Fouling of Waste Heat Recovery: Numerical and Experimental Results

E. Sauret\(^1\), I. Abdi\(^2\) and K. Hooman\(^2\)

\(^1\)School of Chemistry, Physics & Mechanical Engineering
Queensland University of Technology, Brisbane, Qld 4000, Australia
\(^2\)School of Mechanical & Mining Engineering
The University of Queensland, St Lucia, Qld 4072, Australia

Abstract

Numerical results are presented to investigate the performance of a partly-filled porous heat exchanger for waste heat recovery units. A parametric study was conducted to investigate the effects of inlet velocity and porous block height on the pressure drop of the heat exchanger. The focus of this work is on modelling the interface of a porous and non-porous region. As such, numerical simulation of the problem is conducted along with hot-wire measurements to better understand the physics of the problem. Results from the two sources are then compared to existing theoretical predictions available in the literature which are unable to predict the existence of two separation regions before and after the porous block. More interestingly, a non-uniform interface velocity was observed along the streamwise direction based on both numerical and experimental data.

Introduction

Porous heat exchangers are receiving considerable attention as their application can lead to high heat transfer rates usually associated with a limited footprint. This can be of significant importance in engineering applications such as air-cooled condensers wherein heat exchanger size determines the fan or the cooling tower size. Like other surface extension approaches, however, this heat transfer augmentation technique causes extra pressure drop. Thus it is crucial to minimize the total pressure drop while keeping the augmented heat transfer rate. In order to improve the heat transfer thanks to a surface area increase, fins are added to the tube-bundle heat exchangers. New developments led to further improve the efficiency of heat exchangers by replacing the fins by metal foams [4]. Recently, porous heat exchangers, like metal foams, were suggested as alternatives to fins [5, 6, 8, 14]. It appears that despite the efforts made in the literature, such porous heat exchangers are not well understood and therefore not yet optimized for engineering applications such as heat exchangers. Porous heat exchangers are currently designed using the same knowledge gathered for fins over the years. This, however, is not the best analogy. Recent experimental results, for instance, showed that the wake behind a porous-covered pipe is completely different from those of bare and finned tubes in cross flow [12] and the flow structures are detaching from the wake [1]. This is to be expected as fins act like narrow channels to guide the gas flow in the preferred direction(s). While similar to fins in leading to boundary layer interruption, porous covers lead to a random flow distribution within the pores with different local heat transfer patterns and wall heat flux split [11]. In addition, with any partial blockage of the flow area, another unknown is the problem of the interface modeling between a porous and non-porous region. As recently underlined by Nield and Kuznetsov [15], this interface modelling remains an open question in the literature. While physically one expects much lower fluid velocity in the pores compared to that of free flow, capturing this sharp gradient at the interface can add to the difficulties of numerical simulation. Experiments addressing this issue are, surprisingly, rare. Beavers and Joseph [3] were amongst the first to show that sharp gradients at the interface between the porous and fluid regions exist. Their work highlighted the existence of a slip velocity at the interface. From there, authors have established different interface conditions that can be classified into two main types according to Alazmi and Vafai [2]: slip and no-slip boundary conditions. Those authors then establish five main categories for the hydrodynamic interface conditions and four categories for the thermal interface conditions that they critically examined. The different models mostly lead to comparable results except for few specific cases. To show the complexity of the problem, it is interesting to note that all these works were conducted for duct flows where there is no recirculation or wakes which cannot be modeled as internal flows. This paper does not aim at solving the interface problem but it presents a critical comparison of the theoretical analysis and our numerical simulations and experimental data.

