CFD calculation of convective heat transfer coefficients and validation – Part I: Laminar flow

Citation for published version (APA):

Document status and date:
Published: 01/01/2006

Document Version:
Publisher’s PDF, also known as Version of Record (includes final page, issue and volume numbers)

Please check the document version of this publication:

• A submitted manuscript is the version of the article upon submission and before peer-review. There can be important differences between the submitted version and the official published version of record. People interested in the research are advised to contact the author for the final version of the publication, or visit the DOI to the publisher’s website.
• The final author version and the galley proof are versions of the publication after peer review.
• The final published version features the final layout of the paper including the volume, issue and page numbers.

Link to publication

General rights
Copyright and moral rights for the publications made accessible in the public portal are retained by the authors and/or other copyright owners and it is a condition of accessing publications that users recognise and abide by the legal requirements associated with these rights.

• Users may download and print one copy of any publication from the public portal for the purpose of private study or research.
• You may not further distribute the material or use it for any profit-making activity or commercial gain
• You may freely distribute the URL identifying the publication in the public portal.

If the publication is distributed under the terms of Article 25fa of the Dutch Copyright Act, indicated by the “Taverne” license above, please follow below link for the End User Agreement:
www.tue.nl/taverne

Take down policy
If you believe that this document breaches copyright please contact us at:
openaccess@tue.nl
providing details and we will investigate your claim.

Download date: 13. Nov. 2019
CFD calculation of convective heat transfer coefficients and validation – Part I: Laminar flow

Annex 41 – Kyoto, April 3rd to 5th, 2006

Adam Neale¹, Dominique Derome¹, Bert Blocken² and Jan Carmeliet²,³

¹) Dep. of Building, Civil and Environmental Engineering, Concordia University, 1455 de Maisonneuve blvd West, Montreal, Qc, H3G 1M8, corresponding author e-mail: aneale@sympatico.ca
²) Laboratory of Building Physics, Department of Civil Engineering, Katholieke Universiteit Leuven, Kasteelpark Arenberg 40, 3001 Heverlee Belgium
³) Building Physics and Systems, Faculty of Building and Architecture, Technical University Eindhoven, P.O. box 513, 5600 MB Eindhoven, The Netherlands

Abstract

Computational Fluid Dynamics (CFD) simulations of convective heat transfer are considered to be particularly challenging to perform by the CFD community. In this paper, the calculation of convective heat transfer coefficients ($h_c$) by the commercial CFD code *Fluent* is verified by studying two cases of laminar flow between parallel infinite flat plates under different thermal conditions. In the first case, the plates produce a constant heat flux, $q_w$, with a constant free stream temperature. In the second case, the walls are at a constant temperature, $T_w$, with a constant free stream temperature. A grid sensitivity analysis with Richardson extrapolation was performed for both cases to determine the grid independent solutions for $h_c$. The values for $h_c$ reported by Fluent were then compared with analytical values from literature. The percentage error between the analytical and grid independent solutions for $h_c$ is on the order of $10^{-2}$%.

1. Introduction

The surface coefficients for heat and mass transfer ($h_c$ and $h_m$, respectively) are parameters that are generally not easily calculated analytically and difficult to derive from experimental measurements. The values of surface coefficients depend on many variables – flow field, boundary conditions, material properties, etc. In addition, despite the fact that the two transfer processes are mutually dependent, they are often solved as uncoupled phenomena. Finally, although existing
correlations relating $h_c$ and $h_m$ are valid for specific cases, such correlations are applied widely throughout literature.

This paper is the first of a two part study of the option to solve for $h_c$ using Computational Fluid Dynamics (CFD). Part I validates the CFD code *Fluent* for heat transfer in the laminar regime using two cases: 1) parallel flat plates with constant wall temperature and 2) parallel flat plates with constant heat flux. In addition, the heat transfer coefficients calculated with several grid refinements will be compared in a grid sensitivity analysis with Richardson extrapolation. The grid independent solution is compared to analytical values.

