Numerical Investigation of Heat Transfer and Fluid Flow around the Rectangular Flat Plane Confined by a Cylinder under Pulsating Flow

G. Li†, Y. Zheng, H. Yang and Y. Xu

Department of Energy and Environment System Engineering, Zhejiang University of Science and Technology, Hangzhou, 310023, P. R. China

†Corresponding Author Email: gnlene@163.com

(Received October 8, 2014; accepted October 14, 2015)

ABSTRACT

Fluid flow around and heat transfer from a rectangular flat plane with constant uniform heat flux in laminar pulsating flows is studied, and compared with our experimental data. Quantitatively accurate, second-order schemes for time, space, momentum and energy are employed, and fine meshes are adopted. The numerical results agree well with experimental data. Results found that the heat transfer enhancement is caused by the relative low temperature gradient in the thermal boundary layer, and by the lower surface temperature in pulsating flows. In addition, the heat transfer resistance is much lower during reverse flow period than that during forward flow period. The flow reversal period is about 180 degree behind the pulsating pressure waves. Besides, spectrum results of the simulated averaged surface temperature showed that the temperature fluctuates in multiple-peaked modes when the amplitude of the imposed pulsations is larger, whereas the temperature fluctuates in a single-peaked mode when the amplitude of the imposed pulsation is small.

Keywords: Pulsating flow; Heat transfer enhancement; Computational fluid dynamics.

NOMENCLATURE

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>pulsating velocity component</td>
</tr>
<tr>
<td>D</td>
<td>diameter of the airflow tunnel</td>
</tr>
<tr>
<td>cp</td>
<td>heat capacity of air</td>
</tr>
<tr>
<td>f</td>
<td>pulsating frequency</td>
</tr>
<tr>
<td>h</td>
<td>convection coefficient</td>
</tr>
<tr>
<td>H</td>
<td>height of the airflow tunnel</td>
</tr>
<tr>
<td>L</td>
<td>length of the plate heater</td>
</tr>
<tr>
<td>Nu=hl/λ</td>
<td>Nusselt number</td>
</tr>
<tr>
<td>Nu=Nup/Nus</td>
<td>heat transfer enhancement factor</td>
</tr>
<tr>
<td>p</td>
<td>pressure amplitude</td>
</tr>
<tr>
<td>q</td>
<td>input heat flux</td>
</tr>
<tr>
<td>Re</td>
<td>Reynolds number</td>
</tr>
<tr>
<td>Ro,ad</td>
<td>time averaged heat resistance</td>
</tr>
<tr>
<td>Ro,av</td>
<td>space averaged heat resistance</td>
</tr>
<tr>
<td>Ro,ave</td>
<td>total averaged heat resistance</td>
</tr>
<tr>
<td>S</td>
<td>area of the plate heater</td>
</tr>
<tr>
<td>sh</td>
<td>heat source</td>
</tr>
<tr>
<td>St</td>
<td>Strouhal number</td>
</tr>
<tr>
<td>α</td>
<td>absorptivity of the heater</td>
</tr>
<tr>
<td>λ</td>
<td>thermal conductivity of air</td>
</tr>
<tr>
<td>ρ</td>
<td>density of air</td>
</tr>
<tr>
<td>ε</td>
<td>emissivity of the heater</td>
</tr>
<tr>
<td>ν</td>
<td>kinematic viscosity</td>
</tr>
<tr>
<td>σb</td>
<td>Stefan-Boltzmann const</td>
</tr>
<tr>
<td>σt</td>
<td>thermal boundary layer thickness</td>
</tr>
<tr>
<td>θ</td>
<td>phase angle</td>
</tr>
<tr>
<td>s</td>
<td>steady</td>
</tr>
</tbody>
</table>

