Numerical Simulation of Heat Transfer by Natural Convection in Horizontal Finned Channels

Gabriel Gonçalves da Silva Ferreira, Luiz Fernando Lopes Rodrigues Silva
Escola de Química, UFRJ

Paulo L. C. Lage (*), Fábio Pereira dos Santos
Laboratório de Termofluidodinâmica,
Programa de Engenharia Química, COPPE/UFRJ

(*) paulo@peq.coppe.ufrj.br
1) Company Overview

LTFD research lines:
- Theoretical modeling and experimental measurements in multiphase flows
- CFD simulation of multiphase flows and applied CFD (ANSYS CFX, FLUENT, OpenFOAM)
- Population balance
- Numerical Methods
- Parallel Computing

Infrastructure:
Clusters: 128 core Opteron 2356 based cluster (almost 1 TeraFLOPs) and a beowulf cluster based on Intel Q6600
2) Problem Description

2.1) Mixed Convection

Heat transfer mechanisms: conduction, convection, radiation

Two limiting types in convection:
- Forced Convection: flow patterns are determined by external forces.
- Free Convection (Natural Convection): flow patterns are determined by buoyant forces in the heated fluid.

In general:
Mixed convection – both effects must be taken account.
2) Problem Description

2.2) Fins

Maximization of heat transfer rate.

Under Natural/Mixed convection regimes in channel flows:

- $h_{conv}$ is low.
- $h_{conv}$ is easily modified with changes in fin geometry.

Optimum geometry design:

- Experimental investigation.
- CFD simulations (need to validate the models!)

2.3) Objective

Develop and validate a model for natural and mixed convection in a horizontal finned channel.
3) Methodology

Comparison of experimental results obtained by Dogan & Sivrioglu\(^1\) with CFD results.

Experiments measured the effect of size and spacing of fins in \(h_{\text{conv}}\) for different \(T_{\text{base}}\) values.

Software ANSYS 12.1 for CFD simulations:
Design Modeler for geometry.
ICEM CFD for hexahedral meshes.

\(^1\) M. Dogan, M. Sivrioglu, Experimental investigation of mixed convection heat transfer from longitudinal fins in a horizontal rectangular channel: In natural convection dominated flow regimes, Energy Conversion and Management 50 (2009) 2513-2521.
3) Methodology

3.1) Experimental setup
3) Methodology

3.2) Experimental results

Fig. 6. Variation of average convection heat transfer coefficient with fin height (fin spacing, $S/H = 0.18$).
3) Methodology

3.3) Experimental analysis

Experimental conditions
Longitudinal aluminum fins with rectangular section
Air velocity at the channel inlet: \( v = 0.02 - 0.025 \text{ m/s} \) (Re = 250) at ambient conditions, but controlled by valve near the outlet

<table>
<thead>
<tr>
<th>Fin-array set</th>
<th>Fin height ( (H_f/H) )</th>
<th>Fin spacing ( (S/H) )</th>
<th>Fins number ( (N) )</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.25</td>
<td>0.04</td>
<td>59</td>
</tr>
<tr>
<td>2</td>
<td>0.25</td>
<td>0.08</td>
<td>32</td>
</tr>
<tr>
<td>3</td>
<td>0.25</td>
<td>0.12</td>
<td>22</td>
</tr>
<tr>
<td>4</td>
<td>0.25</td>
<td>0.18</td>
<td>15</td>
</tr>
<tr>
<td>5</td>
<td>0.50</td>
<td>0.04</td>
<td>59</td>
</tr>
<tr>
<td>6</td>
<td>0.50</td>
<td>0.08</td>
<td>32</td>
</tr>
<tr>
<td>7</td>
<td>0.50</td>
<td>0.12</td>
<td>22</td>
</tr>
<tr>
<td>8</td>
<td>0.50</td>
<td>0.18</td>
<td>15</td>
</tr>
<tr>
<td>9</td>
<td>0.80</td>
<td>0.04</td>
<td>59</td>
</tr>
<tr>
<td>10</td>
<td>0.80</td>
<td>0.08</td>
<td>32</td>
</tr>
<tr>
<td>11</td>
<td>0.80</td>
<td>0.12</td>
<td>22</td>
</tr>
<tr>
<td>12</td>
<td>0.80</td>
<td>0.18</td>
<td>15</td>
</tr>
</tbody>
</table>
3) Methodology