Part II is a comparative study of heat transfer coefficients calculated using the different turbulence models implemented in *Fluent*. The validity of using wall functions for natural convection cases is also examined.

2. Description of case studies and analytical solutions

2.1. Geometry and boundary conditions

The cases studied in this paper assume that the flow field has become fully developed before the heated region. This assumption is valid when

$$\frac{\partial u}{\partial x} = 0$$

where $u$ is the horizontal component of the velocity at any given height in the flow field (for horizontal plates). Aerodynamically developed flow is a requirement for analytical solution of the thermal boundary layer (Lienhard and Lienhard 2006).

The material properties used in the simulations and the analytical solutions are shown in Table 1. The two cases studied are illustrated in Figure 1.

<table>
<thead>
<tr>
<th>Material properties for air</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Density</strong></td>
</tr>
<tr>
<td><strong>Dynamic Viscosity</strong></td>
</tr>
<tr>
<td><strong>Thermal Conductivity</strong></td>
</tr>
<tr>
<td><strong>Heat Capacity</strong></td>
</tr>
</tbody>
</table>
2.2. Reference temperatures

The goal of the heat transfer simulations is to find the convective heat transfer coefficient $h_{cx}$ at a particular location $x$. This relationship is defined as:

$$q_{wx} = h_{cx} \left( T_{wx} - T_f \right)$$  \hspace{1cm} (2)

where $q_{wx}$ is the heat flux at the wall at $x$, $T_{wx}$ is the temperature of the wall at $x$, and $T_f$ is a reference temperature within the fluid. The actual value used for $T_f$ depends largely upon the geometry used in the problem. An improperly assigned reference temperature can yield a significant error, as will be shown in the case studies presented. Three reference temperatures are used in a comparison exercise to show the effects on the calculation of $h_c$: a constant reference temperature $T_{ref}$ (as used in Fluent to report $h_c$ values), the centerline temperature $T_c$ (taken at $y=0$ on Figure 1), and a bulk temperature $T_b$ which is defined as (Lienhard and Lienhard 2006):

$$T_b = \frac{\int y \rho c_p u T dy}{\dot{m} c_p}$$  \hspace{1cm} (3)
where \( \rho \) is the fluid density, \( c_p \) is the specific heat, \( u \) is the horizontal velocity component, \( T \) is the temperature and \( \dot{m} \) is the mass flow rate. Equation (3) is derived from the rate of flow of enthalpy through a given cross section divided by the rate of heat flow through the same cross section. For the cases shown in this paper, the material properties may be considered constant, and Equation (3) can be simplified to the following form:

\[
T_b = \frac{\sum_{i=1}^{n} (u_i b_i T_i)}{U_{av} b}
\]

where \( u_i \) is the velocity of in the centre of a control volume (CV), \( b_i \) is the height of the CV, \( T_i \) the temperature in the CV, \( U_{av} \) is the velocity averaged over the height and \( b \) is the height of the domain.

It can be shown that the energy balance through any given cross-section with a thickness \( dx \) can be derived to be, (Lienhard and Lienhard 2006):

\[
q_w Pdx = \dot{m} c_p dT_b
\]

which can be rearranged as:

\[
\frac{dT_b}{dx} = \frac{q_w P}{\dot{m} c_p}
\]

where \( q_w \) is the heat flux at the wall, \( P \) is the heated perimeter, \( \dot{m} \) is the mass flow rate, and \( c_p \) is the specific heat. Using the conditions specified in Figure 1 along with the constant wall heat flux boundary condition, the right hand side of Equation (5) becomes a constant value.

\[
\frac{dT_b}{dx} = \frac{q_w P}{\dot{m} c_p} = \frac{q_w 2d}{\rho (b \delta) U_{av} c_p} = \frac{(10)(2)}{(1.225)(0.05)(0.1)(1006.43)} = 3.2444 \text{ K/m}
\]

where \( d \) is the depth of the plates and \( U_{av} \) is the average velocity (equal to 0.1 m/s for the cases studied). Integrating both sides of Equation (6) with respect to \( x \) results in

\[
T_b = 3.2444x + C
\]

By imposing the boundary condition that at \( x=0 \)m the bulk temperature is equal to the inlet temperature \( (T_b = T_w = 283\text{K}) \), Equation (7) becomes

\[
T_b = 3.2444x + 283
\]
Equation (8) will be used to verify that bulk temperatures calculated from Fluent data are consistent with the analytical equations.