Subscripts

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>p</td>
<td>pulsation</td>
</tr>
<tr>
<td>rad</td>
<td>radiation</td>
</tr>
<tr>
<td>rms</td>
<td>root mean square</td>
</tr>
</tbody>
</table>

s
1. INTRODUCTION

The fluid flow and convective heat transfer in pulsating flows is encountered in many engineering applications, such as the compact heat exchanger, cooling system of nuclear reactor, pulse combustor, pulse tube dry cooler, electronic cooling system and arterial flows inside human body. Therefore, great interest is being attracted to find out in what degree does the pulsation of airflow enhances the heat transfer process. Many analytical, experimental and numerical studies exist. However, due to its complicated nature, i.e. many parameters of pulsating flows affect the heat transfer process, previous literatures continues as to whether the pulsation enhance the heat transfer process or not. In those works concluded in positive effects by pulsation, the exactly physical mechanism of in what way does the pulsation enhance the heat transfer process is still unclear, and needs further investigations. Detail CFD (Computational Fluid Dynamics) studies are needed to find out this mechanism. However, even CFD studies of heat transfer in pulsating flows are difficult until the large computer resources could be available in recent years. CFD studies of heat transfer need fine computational grid, the length of which should be much smaller than the thickness of thermal boundary layer.

Among those important pulsating parameters affecting the heat transfer process, the pulsating frequency and pressure amplitude are two most important ones. However, previous works show a variety of contradictory results. In the experimental studies, (Habib et al. 2002) investigated the heat transfer enhancement of a tube with uniform heat flux in laminar pulsating flows at various conditions (f=1-29.5 Hz, Re=780-1987). A heat transfer model $Nu = Nu(f, Re)$ was developed with the deviation of about 10% in different range of Reynolds number. (Dec et al.1992) studied the heat transfer rates in pulse combustion tail pipes in a series of conditions (f=67-101 Hz, Re=3750). It’s concluded that the pressure amplitude must be strong enough to create flow reversal in order to obtain positive heat transfer enhancement factor. (Kearney et al. 2001) investigated the time resolved structure of a thermal boundary layer in laminar pulsating channel flow, concluding that the flow reversal is not necessary for heat transfer enhancement. A few years later, (Moon et al. 2005) confirmed Kearney’s conclusion (Kearney et al. 2001). Moon’s work reported heat transfer enhancement from a rectangular heated block array in a pulsating channel flow at various conditions (f=10-100 Hz, A=0.2-0.3). Results showed that the heat transfer enhancement factor can be larger than 1.2. Many other valuable experimental studies reported an enhancement in heat transfer due to the pulsation (Zohir et al. 2006, Ji et al. 2008, Baffigl and Bartoli 2010). However, some studies such as Elsayed et al.’s work (Elshefei et al. 2008), reported the pulsation would downgrade the heat transfer compared to that in steady flow.

In analytical studies, (Faghi et al. 1979) reported that larger heat transfer enhancement factor could be obtained at higher pulsating frequency. However, (Hemida et al. 2002) found that the heat transfer enhancement factor increases with pressure amplitude, but decreases with pulsating frequency. Some studies (Chang and Tucker, 2004, Chattopadhyay et al. 2006) reported that the pulsation has no effects on the heat transfer process. Other works presented valuable analysis about the influences of Prandtl number (Moon et al. 2005), phase lag and surface temperature distribution (Nika et al. 2007) on the heat transfer process. In numerical studies, (Thyageswaran 2004) developed a near-wall turbulence model for heat transfer in pulsating flow. (Wang and Zhang 2005) found that there is an optimum Womersley number at which heat transfer is maximally enhanced, which is confirmed by Akdag’s work (Akdag 2010). (Selimefendigil et al. 2012) investigated the nonlinearity of the unsteady heat transfer of a cylinder in pulsating cross flow. They developed an accurate, low-order model for the heat source dynamics in a Rijke combustor, which is studied carefully in one of our previous work (Li et al. 2008).

The mechanism controlling the heat transfer enhancement in pulsating flows is unclear, and needs to be further explored. Experimental data is limited, and it is difficult to measure the velocity/temperature profile inside the flow/thermal boundary layer. Few CFD studies exist (Thyageswaran 2004, Wang and Zhang 2005, Akdag 2010, Selimefendigil et al. 2012), and more detail analysis is obvious needed, such as the pulsating mode of the temperature field, time evolution of thermal boundary layer and space distribution of heat transfer resistance. In this work, a detail CFD study was carried out to explore the mechanism controlling the heat transfer in pulsating flows. Numerical results were compared with our experimental data, and detail discussions were made.

