surface heat transfer coefficients are shown in Figure 6.7. As noted earlier, for all fluids the maximum values of the surface heat transfer coefficients are located in the impingement region and the minimum values occur in the exit of section. However, the surface heat transfer coefficients do not increase with increase of the fluid Prandtl number. In Figure 6.7, we find that the surface heat transfer coefficient for the water jet is much higher than those of other fluids studied here. The stagnation point heat transfer coefficient for the water jet is 17 times greater than that for air. In our previous study of laminar impinging jets, the stagnation point heat transfer coefficient for the water jet was 50 times greater than that for air. Li and Garimella (2001) also showed that the stagnation heat transfer coefficient for water jet was 3.7 times higher than that for the FC-77 jet (Pr = 25). The reason for this observation is the same as that for laminar flow. It is that the thermal conductivity of water is much higher than those of the other fluids. Higher thermal conductivity improves heat transfer in the boundary layer and therefore results in a higher heat transfer rate.
Chapter 6

Figure 6.6 Effect of Pr number on Nusselt number distributions for turbulent flow

Figure 6.7 Effect of Pr number on surface heat transfer coefficient distributions for turbulent flow

Distributions of the local Nusselt number and surface heat transfer coefficient for jet of 6 different gases with similar fluid Prandtl number (0.9 for NH₃ and about 0.7 for others) for turbulent flow are shown in the Figures 6.8 and 6.9, respectively. Similar to
the results in our previous study of laminar jets, gases with similar Prandtl number in turbulent flow produce similar values of the local Nusselt number. At the stagnation point, the largest difference among the predicted Nusselt number was only about 15%. However, much greater differences in the predicted surface heat transfer coefficients were noticed among the gases. From the Figure 6.9, it can be seen that the surface heat transfer coefficient distributions for air, argon, nitrogen and ammonia are similar, while the surface heat transfer coefficients for hydrogen and helium are much greater than those for other gases. The stagnation point heat transfer coefficient for the hydrogen jet is 5.5 times greater than that for air, argon, nitrogen and ammonia. In our previous study of laminar flow, this factor was 7. As seen in the earlier section, thermal conductivity values of hydrogen and helium are much higher than those of the other gases tested here, shown in Table 6.1.

![Figure 6.8](image.png)

**Figure 6.8** Local Nusselt number distributions of different gases with similar fluid Prandtl number predicted by the standard k-ε model
6.2.2.2 Correlations between Prandtl number and Nusselt number

Based on the numerical results obtained over a wide range of fluid thermo physical properties, empirical correlations are obtained for the stagnation and average Nusselt numbers in a semi-confined turbulent slot impinging jet. The heat transfer correlations by regression analysis of numerical results using 3 different databases were compared. The correlations were first developed for all 12 fluids tested here (Pr = 0.67 – 71.1). For the next sets of data for regression, we separated the database into two sets: one consisting of 6 gases (Pr<1) and the other for 6 fluids (Pr>1). Note that only for Pr=1, the thermal and velocity boundary layers coincide.

The left and right axes of Figure 6.10 show the best-fit correlations for the stagnation Nusselt number and the average Nusselt number, respectively. The relations between stagnation and average Nusselt numbers and the fluid Prandtl number can be expressed as $Nu_0 = A Pr^b$ and $Nu_{ave} = A' Pr^{b'}$, respectively. The correlations between $Nu_0$ and $Pr$
as well as $\bar{N}u$ and $Pr$ over a range of $X/W = 0 - 20$ are summarized in Table 6.2 for the data sets of all test fluids and 6 fluids ($Pr > 1$). The correlations of $Nu_0$ and $\bar{N}u$ for 6 gases ($Pr < 1$) are not listed in Table 6.2. The database of the 6 gases ($Pr < 1$) is not suitable for regression since the 6 gases have quite similar Prandtl numbers (about 0.7), which results in very high relative error in regression.

Table 6.2 Stagnation and average Nusselt number correlations

<table>
<thead>
<tr>
<th>Materials</th>
<th>Correlations ($Nu_0$)</th>
<th>$R^2$</th>
<th>Correlations ($\bar{N}u$)</th>
<th>$R^2$</th>
</tr>
</thead>
<tbody>
<tr>
<td>12 fluids</td>
<td>$Nu_0 = 15.39 Pr^{0.215}$</td>
<td>0.63</td>
<td>$\bar{N}u = 8.56 Pr^{0.351}$</td>
<td>0.93</td>
</tr>
<tr>
<td>6 fluids ($Pr &gt; 1$)</td>
<td>$Nu_0 = 6.173 Pr^{0.4408}$</td>
<td>0.97</td>
<td>$\bar{N}u = 7.393 Pr^{0.4275}$</td>
<td>0.95</td>
</tr>
</tbody>
</table>

Our simulation results showed that the values of $R^2$ for the correlations of $Nu_0$ and $\bar{N}u$ for 6 gases ($Pr < 1$) were as low as 0.2 and 0.32, respectively. By comparing the numerical results in Table 6.2 with the published experimental data of Li and Garimella (2001), we find that the exponents of Prandtl number for the 6 fluids ($Pr > 1$) are comparable with the published data. Li and Garimella (2001) reported the Prandtl number exponents to be 0.444 and 0.441 for stagnation Nusselt and average Nusselt numbers, respectively. From Table 6.2, we find that the $R^2$ of the correlations for both $Nu_0$ and $\bar{N}u$ for 12 fluids are lower than those for the 6 fluids ($Pr > 1$) jets. According to Bejan’s scale analysis (Bejan (1995)), the Nusselt number depends on Pr number with different exponent values for low Prandtl number fluids ($Pr << 1$) and high Prandtl number fluids ($Pr >> 1$). As mentioned in previous part, Bejan’s scale analysis shows that, for laminar boundary layers flow $Nu \propto Pr^{1/2}$ for fluids ($Pr << 1$)
and $N_u \propto \text{Pr}^{1/3}$ for fluids ($\text{Pr} \gg 1$). Thus, it is reasonable to obtain separate correlations for high and low Prandtl number fluids.