2.3. Heat transfer coefficient

The heat transfer coefficient may be obtained from analytically derived values of the Nusselt number, which should be constant for thermally developed flow between parallel plates. The values will differ slightly based upon the heating conditions as follows (Lienhard and Lienhard 2006):

\[ \frac{N_{u_{Dh}}}{k} = \begin{cases} 
7.541 & \text{for fixed plate temperatures} \\
8.235 & \text{for fixed wall heat fluxes}
\end{cases} \quad (9) \]

where \( D_{h} \) is the hydraulic diameter (typically twice the distance between parallel plates) and \( k \) is the thermal conductivity of air. The appropriate parameters may then be input to yield the following analytical values for \( h_c \):

\[ h_c = \frac{N_{u_{Dh}}k}{D_{h}} = \begin{cases} 
1.825 & \text{for fixed plate temperatures} \\
1.993 & \text{for fixed wall heat fluxes}
\end{cases} \quad W/m^2K \quad (10) \]

3. CFD simulations

3.1. Geometry and boundary conditions

The geometry shown in Figure 1 was reproduced with a mesh that was generated from a preliminary mesh sensitivity analysis. Mesh refinement was applied exponentially towards the wall surfaces. A uniformly spaced mesh was used in the streamwise direction. The initial mesh used for the Constant Heat Flux (CHF) and Constant Wall Temperature (CWT) cases had a total of 19,800 cells (33 in the vertical direction, 600 in the horizontal). A portion of the initial mesh is shown in Figure 2 below.

![Figure 2. Initial mesh used for the CFD simulations.](image)
The boundary conditions for the simulations were input as shown in Figures 3 and 4. The Fluent solution parameters and model information are provided in Appendix A.

The velocity profile used as the inlet condition is a parabolic profile commonly used to describe the flow between parallel plates [1]. When comparing the inlet and outlet profiles from the simulation results, the difference in velocity at a given height is on the order of $10^{-4}$ m/s. Therefore it can be assumed that the flow is indeed already fully developed at the inlet.

The flow field was initialized to the inlet conditions (described in Figure 4). The simulations were iterated until a scaled residual of $10^{-7}$ (Fluent Inc. 2003) was achieved for all the solution parameters involved.

### 3.2. Simulation results

Once the simulations were completed, the bulk temperatures were calculated with Equation (4) using the cell temperature and velocity data. For the case of constant heat flux, the simulation data can be compared to the analytical equation derived in Equation (8). The results from Fluent are plotted in Figure 5, and the resulting trendline equation is very close to the expected equation.

\[
\text{Analytical Bulk Temperatures:} \quad T_b = 3.2444x + 283
\]
Fluent Bulk Temperatures: \[ T_b = 3.2423x + 283 \]

The error increases slightly along the length of the plate. After 3m the difference between analytical and Fluent bulk temperatures is on the order of $10^{-3}$ %.

![Graph showing bulk temperature](image)

**Figure 5. Bulk temperature calculated from Fluent output data**

The convective heat transfer coefficients were calculated with Equation (2), using the three different fluid reference temperatures previously mentioned. The parameters used to solve Equation (2) are outlined in Table 2 below.

The results for the convective heat transfer coefficient indicate that the temperature value used to describe the fluid ($T_f$ from Equation 2) can have a significant effect on the result. The chosen reference temperature must match the one used in the derivation of the equation or correlation used for comparison. The reported values in Fluent are calculated based on a user specified constant reference value, which results in non-constant convective coefficients after thermally developed flow (Fluent Inc. 2003). Correlations that were developed using any other fluid temperature as a reference will not match the results from Fluent. Therefore, care must be taken on which values are used when reporting information from Fluent.