2. NUMERICAL MODEL

This part presents the details of the governing equations and the boundary conditions. A brief introduction on the experimental setup and experimental procedures is also presented in order to make the present studied problem more specified.

2.1 Governing Equations

Prediction of the fluid flow and heat transfer in pulsating flows requires CFD computation with a compressible flow model. However, such computations demand huge computational resources, since the heat transfer in pulsating flows involve various physical phenomena covering a wide range of length and time scales. Considering that the pulsation wave lengths are much larger than the characteristic length of the plane heater, the unsteady, incompressible Navier-Stokes equation along with the energy equation is solved with ANSYS-FLUENT 13.0 segregated solver, as follows,

\[ \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0 \]

\[ \frac{\partial (\rho \mathbf{u})}{\partial t} + \nabla \cdot (\rho \mathbf{u} \mathbf{u}) = -\nabla p + \nabla \cdot (\tau) + \rho \mathbf{g} \]

\[ \frac{\partial \rho e}{\partial t} + \nabla \cdot (\rho \mathbf{u} e) = \nabla \cdot (k \nabla T) + \rho h \]

where \( \rho \) is the density, \( \mathbf{u} \) is the velocity, \( p \) is the pressure, \( \rho e \) is the total energy, \( k \) is the thermal conductivity and \( h \) is the heat generation rate.
Continuum equation
\[
\frac{\partial u_i}{\partial x_j} = 0
\]  
(1)

Momentum conservation equation
\[
\frac{\partial u_i}{\partial t} + \frac{\partial (u_i u_j)}{\partial x_j} = \frac{1}{\rho} \frac{\partial p}{\partial x_i} + \nu \frac{\partial^2 u_i}{\partial x_j^2} + s_h
\]  
(2)

Energy conservation equation
\[
\frac{\partial T}{\partial t} + \frac{\partial (u_i T)}{\partial x_j} = \frac{k}{\rho c_p} \frac{\partial^2 T}{\partial x_j^2} + q
\]  
(3)

where \( i, j = 1, 2, 3 \), \( \rho \) is the air density, \( c_p \) is the heat capacity, \( u \) is air flow velocity vector, \( x \) denotes the spatial coordinate, \( p \) stands for static pressure, \( T \) is the static temperature. Combining with very fine meshes, the laminar flow method gives direct numerical simulation results. In this work, relative fine meshes, second-order schemes for time, space, momentum and energy, are used. Such computations allow week turbulence to be properly solved under the present Reynolds number number of 2109. Radiation is taken into consideration, and the “PI” model is employed. The plane heater is supposed to be a grey obstacle, and the emissivity \( \varepsilon \) is set to be 0.68 (Jones et al. 1977). The heat source in Eq. (3) is as follows (ANSYS Inc. 2010),
\[
s_h = \frac{\varepsilon \sigma_b T^4 - \varepsilon \sigma_b T_0^4}{\rho c_p} \]  
(4)

where \( \sigma_b \) is the Stefan-Boltzmann constant, \( \varepsilon \) and \( \sigma \) is the emissivity and absorptivity of heater, respectively.

2.2 Boundary Conditions

The computational zone and the computational meshes in the crossing section of the plane heater are shown in Fig. 1, which is consistent with the experimental test section (Li et al. 2014). The computational zone is a circle channel with inner diameter \( D=68 \, \text{mm} \) and height \( H=210 \, \text{mm} \). A rectangular plane heater with dimensions of 70 mm \( \times \) 20 mm \( \times \) 1.5 mm is fixed in the middle of the channel. The 70 mm side of the heater is aligned with channel axis. The Reynolds number based on the averaged channel velocity and the channel diameter is 2109. Considering that the computations should cover detail information inside the boundary layer, the length scale of the first grids around the heater fulfills the following demands (Breuer et al. 2003)
\[
y_p y^+ = 0.283 \, \text{mm} \]  
(5)
\[
y_0 A(w/c_0)^{0.5} \]  
(6)