![Figure 6.10 Effect of Prandtl number on stagnation and average Nusselt number for turbulent flow](image)

6.3 Conclusions

The local and average Nusselt numbers and surface heat transfer coefficients on a semi-confined laminar and turbulent slot impinging jets were investigated numerically as a function of fluid Prandtl number over the range of 0.7-71. The variations of Nusselt number for all the test fluids are similar to those for air. The Nusselt number increases with increase of fluid Pr number from 0.7 to 71. For laminar flow, when fluid Prandtl number is higher than 1.1, there are two peak values in the local Nusselt number distributions for $H/W = 6$; the second peak value of Nusselt number becomes predominant with increase of the Prandtl number. On the contrary, for turbulent flow, there is only one peak value of Nusselt number for the fluids studied here and with the
increase of fluid Prandtl number, the location of the maximum Nusselt number shifted from the stagnation point (e.g. air) in the downstream direction. Gases with similar Prandtl number exhibit similar values of Nusselt number, but different values of the surface heat transfer coefficient for both laminar and turbulent flows due to their widely different thermal conductivity. Predictive correlations are proposed for the effect of Prandtl number on the stagnation and average Nusselt numbers.
Chapter 7
Effect of cross flow on turbulent flow and heat transfer characteristics under normal and oblique semi-confined impinging slot jets

7.1 Introduction

Impinging jets are commonly used in industrial dryers and electronics chip cooling. Since in industrial practice it is necessary to use multiple jets, the interaction between jets can have important effect on their heat transfer performance. Hence, the study of cross-flow caused by the spent flow of upstream jets is obviously significant. Despite their practical implications, the normal and oblique impinging jets in cross flow have received very little attention both experimentally and numerically. Few researchers have studied the effect of cross-flow as well as jet inclination angles on slot jet impingement flow and heat transfer behavior. The effect of cross-flow on Nusselt number distributions has important significance in practice and much experimental and modeling work is needed for efficient design of the multiple impinging jet heat transfer devices by accounting for the adverse effects of the cross-flow.

In this chapter, flow and heat transfer characteristics for a single semi-confined turbulent slot jet of air impinging normally or obliquely into an imposed air cross-flow of the same or different temperature are simulated by FLUENT 5. The standard $k-\varepsilon$ and the Reynolds stress models were used as before. Effects of the various flow parameters (e.g. jet-to-cross-flow mass ratio) and geometric parameters (e.g. nozzle-to-target spacing and jet angle) were evaluated at a fixed Reynolds number for equal and unequal temperatures of the jet and cross-flow. These models are selected mainly due
to their earlier applications and good performance for prediction of flow and heat transfer in impingement heat transfer in the presence of cross-flow (Kalita et al. (2002)), and also due to our past experience with these models (chapter 4).

7.2 Results and Discussion

In this work, because the computational domain is asymmetric, the simulation is conducted over the whole domain. Grid-independence tests were conducted for each case. The flow domain simulated is shown in Figure 7.1. The following boundary conditions were applied: i) the impingement surface was specified as an isothermal wall, ii) the confinement surface was considered to be an adiabatic wall, iii) the jet inlet and cross-flow inlet were considered as velocity inlets with uniform velocities, and iv) the outlets were considered as fully developed outflow. The values of 2 and 4.5 percent were chosen arbitrarily for the turbulence intensity for jet inlet and cross-flow inlet, respectively. The turbulence length scale at the inlets for all the calculations was defined to be equal to 0.07 times the respective hydraulic diameter. The turbulence intensity at the boundary is expected to have negligible effect on the impingement heat transfer in the region of interest. As correct velocity field description is essential for the accurate prediction of the temperature field, in this chapter, the flow patterns are discussed first; subsequently heat transfer results will be discussed.

*Figure 7.1 Flow geometry of the impinging slot jet with cross-flow*
7.2.1 Effect of cross flow and jet angles on flow pattern

In this work, four cross-flow mass ratios ($M = 0, 0.05, 0.25$ and $1$), four jet angles ($\theta = -30^\circ, -15^\circ, 15^\circ$ and $30^\circ$) and two nozzle-to-target spacings ($H/W = 2.6$ and $H/W = 8$) were simulated by both the standard $k$-$\varepsilon$ and RSM turbulence models. In this part, for brevity, we only show the streamline contour results for $M = 0.05$ and $1$; $\theta = -30^\circ$ and $30^\circ$. Also, because of the small differences in the streamline contour predictions by the standard $k$-$\varepsilon$ model and the RSM model, we only show the streamline contours predicted by the standard $k$-$\varepsilon$ model in this section.

**Figure 7.2** Streamline contours for $\theta = -30^\circ$ for two structures under small cross flow value, $M = 0.05$ (two figures are not under same scale)
Figure 7.3 Streamline contours for $\theta = -30^\circ$ for two structures under large cross flow value, $M = 1$ (two figures are not under same scale)

$H/W = 2.6$, (0-0.37 kg/s)

$H/W = 8$, (0-0.36 kg/s)

Figure 7.4 Streamline contours for $\theta = 30^\circ$ for two structures under small cross flow value, $M = 0.05$ (two figures are not under same scale)

$H/W = 2.6$, (0-0.23 kg/s)

$H/W = 8$, (0-0.36 kg/s)
Figures 7.2 through 7.5 show the streamline contours for different cross-flow mass values and different jet angles for two H/W values as predicted by the standard k-ε model. From these figures, we can draw the following conclusions for both small nozzle-to-target spacing, H/W = 2.6, and large nozzle-to-target spacing, H/W = 8, and for both positive and negative jet angles when M increases from a small value, M = 0.05, to a relatively large value, M = 1.

a) When M equals 0.05, it is difficult to identify the stagnation point. When M equals 1, there is no stagnation point; the jet does not reach the impingement surface.

b) There are two main vortices for the smaller cross flow value, M = 0.05. One is located upstream of the jet, the other is located downstream.

c) The upstream main vortex disappears and the downstream main vortex becomes smaller and shifts to the confinement surface when the cross-flow value increases to 1.
d) When the cross flow value $M$ equals 1, the downstream main vortex, which is near the confinement surface, increases with the angle of inclination of the jet.

