Scopus

Documents

Al-Waked, R., Nasif, M.S., Morrison, G., Behnia, M.

CFD simulation of air to air enthalpy heat exchanger
(2013) *Energy Conversion and Management*, 74, pp. 377-385. Cited 25 times.

Abstract
A CFD model which supports conjugate heat and mass transfer problem representation across the membrane of air-to-air energy recovery heat exchangers has been developed. The model consists of one flow passage for the hot stream and another for the adjacent cold stream. Only half of each flow passage volume has been modelled on each side of the membrane surface. Three dimensional, steady state and laminar flow studies have been conducted using a commercial CFD package. The volumetric species transport model has been adopted to describe the H2O and air gas mixtures. Mesh dependency has been examined and followed by validation of the CFD model against published data. Furthermore, effects of flow direction at the inlet of the heat exchanger on its thermal effectiveness have been investigated. Simulation results are presented and analysed in terms of sensible effectiveness, latent effectiveness and pressure drop across the membrane heat exchanger. Results have shown that counter-flow configuration has greater sensitivity to the mesh centre perpendicular distance from the membrane when compared to the other two flow configurations (cross-/parallel-flow). However, the lateral mesh element length has shown minimal effect on the thermal effectiveness of the enthalpy heat exchanger. For the quasi-flow heat exchanger, a perpendicular flow direction to the inlets has been found to produce a higher performance in contrast to the non-perpendicular flow.

2-s2.0-84879467167
Document Type: Article
Publication Stage: Final
Source: Scopus