where \( y_p \) is the distance to the wall from the adjacent grid centroid, \( u_0 \) is the averaged velocity inside the circle channel, and \( b_0 \) denotes the distance along the wall from the starting point of the boundary layer. Block partition grid method is employed to save computer resource and computation time. The number of computational meshes is about 2,500,000 with emphasize around the plane heater. The effects of the \( y_p / y^+ \) on the numerical results when \( p_{\text{inlet}}=75 \, \text{Pa} \) is shown in Table 1. As shown in Table 1, \( y_p / y^+ = 0.35 \) is fine enough since the temperature difference is less than 0.5 K for the case of \( y_p / y^+ = 0.28 \).

<table>
<thead>
<tr>
<th>( y_p / y^+ )</th>
<th>Grow Factor</th>
<th>Number of Grow</th>
<th>Transition pattern</th>
<th>( T_w ) (K)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.28</td>
<td>1.05</td>
<td>16</td>
<td>4.2</td>
<td>333.1</td>
</tr>
<tr>
<td>0.35</td>
<td>1.05</td>
<td>16</td>
<td>4.2</td>
<td>333.4</td>
</tr>
<tr>
<td>0.63</td>
<td>1.05</td>
<td>16</td>
<td>4.2</td>
<td>334.7</td>
</tr>
<tr>
<td>0.78</td>
<td>1.05</td>
<td>16</td>
<td>4.2</td>
<td>335.6</td>
</tr>
<tr>
<td>1.17</td>
<td>1.05</td>
<td>16</td>
<td>4.2</td>
<td>337.0</td>
</tr>
</tbody>
</table>

Temperature depended properties of air (Yang and Tao 1998) including density, specific heat capacity, thermal conductivity and viscosity, are used with several UDF (User Define Function) programs. At the inlet and outlet, “velocity inlet” and “pressure outlet” boundary conditions are imposed, respectively. The pulsation is imposed to the inlet using a UDF program to create pressure oscillations (Roux et al. 2011), as follows,
\[
u(t)=u_0(1+\sin(2\pi ft))
\]  
(7)

where \( A \) denotes the pulsating component of velocity at the inlet surface. All UDF programs are written in C language, and interpret into the main program. The “heat flux wall” boundary condition is employed to describe the heated plane, in which the wall temperature is calculated with the following equation (ANSYS Inc. 2010),
\[
T_w = \frac{q - q_{\text{rad}}}{h} + T
\]  
(8)

where \( T_w \) is the wall temperature, \( q \) is total heat flux from the heated plane, which equals to 4.00 W in this work. \( q_{\text{rad}} \) is the radiation heat flux, which equals to \( s_b \) in Eq. (4). \( h \) denotes the convection heat transfer coefficient. In this work, the pulsating frequency is maintained at 15 Hz, and \( A \) is modified to produce different pressure amplitudes (20-175
Pa) according to experimental conditions. Time step is set to be $10^{-3}$ s. The global convergence for continuity, momentum and energy residuals are set to be $10^{-4}$, $10^{-5}$, and $10^{-7}$, respectively.

Details of the experimental setup and the experimental procedures can be found in our previous studies (Li et al. 2014, Li et al. 2013). In brevity, the air flow rate is set to be $1,666.7$ ml/s by a mass flow controller. Two-stage perforated plates are installed to insure the uniformity of inlet airflow. A High Temperature Co-fired Ceramics (HTCC) rectangular plane heater with same dimensions as Fig. 1 is fixed in the middle of the channel. A loudspeaker is installed upstream the inlet to produce pulsating waves. A corrugated vibration-absorptive tube is installed to avoid vibration transmission from the loudspeaker to the HTCC heater. A pressure transducer is installed to measure the pressure waves. Then spectrum analysis is carried out to find out the pulsating frequency and pressure amplitude. The power of the HTCC heater is kept to be 4.00W, and the pulsating frequency is maintained at $f=15$ Hz, and the pressure amplitude, $p_{rms}$, is varied in the range of 25-175 Pa. All measured experimental data have an inaccuracy less than 4.1% (Li et al. 2014).