These results indicate that when a small amount of cross-flow ($M=0.05$) is introduced, a recirculation region develops near the impingement surface on the upstream side of the jet. At a higher cross-flow value ($M=1$), the recirculation region disappears. When the cross-flow mass is increased, the reattachment length of the main recirculation bubble on the side of outflow boundary decreases in size and shifts towards the confinement surface.

For a fully developed turbulent slot jet, the maximum heat transfer rate occurs typically at the stagnation point. However, for a turbulent impinging flow, subjected to a cross-flow, the separation point for the maximum heat transfer is different from that for the impingement jet without cross-flow. Therefore, in this section, we first present the effect of cross-flow on the heat transfer rate distributions and compare the simulation results with earlier reported data by Mujumdar et al. (1985). Subsequently, the studies of the effect of cross flow, jet angles, nozzle-to-target spacing and temperature difference between the jet and cross flow on heat transfer are presented. The heat transfer coefficient over the impingement surface was normalized in the form of a local Nusselt number for an isothermal impingement surface.

7.2.2 Effect of cross-flow on normal impinging jet heat transfer rate

Figure 7.6 shows the effect of cross-flow on the local Nusselt number distributions predicted by the standard $k-\varepsilon$ model. The results show that the maximum value of the Nusselt number decreased and shifted downstream as $M$ increased; also, its value in
the downstream region increased. Mujumdar et al. (1985) reported similar results. The decrease and shift of the maximum Nusselt number may be explained by examining the streamline contours when cross-flow is introduced. According to the streamline contours shown earlier, the stagnation point disappears and the size of the main recirculation bubble on the side of the outflow boundary (downstream) decreases. Cross-flow decreases the pressure gradient in the impingement wall region reducing the size of the recirculation bubbles, and consequently reducing the Nusselt number.

Based on the above comparison, we studied the effects of cross-flow and jet inclination angle on heat transfer characteristics. As with the flow study, four values of the cross-flow parameters were chosen ($M = 0, 0.05, 0.25$ and $1$) with four cases of varying jet angle ($\theta = -30^\circ, -15^\circ, 15^\circ$ and $30^\circ$) for both small and large nozzle-to-target spacings, $H/W = 2.6$ and $8$.

![Figure 7.6 Effect of cross-flow on Nusselt number distributions predicted by the standard k-\(\epsilon\) model](image-url)
7.2.3 Effect of jet angle on local heat transfer rate distribution

As noted earlier, jet impinging at an angle on a higher temperature plate creates an asymmetrical flow pattern. In Figures 7.7-7.10, the local Nusselt number predicted by both the standard k-ε and RSM models are plotted vs \( X/W \) for different angles of \( \theta \) at constant values of H/W and Re for \( M = 0, 0.05, 0.25 \) and 1, respectively.

![Figure 7.7](image1.png)

**Figure 7.7** Effect of \( \theta \) on Nusselt number predicted by the standard k-ε and RSM models for \( M = 0 \)
(The values of H/W, Re, M, \( \theta \), \( T_j \), \( T_{cf} \), \( T_w \), and W are same for both sides)

![Figure 7.8](image2.png)

**Figure 7.8** Effect of \( \theta \) on Nusselt number predicted by the standard k-ε and RSM models for \( M = 0.05 \)
(The values of H/W, Re, M, \( \theta \), \( T_j \), \( T_{cf} \), \( T_w \) and W are same for both sides)
From the above figures, we note that the peak values of the Nusselt number are still in the downstream region from the stagnation plane for the range of $\theta$ from $-30^\circ$ to $30^\circ$ when the values of $M$ change from 0 to 1. However, the location of the peak value of Nusselt number shifts upstream (i.e., moving towards the stagnation point) when the
angle $\theta$ increases from $-30^\circ$ to $30^\circ$ for the range of cross-flow values from 0 to 1. In addition, the value of the peak Nusselt number increases when the angle $\theta$ changes from negative to positive. In terms of the values and shapes of the Nusselt number distributions, the standard k-\(\varepsilon\) and RSM models produce similar results.

7.2.4 Effect of cross-flow and jet angles on the average Nusselt number

Many authors investigated the average Nusselt number in the impingement region for the normal impinging jet without cross flow. For example, Abdlmonem et al. (2000) reported average Nusselt numbers for obliquely impinging turbulent jets. However, there is little information available on the average Nusselt number for obliquely impinging jets in cross-flow. In this section, we present the average Nusselt number vs cross flow values, M, for different jet angles as predicted by both the standard k-\(\varepsilon\) and RSM turbulence models.

Figures 7.11 and 7.12 show the effect of M on the average Nusselt number for different jet angles for $H/W = 2.6$ and $H/W = 8$, respectively. From these figures, we find that, for both $H/W = 2.6$ and $H/W = 8$, the average Nusselt number increases with the increase of the jet angles for the range of cross-flow values from zero to one. Figure 7.12 shows that for $H/W = 8$, the average Nusselt number decreases with increase of the cross-flow value from 0 to 1. However, interestingly for $H/W = 2.6$, the average Nusselt number remains nearly unchanged when cross-flow values increased from 0.25 to 1 as shown in Figure 7.11. A possible reason for this observation is that for small the nozzle-to-target spacing, $H/W = 2.6$, the flow behaves more like channel flow, (Figures 7.2 and 7.3), and there is little effect of the cross-flow on the Nusselt number.
When the cross-flow value is small, \( M < 0.25 \), the decrease in mass of the wall jet for \( H/W = 2.6 \) is faster than that for \( H/W = 8 \) causing a steeper slope of \( \text{Nu}_{\text{ave}} \) for \( M < 0.25 \).