Experiments

Set Up

Using experimental setup shown in figures 1 and 2, it is possible to observe flow past a block of metal foam at different Reynolds numbers. The experimental setup consists of an open loop suction wind tunnel. Air is drawn into the intake bell- mount by a fan rotor driven by a 17 kW electric motor. The intake consists of a fine mesh screen that is used as a filter to prevent unwanted particles, followed by a honeycomb section containing 1700 cardboard cylinders. Removable flow-smoothing screens are located immediately downstream of these cylinders [10]. The test section of the wind tunnel is a square one. The size of the test section is 300x300x2000 mm3 located in the School of Mechanical and Mining Engineering at the University of Queensland. The test section walls have been constructed out of transparent Plexiglas that allows photography of flow field. The air velocity at the test section inlet is measured by means of a 55P05 Dantec hotwire probe. In figure 1 the streamwise and the transverse directions are indicated by x and y axes, respectively.

Figure 1. Wind Tunnel Schematic.
All the measurements are made at three different inlet velocities (U) of 3, 6 and 12 m/s. The free stream turbulence level of empty test section was measured as 0.5% at the lowest velocity.

**Hotwire Anemometry**

In hot-wire measurements, a Dantec 53P15 single sensor hotwire probe was used. The probe has 1.25 mm long platinum-plated tungsten wire sensing elements of 5µm diameter and is operated in constant temperature mode with an over-heat ratio set to 1.8. The probe was calibrated in the free stream using Dantec 54T29 reference velocity probe. The probe was mounted to a computer controlled three-axis traverse system. Streamwise velocity fluctuations were acquired at linearly spaced cross-section in X downstream of the cylinder with a resolution of 10µm, with sufficient sampling frequency of 10 kHz to resolve the smallest scales and sufficiently long sample lengths for statistical convergence (120 seconds at each point). The uncertainty relative to the maximum velocity at 95% confidence is 1.3%.

**Samples**

Three metal foam samples with different heights (10, 20 and 50mm) were used to conduct this experiment. The foams are made of an open cell aluminum foam sheet, bonded to a stainless steel plate with dimensions 320 x 270 mm (width x length). Foams consist of ligaments forming a network of interconnected cells. The cells are randomly oriented and are mostly homogeneous in size and shape. The foams PPI is 10 and the effective density varies from 3% to 5% of a solid of the same material.

**Numerical Modelling**

**Computational Domain**

The 2D computational domain is presented in figure 3 with a total length L=2m, a total height H=0.5m. The foam colored in blue is 0.27m long (L_f). Two foam heights have been experimentally and numerically investigated, $H_f=0.02$ and 0.05m. In the computational domain, the foam is placed in the middle of the domain so sufficient length before and after the foam are introduced to correctly model the flow.

**Results**

Comparisons CFD-Experiments

The comparison between the experiments and the numerical results is presented in figure 5 for the 20-mm foam height. The trends of the velocity along the interface are properly reproduced by the numerical model. At the inlet of the foam, the velocity is underestimated by the CFD as the inlet velocity increases. Near the end of the foam, the velocity continues to decrease for the 50-mm foam. Recirculation is even observed at $U_{in}=12m.s^{-1}$.

The non-dimensional velocity profiles at different locations along the foam are plotted in figure 7. The numerical simu-
The phenomenon, nonetheless, is not observed nor accounted for in the theoretical models developed by Beavers and Joseph [3] and later applied by some authors including Kuznetsov [13] and de Lemos [18]. As a result of a constant and uniform interface velocity, Kuznetsov’s velocity profile remains the same in the porous block. However, comparing the inlet and outlet velocity profiles from our CFD simulation, one notes a difference in the profile shapes. This asks for a more detailed simulation and experiment looking into the pore velocity distribution close to the interface. We leave this, however, for a future report.

**Conclusions**

Hot wire measurements are conducted along with numerical simulation of flow through a partly-porous heat exchanger for waste heat recovery applications. The main goal is to improve...
our understanding of the interface of a porous and non-porous region. While numerical results are showing closer agreement to experimental data, theoretical results, obtained based on a velocity jump boundary condition, are found to be inaccurate asking for more detailed analysis of the problem. Two separation regions before and after the porous block were observed. Furthermore, a non-uniform interface velocity was observed along the streamwise direction based on collected experimental data. None of these, separation and non-uniform interface velocity, could have been predicted using theoretical models. Further development of interface models using the data collected here is then left for a future study.

Acknowledgements

References