3.4) Geometrical Modelling of Fin-array set 8

Geometry 2 (entire channel):
3) Methodology

3.4) Geometrical Modelling of Fin-array set 8

Isometric view of geometry 1
3) Methodology

3.5) Numerical Model

Transient 3D analysis
Equations:
  Continuity
  Momentum
  Thermal energy
  Turbulence Model (SST)
Equation of State (Ideal Gas)
Empirical Correlations for $k$, $C_p$ and $\mu$ for air at atmospheric pressure based on experimental data on the range: $250 \text{ K} < T < 400 \text{ K}$
3) Methodology

3.5) Numerical Model – boundary conditions

- Fins efficiency > 99% (estimative based on the experimental results): constant $T_{\text{base}}$ for fin surfaces

- Boundary conditions:

  Heated and fin surfaces: No-slip, $T$ specified (53°C)
  Other walls: No-slip, adiabatic.
### 3) Methodology

#### 3.5) Numerical Model – boundary conditions

Inlet and outlet boundary conditions for cases 1 and 2:

<table>
<thead>
<tr>
<th>Case</th>
<th>Geometry</th>
<th>B.C. inlet</th>
<th>B.C. outlet</th>
</tr>
</thead>
</table>
| 1    | 1        | Opening - entrainment  
T = 25 °C  
Relative Pressure = 0 Pa | Opening - entrainment  
T = mean temperature of exiting fluid  
Relative Pressure = 0 Pa |
| 2    | 2        | Opening – entrainment  
T = 25 °C  
Relative Pressure = 0 Pa | Mass flow rate corresponding to  
v = 0,025 m/s |
4) Meshes for geometric modeling

Three separated meshes to test geometric modeling:

Mesh 1 – rectangular channel
- 836,224 nodes
- 775,986 elements (100% hexaedra)

Mesh 2 – nozzle
- 44,664 nodes
- 40,500 elements (100% hexaedra)

Mesh 3 – chamber+valve
- 264,516 nodes
- 253,028 elements (100% hexaedra)
4) Meshes for geometric modeling

Side view of mesh 1:
4) Meshes for geometric modeling

Isometric view of mesh 2:
4) Meshes for geometric modeling

Isometric view of mesh 3:
5) Preliminary Results

Heat transfer rate for cases 1 and 2:

<table>
<thead>
<tr>
<th>Case</th>
<th>B.C. inlet</th>
<th>B.C. Outlet</th>
<th>Q [W]</th>
</tr>
</thead>
</table>
| 1     | Opening - entrainment  
T = 25 °C  
Relative Pressure = 0 Pa | Opening - entrainment  
T = mean temperature of exiting fluid  
Relative Pressure = 0 Pa | 20,6   |
| 2     | Opening – entrainment  
T = 25 °C  
Relative Pressure = 0 Pa | Mass flow rate corresponding to  
v = 0,025 m/s | 45,4   |
|       |                                                |                                                | 37,3   |
5) Preliminary Results

Comparison between cases 1 and 2 vector plots at the channel outlet:
5) Preliminary Results

Case 2 streamlines: Fresh air reaches only first half of the finned section!
6) Conclusions and next steps

The larger error of case 1 (44.8%) compared to case 2 (21.6%) and the behavior of the flow at the outlet show that the geometric modeling used in case 1 is not a good approximation. Some geometries can’t be simplified for natural convection.

The experimental mass flow rate is given by the article as mean velocity at the channel inlet but with a large error margin (0.02-0.025 m/s with and error of 0.015 m/s). Present simulations were made with the mass flow rate corresponding to the higher velocity value. Smaller values could give results closer to the experimental one.

Next steps:
- Use mass flow rate corresponding to the lower velocity case.
- Run case 2 with a more refined mesh.