![Figure 7.11 Effect of M on average Nusselt number predicted by the standard k-\( \varepsilon \) and RSM models for \( H/W = 2.6 \)](image)

![Figure 7.12 Effect of M on average Nusselt number predicted by the standard k-\( \varepsilon \) and RSM models for \( H/W = 8 \)](image)

### 7.2.5 Effect of temperature difference between the jet and cross-flow

Figure 7.13 shows the effect of temperature difference between the jet and the cross-flow on Nusselt number predicted by the standard k-\( \varepsilon \) and the Reynolds stress model.
turbulence models. As above, the standard k-ε model and the Reynolds stress model produce similar results for the Nusselt number distributions for the same temperature difference. As expected, the temperature difference between the jet and the cross-flow has a noticeable effect on the Nusselt number distributions. The Nusselt number for the whole flow region increased with the increase of cross-flow temperature; however, the increase is not very significant. For example, when the cross-flow temperature increased from 320 K to 430 K, the maximum Nusselt number increased by only 30% as predicted by both the standard k-ε and Reynolds stress models.

![Figure 7.13](image)

**Figure 7.13** Effect of temperature difference between the jet and cross-flow on the Nusselt number predicted by the standard k-ε model

### 7.3 Conclusions

In this chapter, the effect of jet-to-cross-flow mass ratio and jet angle on both local and average Nusselt numbers under a semi-confined impinging turbulent slot jet at different nozzle-to-target spacings was studied. Analysis of the numerical results shows that the maximum values of the Nusselt number decreased and shifted downstream as M increased from zero to one, while the Nusselt number values in the
downstream region increased. The peak Nusselt number increased and shifted upstream when the jet angle changed from negative 30° to positive 30°. The average Nusselt number decreased with the increase of the cross-flow value, however, with different slopes for different H/W values. Moreover, the temperature difference between the jet and the cross-flow has an effect on the Nusselt number distributions. The maximum Nusselt number increased by 30% when the cross-flow temperature increased from 320 K to 430 K.
Chapter 8
Heat transfer under turbulent multiple slot impinging jets of gas-particle suspension

8.1 Introduction

This chapter presents the heat transfer characteristics for a dilute gas-particle flow through multiple slot impinging jets computed numerically. A new Eulerian-Lagrangian model was employed in the CFD computation. The two-phase turbulent flow is simulated using the standard $k-\varepsilon$ model and Lagrangian particle tracking, which includes coupling terms to simulate the fluid-particle interactions. Version 6 of FLUENT was used to solve the relevant governing equations. User Defined Functions (UDF) capability was used to modify the model to include the conduction heat transfer between the particles and the wall. The numerical results are compared with the available experimental data. It is shown that good agreement between the experiment and simulation is obtained only if the conduction heat transfer due to particle-wall collisions is considered in the model. Effects of wall parameters (properties of wall materials, wall reflection coefficient, and impingement wall temperature), particle factors (particle diameter, particle material properties, and loading ratios) and inlet conditions (jet Reynolds number, and direction of the gravitation vector) on the heat transfer rate are discussed.

The multiple slot impinging jets investigated in this work are symmetric and the flow configuration simulated is shown in Figure 8.1. The following boundary conditions were applied for the different cases considered: i) the impingement surface was specified as a constant temperature wall, ii) the confinement surface is considered to be
an adiabatic wall, iii) the gas and particles were introduced at the inlet with the same and constant values of velocity and temperature, iv) the pressure in outlet was specified to be atmospheric, and v) symmetry boundary conditions were applied along the lines of symmetry. The turbulence intensity at the inlet and outlet for all the calculations was specified to be 2 %. In all of the following numerical simulations, the nozzle width $W$, nozzle-to-target non-dimensional distance $H/W$ and the temperature of inlet particle and gas are fixed at 4 mm, 2 and 300 K, respectively.

**Figure 8.1** Flow configuration of the multiple impinging slot jets

### 8.2 Results and Discussion

#### 8.2.1 Comparison between experiment and simulation

In this section, local heat transfer coefficient predicted by FLUENT 6.0 is compared with the experimental data reported by Yokomine et al. (2002). In Figure 8.2, the local heat transfer coefficient is plotted vs dimensionless length ($X/W$) for non-dimensional jet separation spacing $L/W = 5$ and $L/W = 7$. In Figure 8.2, heat transfer is calculated without considering the conduction heat transfer due to particle wall collision.
From Figure 8.2, it is noted that, for both $L/W = 5$ and $L/W = 7$, the numerical model greatly under-predicts the local heat transfer coefficients under multiple slot impinging jets of gas-particle suspensions. For both the $L/W$ values, the maximum difference between experimental results and numerical results was located at stagnation point and the minimum difference occurred at the exit. The experimental heat transfer coefficient at the stagnation point was nearly 9 times higher than the simulated heat transfer coefficient, while the experimental minimum value of heat transfer coefficient at exit was about 4 times higher than the simulated one. The average experimental heat transfer rates were 6.4 and 5.6 times higher than the numerically predicted values for $L/W = 5$ and $L/W = 7$, respectively.

One plausible reason for this deviation is that the conduction heat transfer between the particle and wall is neglected in the original gas-particle heat transfer model in FLUENT 6. It seems that the conduction heat transfer between particles and the wall is
important in impinging jet flows. We have incorporated conduction heat transfer due to the particle-wall collisions in the simulations described in the following sections.

### 8.2.2 Effect of conduction heat transfer

In this section, the influence of the addition of the conduction heat transfer due to particle-wall collisions is discussed for two multiple impinging jet cases for which the boundary conditions and structural parameters are the same as those of Yokomine’s experiment (2002) and shown in Table 8.1. The total heat transfer $Q$ is plotted for two cases with different $L/W$ values. $Q$ is expressed as:

$$Q = m_g c_{pg} (T_{outg} - T_{inj}) + m_p c_{pp} (T_{outp} - T_{inj})$$  \hspace{1cm} (8.1)

<table>
<thead>
<tr>
<th>$L/W$</th>
<th>$W$ (mm)</th>
<th>$H/W$</th>
<th>$Re_j$</th>
<th>$T_j$ (K)</th>
<th>$T_w$ (K)</th>
<th>$Lo$</th>
<th>$Dp$ (µm)</th>
<th>$\xi$</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>4</td>
<td>2</td>
<td>4600</td>
<td>300</td>
<td>400</td>
<td>0.9</td>
<td>18.3</td>
<td>0.9</td>
</tr>
<tr>
<td>7</td>
<td>4</td>
<td>2</td>
<td>4600</td>
<td>300</td>
<td>400</td>
<td>0.9</td>
<td>18.3</td>
<td>0.9</td>
</tr>
</tbody>
</table>