3. RESULTS AND DISCUSSIONS

Verification of the CFD model is presented at first with the help of experimental data. Then the temperature distributions and the temperature pulsating mode are discussed. Finally, the mechanism of heat transfer enhancement in pulsating flows is explored.

3.1 Verification of CFD Model

The simulated averaged surface temperature and the experimental data is shown in Fig. 2. The heat transfer enhancement factor $Ntu$ can be calculated with above temperature data, which is shown in Fig. 3. The calculation follows the following formula.

$$ q = \varepsilon \delta_0 (T^4 - T_0^4) + h(T - T_0) $$

$$ Ntu = \frac{hL}{\lambda} $$

$$ Ntu_p = \frac{q_p - \varepsilon \delta_0 (T_p^4 - T_0^4) \lambda_r (T_p - T_0)}{q_s - \varepsilon \delta_0 (T_s^4 - T_0^4) \lambda_r (T_p - T_0)} $$

where $q_p$ and $T_p$ are the consumed power of the plane heater and the averaged surface temperature of the HTCC heater in pulsating airflow, respectively, whereas the $q_s$ and $T_s$ are the corresponding values in steady airflow, respectively. $\lambda_r$ and $\lambda_t$ are the heat conductivity of the air in pulsating airflow and in steady airflow, respectively. $T_0$ is the temperature of inlet airflow, and it is maintained at 296.5 K. As shown in Fig. 2, the simulated temperature agrees well with the experimental data. However, CFD computations over-predict (about 3.0 K, less than 1.0%) the averaged surface temperature in pulsating conditions with large pressure amplitude ($p_{rms}$ > 60 Pa). As a result, the simulated heat transfer enhancement factor is under-predicted (about 10.0%) in those cases, shown in Fig. 3. This finding reveals that the accurate prediction of temperature field is critical to predict the heat transfer enhancement factor.

![Fig. 2. Comparison between simulated averaged surface temperature and experimental data.](image1)

![Fig. 3. Comparison between simulated heat transfer enhancement factor and experimental data.](image2)
large ($p_{\text{rms}} \geq 100 \text{Pa}$). The peak-peak amplitude (about 3.0 K to 7.0 K) of the averaged surface temperature decreases firstly, and then increases with the pressure amplitude. The corresponding spectrum analyzed results are shown in Fig. 5. The temperature amplitude at 15 Hz decreases firstly, and then increases with pressure amplitude. However, the temperature amplitude at 30 Hz is larger than that at 15 Hz when the pressure amplitude lies between 100 Pa and 175 Pa. This phenomenon is accompanied by the discrepancy between simulated averaged surface temperature and the experimental data. However, there are no evidences to correlate these two phenomena, and this needs to be further studied.

3.2 Temperature Field

The averaged temperature field in the plane of $x=0$ m is presented in Fig. 6. In the steady flow condition, the temperature increases in the $z$ direction near the wall surface, and the temperature distribution shape is consistent with those in textbooks. In the pulsating condition, the temperature field is an averaged temperature field from 66 instant temperature fields within a pulsation cycle, since the time step and pulsating frequency of the CFD computation is $10^{-3}$ s and 15 Hz, respectively. Obvious difference can be found in Fig. 6. The temperature level of the averaged temperature field in pulsating condition is much lower than that in steady flow condition. Therefore, it’s worth to probe into the pulsation mode of the temperature field in pulsating condition. Firstly, a pulsation cycle was selected when the pressure amplitude equals 75 Pa, which is shown in Fig. 7. Fig. 7 also presents the time evolution of $z$-velocity. Secondly, eight instants are selected, which are shown in Fig. 7, marked with small circle.

![Fig. 4. Time history of the simulated averaged surface temperature.](image1)

![Fig. 5. Spectrum results of the simulated averaged surface temperature pulsations.](image2)

![Fig. 6. Temperature field of the steady state and the averaged temperature field of the pulsating state when $p_{\text{rms}}=75 \text{ Pa}$.](image3)

![Fig. 7. One pulsation cycle for the study of temperature field when $p_{\text{rms}}=75 \text{ Pa}$.](image4)
the downstream zones in the second half cycle when the flow is in reversal.

