World Academy of Science, Engineering and Technology
International Journal of Mechanical and Mechatronics Engineering
Vol:13, No:03, 2019

Numerical Simulation of the Heat Transfer Process in a Double Pipe Heat Exchanger

Authors : J. I. Corcoles, J. D. Moya-Rico, A. Molina, J. F. Belmonte, J. A. Almendros-Ibanez

Abstract : One of the most common heat exchangers technology in engineering processes is the use of double-pipe heat exchangers (DPHx), mainly in the food industry. To improve the heat transfer performance, several passive geometrical devices can be used, such as the wall corrugation of tubes, which increases the wet perimeter maintaining a constant cross-section area, increasing consequently the convective surface area. It contributes to enhance heat transfer in forced convection, promoting secondary recirculating flows. One of the most extended tools to analyse heat exchangers' efficiency is the use of computational fluid dynamic techniques (CFD), a complementary activity to the experimental studies as well as a previous step for the design of heat exchangers. In this study, a double pipe heat exchanger behaviour with two different inner tubes, smooth and spirally corrugated tube, have been analysed. Hence, experimental analysis and steady 3-D numerical simulations using the commercial code ANSYS Workbench v. 17.0 are carried out to analyse the influence of geometrical parameters for spirally corrugated tubes at turbulent flow. To validate the numerical results, an experimental setup has been used. To heat up or cool down the cold fluid as it passes through the heat exchanger, the installation includes heating and cooling loops served by an electric boiler with a heating capacity of 72 kW and a chiller, with a cooling capacity of 48 kW. Two tests have been carried out for the smooth tube and for the corrugated one. In all the tests, the hot fluid has a constant flowrate of 50 l/min and inlet temperature of 59.5°C. For the cold fluid, the flowrate range from 25 l/min (Test 1) and 30 l/min (Test 2) with an inlet temperature of 22.1°C. The heat exchanger is made of stainless steel, with an external diameter of 35 mm and wall thickness of 1.5 mm. Both inner tubes have an external diameter of 24 mm and 1 mm thickness of stainless steel with a length of 2.8 m. The corrugated tube has a corrugation height (H) of 1.1 mm and helical pitch (P) of 25 mm. It is characterized using three non-dimensional parameters, the ratio of the corrugation shape and the diameter (H/D), the helical pitch (P/D) and the severity index ($SI = H^2/P \times D$). The results showed good agreement between the numerical and the experimental results. Hence, the lowest differences were shown for the fluid temperatures. In all the analysed tests and for both analysed tubes, the temperature obtained numerically was slightly higher than the experimental results, with values ranged between 0.1% and 0.7%. Regarding the pressure drop, the maximum differences between the values obtained numerically, and the experimental values were close to 16%. Based on the experimental and the numerical results, for the corrugated tube, it can be highlighted that the temperature difference between the inlet and the outlet of the cold fluid is 42%, higher than the smooth tube.

Keywords : corrugated tube, heat exchanger, heat transfer, numerical simulation
Conference Title : ICEHTFMT 2019 : International Conference on Experimental Heat Transfer, Fluid Mechanics and Thermodynamics
Conference Location : Madrid, Spain
Conference Dates : March 26-27, 2019