From Figure 8.3, it is found that the conduction heat transfer due to the particle-wall collisions has a significant effect on the total heat transfer rate. The heat transfer rate increased 5.4 and 6.8 times when the conduction heat transfer due to particle-wall collisions was included for the structural parameters $L/W = 5$ and $L/W = 7$, respectively. Earlier in Figure 8.2, it was shown that the numerical gas-particle models significantly under-predicted the impinging jet heat transfer rate. The average experimental heat transfer rate was 6.4 and 5.6 times higher than the predicted values for $L/W = 5$ and $L/W = 7$, respectively. Therefore, if the additional conduction heat transfer due to particle-wall collisions is considered, the simulated results for both $L/W = 5$ and $L/W = 7$ compared favorably with the experimental data.
The magnitude of the conduction heat transfer between particles and wall depends on many factors including particle and wall properties. The effects of different numerical and experimental parameters such as wall factors (properties of wall materials, wall reflection coefficient, and impingement wall temperature), particle factors (particle loading ratios, particle diameter, and particle material properties), and inlet conditions (jet Reynolds number, and direction of the gravitation vector) are discussed in the following sections. Five materials, graphite, glass, copper, aluminum and steel are studied in this chapter. Table 8.2 lists their properties.

Table 8.2 Material properties of graphite, glass, copper, aluminum and steel

<table>
<thead>
<tr>
<th>Materials</th>
<th>Density (kg/m$^3$)</th>
<th>Heat Capacity (J/kg)</th>
<th>Conductivity (W/m-K)</th>
<th>Elastic Modulus (N/m$^2$)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Graphite</td>
<td>2250</td>
<td>690</td>
<td>24</td>
<td>4.80×10$^9$</td>
</tr>
<tr>
<td>Glass</td>
<td>2460</td>
<td>737</td>
<td>35</td>
<td>8.69×10$^9$</td>
</tr>
<tr>
<td>Aluminum</td>
<td>2705</td>
<td>900</td>
<td>227</td>
<td>6.90×10$^{10}$</td>
</tr>
<tr>
<td>Copper</td>
<td>8940</td>
<td>385</td>
<td>391</td>
<td>1.15×10$^{11}$</td>
</tr>
<tr>
<td>Steel</td>
<td>7870</td>
<td>486</td>
<td>52</td>
<td>2.05×10$^{11}$</td>
</tr>
</tbody>
</table>
8.2.3 Effects of wall factors

8.2.3.1 Effect of wall material

Three wall materials, aluminum, copper and steel, were studied in this work. Their properties are listed in Table 8.2. Figure 8.4 shows the effect of impingement surface materials on heat transfer rate. When the conduction heat transfer contribution was neglected, the wall materials with the same roughness had no influence on the flow and heat transfer behavior. However, as before, when conduction heat transfer was considered, the total heat transfer rate significantly increased, although the heat transfer difference between the three wall materials, aluminum, copper and steel, was small. In other words, Figure 8.4 shows that the thermo-physical properties of the wall material have negligible effect on the heat transfer rate. This is probably due to the fact that the thermal boundary condition (isothermal wall) is forced externally on the wall.

![Figure 8.4](image)
8.2.3.2 Effect of reflection coefficient on heat transfer

In Figure 8.5, it is shown that the wall reflection coefficient has only a secondary effect on heat transfer for both WOS and WS conditions. Under the present boundary conditions, when the conduction heat transfer between particle and wall was included, the maximum heat transfer rate occurred when the reflection coefficient was equal to 0.6. The effect of reflection coefficient on total heat transfer is nonlinear. The reflection coefficient not only affected the flow behavior of gas and suspended particles, but also affected the magnitude of the heat transferred by the conduction between the particle and wall. However in the present study, the heat transfer due to particle wall collision is calculated under the assumption of elastic collision, which makes the study of effect of reflection coefficient is not very practical. Nonetheless, to characterize the effect of gas-particle suspension on heat transfer in impinging jets systems, further fundamental work is needed to correlate the reflection coefficient with the heat transfer rate.

\[ \zeta \]

Figure 8.5 Effect of reflection coefficient on heat transfer for both WOS and WS conditions
8.2.3.3 Effect of impingement wall temperature

It is well known that the temperature difference between the impingement wall and inlet gas and particles not only affects the convection heat transfer but also affects the conduction heat transfer between the particles and the wall. Figure 8.6 shows the predicted heat transfer behavior for three values of the impingement wall temperature for both WOS and WS conditions. Figure 8.6 shows that the heat transfer rate increases with the increase of the impingement wall temperature for both WS and WOS conditions. However, under WS conduction, the heat transfer increased with a higher slope than under the WOS condition. Thus, the conductive contribution appears to be enhanced by larger temperature difference.

![Figure 8.6 Effect of impingement wall temperature on heat transfer](image)

8.2.4 Effects of particle factors on heat transfer

8.2.4.1 Effect of coefficient C and the loading ratio

The heat source (Eq. 3.33) was calculated under the assumptions that the particle-wall collisions were elastic, and the particles were spherical with a fixed diameter. However, these assumptions are not fully valid. So in order to account for inelastic collisions and non-sphericity of the particles, we added a constant coefficient C in
Eq.3.33 to introduce variable weight for the heat source term. First the effects of loading ratio and constant, C, on heat transfer were investigated individually, shown in Figure 8.7 and 8.8, respectively, and then we studied the coupled effects of particle loading ratio and constant C and developed empirical correlations between the average heat transfer rate Q and Lo for two values of C (viz. 1.0 and 1.5) and for both L/W = 5 and L/W = 7.