This implies that the flow reversal occurred in the second half cycle, in other word, the flow reversal is inconsistent with the positive pressure waves. One possible mechanism for the displacement between the oscillating temperature and the flow reversal may caused by the inconsistent between the pressure-induced pulsating velocity and the pulsating pressure during one cycle. The highest pressure occurs when the higher temperature fluid is in the upstream zones, acting as a piston to the incoming fluid. This is confirmed in Figs. 7-8. According the previous studies (Pedley, 1976), the actual pulsating velocity amplitude departs from the pressure-based pulsating velocity amplitude excited by pressure gradient when the Strouhal number $St > 0.3$, and lags behind the pulsating pressure waves. In the present work, the estimated Strouhal number is larger than 2.0, therefore, the flow reversal lags behind the pulsating pressure waves, and causing the above oscillating temperature mode. The distribution of simulated temperature near the left side of plane heater is shown in Fig. 9. In steady flow condition, the temperature increases sharply along the heater, and then increased in a tender way. However, the temperature distribution is very flat in the pulsating condition. In order to find out the mechanism controlling the heat transfer enhancement in pulsating flows, thermal boundary needs to be studied. The averaged thickness of thermal boundary layer in pulsating condition is much larger than that in steady flow condition, which is shown in Fig. 10. Considering that the heat transfer resistance is direct proportional with the temperature difference, and is inverse proportional with boundary layer thickness. Therefore, the heat transfer resistance in pulsating condition is much less than that in steady flow condition.

$$R_{h,d} = \frac{\int T_h(z,t) - 299.5}{\delta(z,t)} dt$$ (12)
\[ R_{\text{avg}} = \int_0^{T} \frac{T_w(z,t) - 299.5}{\delta(z,t)} \, dz \]  
\[ R_{\text{ave}} = \int_0^{T} \frac{T_w(z,t) - 299.5}{\delta(z,t)} \, dz \, dt \]

where the temperature of 299.5 K is the definition of the locations of thermal boundary, and equals to 101\% of the inlet temperature. \( R_{\text{ad}}, R_{\text{av}} \) and \( R_{\text{ave}} \) are the time averaged heat transfer resistance, space averaged heat transfer resistance and total averaged heat transfer resistance, respectively. \( T_w \) is the surface temperature, and \( \delta \) is the thickness of thermal boundary layer. Time averaged heat transfer resistance and space averaged heat transfer resistance are shown in Fig. 11 and Fig. 12, respectively. In steady flow condition, the curves in Figs. 11-12 are calculated from Fig. 6(a). In pulsating condition, the curves in Figs. 11-12 are calculated from 66 instant cases within a complete cycle shown in Fig. 7. It should be reminded that Fig. 11 shows the spatial distribution of the heat transfer resistance and Fig. 12 shows the time evolution of the heat transfer resistance.

![Fig. 12. Evolution of the simulated heat resistance in pulsation cycle when \( p_{\text{rms}}=75 \text{ Pa} \).](image)

4. CONCLUSION

The heat transfer from a rectangular plane heater in steady and pulsating flows is explored numerically and experimentally. Fine computational meshes, second-order schemes for time, space, momentum and energy, are employed. The numerical results agree well with experimental data. Several conclusions are drawn as follows,

(1) Heat transfer enhancement in pulsating flows is generated by relative low temperature gradient in the thermal boundary layer, and by lower surface temperature.

(2) Heat transfer resistance is much lower during the reverse flow period than that during forward flow period. In addition, the flow reversal period is about 180 degree behind the pulsating pressure waves.

(3) The simulated space averaged surface temperature of the plane heater is time variant, and fluctuates in multiple-peaked modes when the amplitude of the imposed pulsation was larger enough.

ACKNOWLEDGEMENTS

This work has been supported by the National Natural Science Foundation of China (51476145, 51476146, 51106140), the Zhejiang Provincial Natural Science Foundation of China (Z1110695),
and the Cross Research Foundation of Zhejiang University of Science and Technology (2013JC02Z).

REFERENCES


ANSYS Inc. (2010). ANSYS FLUENT user's guide.