The convective coefficients calculated from the centerline temperatures are more realistic and follow the expected trend, but they under-predict the $h_c$ values by about 20% for the CHF solution and by about 24% for the CWT solution.

The bulk temperature yielded the best solution for the convective heat transfer coefficient, resulting in an error margin of less than 0.5% for both cases (after thermal development). Since the bulk temperature calculation is dependent on the
grid used, a grid sensitivity and discretization error analysis was performed to determine what the grid independent solution would be.

| Table 2. Convective heat transfer coefficient solution parameters |
|-----------------|-----------------|-----------------|
|                 | CHF – Case (a)  | CWT – Case (b)  |
| $q_w(x)$        | $q_w = 10 \text{ W/m}^2$ | $q_w(x) \rightarrow \text{From Fluent}$ |
| $T_w(x)$        | $T_w(x) \rightarrow \text{From Fluent}$ | $T_w = 293 \text{ K}$ |

$h_{cc} = \frac{q_w(x)}{T_w(x) - T_f(x)}$

$h_{cRef}(x) = \frac{10}{T_w(x) - 283}$  

$h_{cRef}(x) = \frac{q_w(x)}{293 - 283}$

$T_f(x) = T_{ref} = 283 \text{ K}$  

(Constant value specified in Fluent (Fluent Inc. 2003))

$h_{cc}(x) = \frac{10}{T_w(x) - T_c(x)}$  

$h_{cc}(x) = \frac{q_w(x)}{293 - T_c(x)}$

$T_f(x) = T_c(x)$  

(Horizontal temperature profile at the center of the flow ($y = 0$))

$h_{cb}(x) = \frac{10}{T_w(x) - T_b(x)}$  

$h_{cb}(x) = \frac{q_w(x)}{293 - T_b(x)}$

$T_f(x) = T_b(x)$  

(Bulk Temperature calculated at different $x$ positions from the Fluent Data)
Figure 6. Convective heat transfer coefficients for constant wall heat flux

Figure 7. Convective heat transfer coefficients for constant wall temperature
3.3. Grid Sensitivity Analysis

For the purposes of the grid sensitivity analysis, the convective heat transfer coefficients calculated are compared for different grid densities at \( x = 2.5 \) m. The process was repeated for both the CHF and CWT cases to compare the grid dependency for the two different boundary conditions. Only the coefficients calculated from the bulk temperature are part of this comparison.

The initial grid used for the simulations had a total of 19,800 cells. It was decided to proceed with several coarser grids and one finer mesh. The details of the different meshes are presented in Table 3. The notation \( \phi_h \) is adopted to describe the solution for the finest mesh. The subsequent meshes are all notated with respect to the finest mesh. The next grid size has cell dimensions doubled in both directions, hence the notation \( \phi_{2h} \).

Table 3. Mesh dimensions

<table>
<thead>
<tr>
<th>( \phi_h ) (80400)</th>
<th>( \phi_{2h} ) (19800)*</th>
<th>( \phi_{4h} ) (5100)</th>
<th>( \phi_{8h} ) (1200)</th>
<th>( \phi_{16h} ) (300)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of cells in the Y Direction</td>
<td>67</td>
<td>33</td>
<td>17</td>
<td>8</td>
</tr>
<tr>
<td>Number of cells in the X Direction</td>
<td>1200</td>
<td>600</td>
<td>300</td>
<td>150</td>
</tr>
<tr>
<td>Smallest cell height (m)</td>
<td>4.202E-04</td>
<td>8.749E-04</td>
<td>1.775E-03</td>
<td>3.948E-03</td>
</tr>
<tr>
<td>Smallest cell width (m)</td>
<td>0.0025</td>
<td>0.005</td>
<td>0.01</td>
<td>0.02</td>
</tr>
<tr>
<td>Total number of cells</td>
<td>80400</td>
<td>19800</td>
<td>5100</td>
<td>1200</td>
</tr>
</tbody>
</table>