From Figure 8.7, we find that when conduction heat transfer is included, the particle loading ratio has a significant effect on heat transfer. On the other hand, as expected, the loading ratio has negligible effect on heat transfer if the heat transfer between particles and wall collisions is neglected. From Figure 8.7, it is noted that the heat transfer rate monotonously increased from 2.1 to 12.1 times when the loading ratio increased from 0.3 to 1.8. The experimental study of Yoshida (1990) of heat transfer of a 2D impinging jet with gas-solid suspensions also showed that the Nusselt number increased with the increase of the particle loading ratio in the range of 0 to 0.8. However, for the gas-particle pipe flow, Avila and Cervantes (1995) found that the
average heat transfer rate increased with the increase of loading ratio when the ratio was greater than 1. When the loading ratio was less than 1, a net reduction of heat transfer rate was produced due to the reduction in turbulence intensity and the suspension heat capacity effect was small. From the monotonically increasing heat transfer rate with the increase of particle loading ratio in impinging jet systems, it can be concluded that the main mechanism of heat transfer enhancement in impinging jet gas-particle system was due to the conduction heat transfer between the particles and the wall.

![Graph](image)

**Figure 8.8** Effect of constant C in heat source term on computed heat transfer for both WOS and WS conditions

Figure 8.8 shows the effect of constant, C, in Eq.3.33 on heat transfer rate. The simulations were carried out for three values of C viz. 0.5, 1.0 and 1.5. It was noted that with the increase of the magnitude of C, the total heat transfer, Q, increased. The larger constant results in a larger heat source term; consequently more heat is transferred to the particles from the hotter wall. However, comparing Figures 8.7 and 8.8, it can be seen that the loading ratio has a bigger effect on heat transfer than the
value of C. For example, for a similar rise (1.5 times higher) in C and loading ratio, the
effect was 40% greater for loading ratio. From Figure 8.8, it has been seen that the
total heat transfer rate is sensitive to the constant in Eq. 3.33. When the constant C
increased from 1.0 to 1.5, heat transfer increased by 30%, and these results are in the
right range of the experimental data of Yokomine et al. (2002), indicating that the
value of C probably lies in the narrow range of 0.5 to 1.5.

Figures 8.9a and 8.9b show the effect of particle loading ratio on heat transfer rate for
C = 1.0 and C = 1.5, respectively. Figures 8.10a and 8.10b are the results for L/W = 7.
The resulting empirical correlations between the average heat transfer rate Q and Lo
for two values of C (viz. 1.0 and 1.5) and for both L/W = 5 and L/W = 7 are listed in
Table 8.3.

Table 8.3 Correlations between heat transfer and loading ratio

<table>
<thead>
<tr>
<th>C</th>
<th>Correlations (L/W=5)</th>
<th>R²</th>
<th>Correlations (L/W=7)</th>
<th>R²</th>
</tr>
</thead>
<tbody>
<tr>
<td>C=1.0</td>
<td>$Q = 815Lo^{1.2}$</td>
<td>0.99</td>
<td>$Q = 1175Lo^{0.62}$</td>
<td>0.99</td>
</tr>
<tr>
<td>C=1.5</td>
<td>$Q = 771Lo^{1.56}$</td>
<td>0.88</td>
<td>$Q = 1524Lo^{0.76}$</td>
<td>0.99</td>
</tr>
</tbody>
</table>

As expected, Figures 8.9 and 8.10 show that under the WS condition, the particle
loading ratio has a significant positive effect on the heat transfer rate as the heat
transfer rate increases with increase of the particle loading ratio for both C values, 1.0
and 1.5. On the contrary, the loading ratio has negligible effect on heat transfer if
conduction heat transfer is neglected. The dependence of total heat transfer Q on
particle loading ratios is described empirically as $Q = aLo^b$. Correlations between
particle loading ratio and heat transfer were obtained for both C = 1.0 and C = 1.5 and
both L/W = 5 and L/W = 7. The regression analysis indicates that the loading ratio had
a higher exponent value for $C = 1.5$ than that for $C = 1.0$ for both $L/W$ values. For a higher value of $C$, an increase in the source term (Eq. 9) can be expected, which consequently would increase the total heat transfer. However, these correlations are valid only over the parameter ranges examined in this work.

Figure 8.9a Effect of particle loading ratio on heat transfer for $L/W = 5$, $C = 1.0$

Figure 8.9b Effect of particle loading ratio on heat transfer for $L/W = 5$, $C = 1.5$
The outlet gas temperature distributions were plotted vs outlet width for both WOS and WS conditions in Figure 8.11. It was noted that under WS condition, the outlet gas temperature distributions increase with the increase of the loading ratio, shown in Figure 8.11. Moreover, the outlet gas temperature distributions for the WS cases are all higher than those for WOS conditions. On the other hand, under WOS condition, the outlet gas temperature distributions decreased with increase of the loading ratio. The gas-particle mixture has a higher thermal-capacity with increase of the particle loading.
ratio, therefore, if the heat transfer due to particle-wall collisions is neglected, the outlet gas temperature decreases with the increase of particle loading ratio.

\[ \frac{X}{W} \]

<table>
<thead>
<tr>
<th>Lo</th>
<th>WS</th>
<th>T (K)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.3</td>
<td>0.5</td>
<td>300</td>
</tr>
<tr>
<td>0.9</td>
<td>0.5</td>
<td>305</td>
</tr>
<tr>
<td>1.35</td>
<td>0.5</td>
<td>310</td>
</tr>
<tr>
<td>1.8</td>
<td>0.5</td>
<td>315</td>
</tr>
</tbody>
</table>

\( \text{Lo}=0.3, \text{WS} \quad \text{Lo}=0.9, \text{WS} \quad \text{Lo}=1.35, \text{WS} \quad \text{Lo}=1.8, \text{WS} \)

\[ \text{Re}=4600, \zeta=0.3, \frac{L}{W}=5 \quad \text{Dp}=18.3 \mu m, \text{Graphite} \]

