

## Documents

Menni, Y., Chamkha, A., Zidani, C., Benyoucef, B.

**Baffle orientation and geometry effects on turbulent heat transfer of a constant property incompressible fluid flow inside a rectangular channel**  
(2019) *International Journal of Numerical Methods for Heat and Fluid Flow*, . Cited 1 time.

**Abstract**

Purpose: A computational fluid dynamics (CFD) analysis has been carried out on the aerodynamic and thermal behavior of an incompressible Newtonian fluid having a constant property and flowing turbulently through a two-dimensional horizontal high-performance heat transfer channel with a rectangular cross section. The top surface of the channel was kept at a constant temperature, while it was made sure to maintain the adiabatic condition of the bottom surface. Two obstacles, with different shapes, i.e. flat rectangular and V-shaped, were inserted into the channel; they were fixed to the top and bottom surfaces of the channel in a periodically staggered manner to force vortices to improve the mixing and consequently the heat transfer. The first fin-type obstacle is placed on the heated top channel surface, and the second baffle-type one is placed on the insulated bottom surface. Five different obstacle situations were considered in this study, which are referred as cases FF (flat fin and flat baffle), FVD (flat fin and V-downstream baffle), FVU (flat fin and V-upstream baffle), VVD (V-downstream fin and V-downstream baffle) and VVU (V-Upstream fin and V-upstream baffle). Design/methodology/approach: The flow model is governed by Reynolds-averaged Navier–Stokes equations with the k-epsilon turbulence model and the energy equation. These governing equations are discretized by the finite volume method, in two dimensions, using the commercial CFD software FLUENT software with the Semi Implicit Method for Pressure Linked Equations (SIMPLE) algorithm for handling the pressure-velocity coupling. Air is the test fluid with the flow rate in terms of Reynolds numbers ranging from 12,000 to 32,000. Findings: Important deformations and large recirculation regions were observed in the flow field. A vortex causes a rotary motion inside the flow field, which enhances the mixing by bringing the packets of fluid from the near-wall region of the channel to the bulk and the other way around. The largest value of the axial variations of the Nusselt number and skin friction coefficient is found in the region facing the baffle, while the smallest value is in the region near the fin, for all cases. The thermal enhancement factor (TEF) was also introduced and discussed to assess the performance of the channel for various obstacle situations. It is found that the TEF values are 1.273-1.368, 1.377-1.573, 1.444-1.833, 1.398-1.565 and 1.348-1.592 for FF, FVD, FVU, VVD and VVU respectively, depending on the Re values. In all cases, the TEF was found to be much larger than unity; its maximum value was around 1.833 for FVU at the highest Reynolds number. Therefore, the FVU may be considered as the best geometrical configuration when using the obstacles to improve the heat transfer efficiency inside the channel. Originality/value: This study can be a real application in the field of shell-and-tube heat exchangers and flat plate solar air collectors. © 2019, Emerald Publishing Limited.

2-s2.0-85067011495

**Document Type:** Article

**Publication Stage:** Article in Press

**Source:** Scopus