* Original mesh

It can be shown (Ferziger and Peric 1997) that the discretization error of a grid is approximately

\[
\varepsilon_h \approx \frac{\phi_h - \phi_{2h}}{2^a - 1}
\]  

(11)

where \( a \) is the order of the scheme and is given by

\[
a = \frac{\log \left( \frac{\phi_{2h} - \phi_{4h}}{\phi_h - \phi_{2h}} \right)}{\log(2)}
\]  

(12)
In both equations the “2” refers to the increase in dimensions of the mesh. From Equation (12), it follows that a minimum of three meshes are required to determine the discretization error. In order to prevent a calculation error from the logarithm of a negative number, the three solutions must be monotonically converging [2].

The theory of Richardson Extrapolation states that the solution from the finest mesh can be added to the discretization error found in Equation (11) to attain an approximate grid independent solution. In equation form this can be stated as:

\[ \Phi = \phi_h + \varepsilon_h^d \]  

(13)

The results from the grid sensitivity analysis are shown in Table 4 and plotted below in Figures 8 and 9.

### Table 4. Discretization error and Richardson Extrapolation Results

<table>
<thead>
<tr>
<th>Order of the scheme</th>
<th>Discretization Error (\varepsilon_h^d) (W/m(^2)K)</th>
<th>Finest mesh solution (\phi_h) (W/m(^2)K)</th>
<th>Richardson Solution (\Phi) (W/m(^2)K)</th>
<th>Analytical solution (h_c) (W/m(^2)K)</th>
</tr>
</thead>
<tbody>
<tr>
<td>CHF</td>
<td>1.460</td>
<td>2.297x10(^{-3})</td>
<td>1.990578</td>
<td>1.992875</td>
</tr>
<tr>
<td>CWT</td>
<td>1.858</td>
<td>1.001x10(^{-3})</td>
<td>1.824089</td>
<td>1.825090</td>
</tr>
</tbody>
</table>

![Graph](image_url)

**Figure 8.** Grid convergence of the heat transfer coefficient for constant heat flux and relative error compared with Richardson solution
Figure 9. Grid convergence of the heat transfer coefficient for constant wall temperature and relative error compared with Richardson solution

Note that the actual order of the scheme \((a)\) is higher than the discretization scheme used in the Fluent solver (shown in Appendix A).

4. Conclusions

A validation exercise was performed by comparing the computed convective heat transfer coefficients \((h_c)\) for laminar air flow between parallel plates by Computational Fluid Dynamics to analytical solutions. The CFD simulations were performed for constant wall temperature and constant heat flux conditions. The importance of a correct reference temperature was confirmed.

The CFD results showed a good agreement with the analytical solutions, indicating a proper performance of the CFD code, at least for the cases studied.

Finally, a grid sensitivity analysis was performed on the mesh for both wall boundary conditions. The discretization error for \(h_c\) was calculated at a given location on the plate and Richardson extrapolation was used to compute the grid independent solution. The resulting \(h_c\) values had good agreement with analytical values from literature. The percentage error between the analytical and the grid independent solutions for \(h_c\) is on the order of \(10^{-2}\) %.
## Appendix A: Fluent solution parameters

<table>
<thead>
<tr>
<th>Model</th>
<th>Settings</th>
</tr>
</thead>
<tbody>
<tr>
<td>Space</td>
<td>2D</td>
</tr>
<tr>
<td>Time</td>
<td>Steady</td>
</tr>
<tr>
<td>Viscous</td>
<td>Laminar</td>
</tr>
<tr>
<td>Heat Transfer</td>
<td>Enabled</td>
</tr>
<tr>
<td>Solidification and Melting</td>
<td>Disabled</td>
</tr>
<tr>
<td>Radiation</td>
<td>None</td>
</tr>
<tr>
<td>Species Transport</td>
<td>Disabled</td>
</tr>
<tr>
<td>Coupled Dispersed Phase</td>
<td>Disabled</td>
</tr>
<tr>
<td>Pollutants</td>
<td>Disabled</td>
</tr>
<tr>
<td>Soot</td>
<td>Disabled</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Equation</th>
<th>Solved</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Flow</td>
<td>yes</td>
<td></td>
</tr>
<tr>
<td>Energy</td>
<td>yes</td>
<td></td>
</tr>
</tbody>
</table>

