Abstract :

The turbulent heat transfer by a confined jet flowing inside a hot cylindrical cavity is investigated numerically in this paper. This configuration is found in several engineering applications such as air conditioning and the ventilation of mines, deadlock, or corridors. The parameters investigated in this work are the Reynolds number (Re, 20,000 ≤ Re ≤ 50,000) and the normalized distance L_f between jet exit and the cavity bottom (L_f, $2 \leq L_f \leq 12$). The numerical predictions are performed by finite volume method using the second order one-point closure turbulence model (RSM). The Nusselt number increases and attains maximum values at stagnation points, after it decreases. For an experimental test case available in the literature $L_f = 8$, the numerical predictions are in good agreement. Processes of heat transfer are analyzed from the flow behavior and the underlying mechanisms. The maximum local heat transfer between the cavity walls and the flow occurs at $L_f = 6$ corresponding to the length of the potential core. Nusselt number at the stagnation point is correlated versus Reynolds number Re and impinging distance L_f; [$Nu_0 = f(Re, L_f)$Nu0=f(Re,Lf)].