**Figure 8.11** Effect of loading ratio on outlet gas temperature distributions for WOS and WS conditions

8.2.4.2 Effect of particle diameter

Figures 8.12 and 8.13 display the effect of particle diameter on the average heat transfer rate for \( \frac{L}{W} = 5 \) and 7, respectively. For both values of \( \frac{L}{W} = 5 \) and 7, as expected the particle diameter had negligible effect on heat transfer if the particle-wall conduction heat transfer is neglected. By contrast, if the conduction heat transfer between particle and wall is considered, the magnitude of particle diameter had significant effect on heat transfer rate. For example, the smaller the particle diameter, the greater is the heat rate by conduction. Heimann and Schlunder (1986) also reported that the contact heat transfer coefficient from a hot wall to particle is an inverse function of the particle diameter. Their model predicts a heat transfer coefficient that varies approximately as \( 1/\text{Dp} \). In this work, empirical correlations between the heat transfer rate \( Q \) and particle diameter \( \text{Dp} \) were obtained by regression of the numerical results under the WS condition for both \( \frac{L}{W} = 5 \) and \( \frac{L}{W} = 7 \). The correlations are
$Q = 2698Dp^{-0.53}$ and $Q = 5873Dp^{-0.61}$, with $R^2 = 0.97$ and 0.99 for $L/W = 5$ and $L/W = 7$, respectively. It is noted that the exponent values of particle diameter from the regression of numerical data are quite comparable for $L/W = 5$ (-0.5) and $L/W = 7$ (-0.6). The exponent obtained in this work is lower than the Heimann and Schlunder value of approximately -1.0. However, it may be noted that in their case purely conduction heat transfer was considered while the predicted $Q$ in this work includes both conduction and convection contributions due to the presence of particle in the gas.

**Figure 8.12** Effect of particle diameter on heat transfer for $L/W=5$

**Figure 8.13** Effect of particle diameter on heat transfer for $L/W=7$
8.2.4.3 Effect of particle material

From an analysis of the conduction heat transfer between the particle and wall, it is expected that the particle material properties will have important effects on the conduction heat transfer rate due to particle-wall collisions. Figure 8.14 presents the total heat transfer rates of three particle materials, copper, glass and graphite. The choice of the three materials for the present study was determined by their frequent application in the research work. The properties of the three materials studied in this work are listed in Table 8.2. From Figure 8.14, it is noted that the particle material has no effect on the heat transfer rate under WOS condition since conduction contribution is ignored. On the contrary, if the conduction heat transfer between the particle and wall is included, the particle material has significant effect on the heat transfer rate. In this work, the dependency of the total heat transfer \( Q \) on particle properties such as \( \rho \), \( k \) and \( C_p \) can be expressed empirically as

\[
Q = a \left( \frac{\rho}{kC_p} \right)^b
\]

The correlations between particle properties such as \( \rho \), \( k \), \( C_p \) and heat transfer \( Q \) were obtained by regression. The resulting correlation is

\[
Q = 8786 \left( \frac{\rho}{kC_p} \right)^{1.33}
\]

with \( R^2 \) of 0.94 for \( L/W = 5 \). A similar type of correlation was obtained for \( L/W = 7 \), but with a poor fit and a low \( R^2 \). The reason is that both the coefficient and exponent are dependent on parameters of the gas, particles as well as the geometry.
8.2.5 Effect of inlet conditions

8.2.5.1 Effect of inlet Reynolds number

Figure 8.15 shows the computed heat transfer rates for three values of the inlet Reynolds number under both WOS and WS conditions. It is noted that the jet Reynolds number has an effect on the heat transfer rate under both WS and WOS conditions. However, it is seen that the inlet Reynolds number has a much greater effect on the heat transfer rate under the WS condition than under the WOS condition. The heat transfer Q increased by four fold when the Reynolds number increased from 2500 to 10,000 under WS condition. The reason is that higher velocities of impact cause shorter contact times of collision, which favors the conduction heat transfer.
8.2.5.2 Effect of gravity direction on heat transfer

In our previous section of this work, we compared our numerical data with Yokomine’s experimental data (2002). In Yokomine’s experiment, the inlet velocities of gas and particles were in the opposite direction of the gravity. Since in some industrial drying applications, the gas and particles are injected in the channel in the same direction as that of the gravity, some simulations were conducted to study the effect of gravity direction on the heat transfer rate. Some typical results are shown in Figure 8.16. From Figure 8.16, we find that under Yokomine’s experimental condition, when gravity direction changed to the same direction as that of the inlet gas and particle velocities, a slight increase in the heat transferred from the wall occurs. Under WS condition, slightly more heat is transferred from the wall if the gravity direction is the same as that of the inlet gas and particles velocities as gravity assists the particle motion, and higher velocities of impact are expected resulting in higher heat transfer rate.
8.3 Conclusions

A new model of heat transfer was developed in this study to predict the heat transfer for multiple turbulent slot impinging jets of dilute gas-particle suspensions by taking into account the conduction heat transfer due to particle-wall collisions in the Eulerian-Lagrangian model. The predicted results approach experimental observations when conduction heat transfer between the particle and wall is included in the model. The effects of different numerical and experimental parameters such as wall factors (properties of wall materials, wall reflection coefficient, and impingement wall temperature), particle factors (particle loading ratios, particle diameter, and particle material properties), and inlet conditions (jet Reynolds number, and direction of the gravitation vector) are discussed. The simulation results indicate that particle diameter; particle material properties and loading ratio have significant effects on the predicted heat transfer rate when the conduction heat transfer between particle and wall is included in the model. Regression analysis was conducted to develop correlations
demonstrating the effects of particle diameter and the particle materials on the heat transfer. The simulation results indicate that with the conduction heat transfer due to particle collision on the wall, the jet Reynolds number and the impingement wall temperature have larger effects on the total heat transfer rate than heat transfer without the particle-wall conduction. It is also found that more heat would be transferred from the wall if the gas and particles were introduced into the impinging jets oriented in the same direction as gravity, and the magnitude of this increase will depend on the size and density of the particles. Thermo-physical properties of the wall material have negligible effect on the heat transfer rate.
Conclusions

9.1 General conclusions

The flow and heat transfer behavior of turbulent semi-confined impinging slot jets has been simulated using computational fluid dynamics for various configurations, operating parameters and fluid properties. Predictive performance of turbulence models such as the standard k-ε and RSM models, and different wall functions such as standard and non-equilibrium wall functions were compared with experimental data. The simulated results compare favorably with the experimental results for large nozzle-to-target spacing (e.g. H/W = 12), but the predictions for small nozzle-to-target spacing (e.g. H/W = 2.6) call for further improvement.