| Numerics | Enabled | Absolute Velocity Formulation | yes |

### Relaxation:

<table>
<thead>
<tr>
<th>Variable</th>
<th>Relaxation Factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pressure</td>
<td>0.3</td>
</tr>
<tr>
<td>Density</td>
<td>1</td>
</tr>
<tr>
<td>Body Forces</td>
<td>1</td>
</tr>
<tr>
<td>Momentum</td>
<td>0.7</td>
</tr>
<tr>
<td>Energy</td>
<td>1</td>
</tr>
</tbody>
</table>

### Solver:

<table>
<thead>
<tr>
<th>Variable</th>
<th>Termination Type</th>
<th>Residual Criterion</th>
<th>Reduction Tolerance</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pressure</td>
<td>V-Cycle</td>
<td>0.1</td>
<td>0.7</td>
</tr>
<tr>
<td>X-Momentum</td>
<td>Flexible</td>
<td>0.1</td>
<td>0.7</td>
</tr>
<tr>
<td>Y-Momentum</td>
<td>Flexible</td>
<td>0.1</td>
<td>0.7</td>
</tr>
<tr>
<td>Energy</td>
<td>Flexible</td>
<td>0.1</td>
<td>0.7</td>
</tr>
</tbody>
</table>

### Discretization Scheme

<table>
<thead>
<tr>
<th>Variable</th>
<th>Scheme</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pressure</td>
<td>Standard</td>
</tr>
<tr>
<td>Pressure-Velocity Coupling</td>
<td>SIMPLE</td>
</tr>
<tr>
<td>Momentum</td>
<td>First Order Upwind</td>
</tr>
<tr>
<td>Energy</td>
<td>First Order Upwind</td>
</tr>
</tbody>
</table>

### Solution Limits

<table>
<thead>
<tr>
<th>Quantity</th>
<th>Limit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Minimum Absolute Pressure</td>
<td>1</td>
</tr>
<tr>
<td>Maximum Absolute Pressure</td>
<td>5000000</td>
</tr>
<tr>
<td>Minimum Temperature</td>
<td>1</td>
</tr>
<tr>
<td>Maximum Temperature</td>
<td>5000</td>
</tr>
</tbody>
</table>
Appendix B: Nomenclature

a Order of the discretization error scheme (-)
b Distance between parallel plates (m)
cp Specific heat (J/kgK)
d Depth of the parallel plates (m)
Dh Hydraulic diameter (m)
hc Convective heat transfer coefficient (W/m²K)
k Thermal conductivity (W/m-K)
L Length of domain (m)
m Mass flow rate (kg/s)
\( \dot{m} \) Mass flow rate (kg/s)
\( N_U_{Dh} \) Nusselt number calculated with the hydraulic diameter (-)
P Heated perimeter of the domain (m)
q Heat flux (W/m²)
u Velocity component in the x-direction (m/s)
v Velocity component in the y-direction (m/s)
T Temperature (K)
U Velocity magnitude (m/s)

Greek symbols
\( \varepsilon_h \) Discretization error (units based on parameter analyzed)
\( \phi_h \) Solution for finest mesh (units based on parameter analyzed)
\( \phi_{n\times h} \) Solution for a mesh with cell dimensions \( n \) times the finest mesh
\( \Phi \) Richardson Extrapolation solution (units based on parameter analyzed)
\( \mu \) Dynamic viscosity (kg/m-s)
\( \rho \) Density (kg/m³)

Subscripts
AV Average property
b Bulk property
c Property taken at the centerline of the domain
f Fluid property
i Property of an element i
ref Reference property
x Property taken at a location x
∞ Free stream property
References