Different definitions of Nusselt number have been evaluated and the corresponding distributions compared for small and large ∆Ts between the jet and the target surface. The local Nusselt number for an isothermal impingement surface can be defined as $Nu_s = \frac{h_x W}{\lambda}$, where $\lambda$ is the fluid thermal conductivity, $h_x = \frac{q}{\Delta T}$, and $\Delta T$ is the temperature difference between the jet and impingement surface. In this work, $\lambda$ was calculated at jet temperature, film temperature and impingement wall temperature and the corresponding Nusselt numbers were denoted by $Nu_j$, $Nu_f$ and $Nu_w$, respectively. Comparison among the three Nusselt definitions shows that small temperature differences (e.g. up to 50°C) show only minor differences in the local $Nu_j$, $Nu_f$ and $Nu_w$ values. However, large temperature differences (over 100°C) can result in
significant differences among the three types of Nusselt numbers for both heating and cooling. It was shown that the jet Nusselt number, $N_{uj}$, was the best choice among the three Nusselt number definitions to report the heat transfer rate since under large $\Delta T$, the jet Nusselt number spreads the least. The jet Nusselt number definition allows the designer to use previously published correlations, which were obtained at small $\Delta T$. This conclusion is valid regardless of the jet Reynolds number and the nozzle-to-surface spacing.

The effect of fluid Prandtl number on local, stagnation and average heat transfer of semi-confined laminar and turbulent impinging slot jet was investigated over the range of Prandtl number 0.7-71. The variations of Nusselt number for all the tested fluids are similar to those for air. The Nusselt number increases with the increase of fluid Pr number from 0.7 to 71 for both laminar and turbulent flows. Gases with similar Prandtl number exhibit similar values of Nusselt number, but different values of the surface heat transfer coefficient due to their widely different thermal conductivities. Empirical correlations are developed for the stagnation and average Nusselt numbers as a function of the fluid Prandtl number on the basis of extensive numerical experiments for both laminar and turbulent flows.

The flow and heat transfer characteristics of a single semi-confined turbulent slot jet of air impinging normally or obliquely into an imposed air cross-flow of the same or different temperature were simulated using the standard $k-\varepsilon$ model and Reynolds stress model. The influence of various flow parameters (e.g. jet-to-cross-flow mass ratio) and geometric parameters (e.g. nozzle-to-target spacing and jet angle) were evaluated at a fixed Reynolds number for equal and unequal temperatures of the jet and cross-flow. It
was found that the maximum values of the Nusselt number decreased and shifted downstream as $M$ increased from zero to one. On the other hand, the peak Nusselt number increased and shifted upstream when the jet angle changed from negative $30^\circ$ to positive $30^\circ$. The average Nusselt number decreased with the increase of the cross-flow value with different slopes for different H/W values. Moreover, the temperature difference between the jet and the cross-flow has an effect on the Nusselt number distributions.

Heat transfer in a dilute gas-particle suspension flow through multiple slot impinging jets has been evaluated using a new Eulerian-Lagrangian method, which accounts for the conduction heat transfer due to particle-wall collisions. The numerical results were compared with the experimental data of Yokomine et al. (2002). Good agreement between the experiment and simulation is obtained only if the additional conduction heat transfer due to particle-wall collisions is considered in the model. The effects of particle diameter, particle and wall material properties and particle loading ratios, jet Reynolds number, wall reflection coefficient, impingement wall temperature and direction of the gravitation vector are discussed. The simulation results indicate that particle diameter, particle material properties and loading ratio have significant effects on the predicted heat transfer rate only if the conduction heat transfer between particle and wall was included. With the conduction heat transfer due to particle collision on the wall, the jet Reynolds number and wall temperature have larger effects on the heat transfer rate than under WOS condition. In addition, it is found that more heat would be transferred from the wall if the gas and particles were introduced into the impinging jets oriented in the same direction as gravity.
9.2 Major contributions to knowledge

As a result of the present study, the following contributions have been made to the existing knowledge:

1. In view of the engineering practice, the results provided here can be used by the design engineers to estimate the heat transfer rates and effects of various parameters on impinging jet systems, which were not reported earlier. In addition, simple correlations are developed to use for systems with large temperature difference between the jet and impingement surface, correlations for higher Prandtl number fluids, and impinging jet flow with gas particle suspensions.

2. The numerical results generated in this work provide comparative analysis of relative merits and limitations of commonly used turbulence models in engineering practice. In addition, a new model is proposed which can form the basis of a future effort in developing a model for gas particle heat transfer in turbulent flow.

9.3 Recommendations for future work

This study shows that commercial CFD software such as FLUENT solves the impinging jet problems reasonably well. However, simulated results indicate some deviations from the experimental data near the stagnation point, especially for small nozzle-to-target spacing. Thus, there is thus room for improvement in the simulation. More work is needed to address this issue from two aspects: (i) develop modified turbulence models, and (ii) explore some other existing models such as large-eddy simulation (LES) model and direct numerical simulations (DNS) approach.
The numerical studies for other nozzle geometries, for example, round jet, square jet, are needed under large $\Delta T$. It remains to be shown whether the jet Nusselt number is still a best choice for other jet geometries.

Empirical correlations for stagnation and average Nusselt numbers as a function of the fluid Prandtl number had been obtained. More work is needed to obtain empirical correlations for stagnation and average Nusselt numbers as functions of not only fluid Prandtl number but also the geometric parameters, e.g. $Nu_0 (or Nu_{ave}) = a \Pr^b \left( \frac{H}{W} \right)^c$.

It is shown in this work that the conduction heat transfer between the particle and wall has a significant effect on the heat transfer characteristics of impinging jet flow with dilute particle suspensions. To characterize the effect of gas-particle suspension on heat transfer in impinging jets systems, further fundamental work, both experimental and modeling, is needed to correlate the reflection coefficient with the heat transfer rate.


References


References


References


References


References


References


Publications


