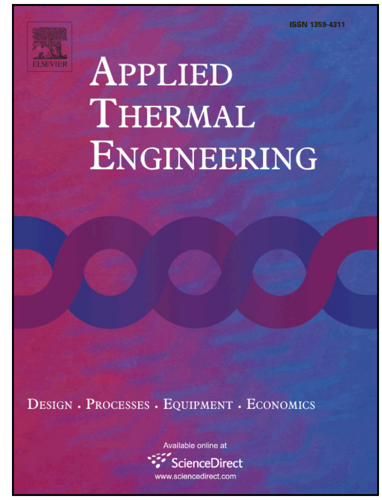
Accepted Manuscript



Computational Fluid-Dynamics modeling of supersonic ejectors: screening of turbulence modeling approaches

Giorgio Besagni, Fabio Inzoli

PII: S1359-4311(16)32829-0
DOI: http://dx.doi.org/10.1016/j.applthermaleng.2017.02.011
Reference: ATE 9891

To appear in: *Applied Thermal Engineering*

Received Date: 28 October 2016
Revised Date: 22 January 2017
Accepted Date: 4 February 2017

Please cite this article as: G. Besagni, F. Inzoli, Computational Fluid-Dynamics modeling of supersonic ejectors: screening of turbulence modeling approaches, *Applied Thermal Engineering* (2017), doi: http://dx.doi.org/10.1016/j.applthermaleng.2017.02.011

Computational Fluid-Dynamics modeling of supersonic ejectors: screening of turbulence modeling approaches

Giorgio Besagni*, Fabio Inzoli

Politecnico di Milano, Department of Energy, Via Lambruschini 4, 20156

*Corresponding author: Giorgio Besagni, +39-02-2399-3826; giorgio.besagni@polimi.it; Address: Politecnico di Milano, Department of Energy Via Lambruschini 4a, 21056 Milan, Italy.

ABSTRACT

Ejector refrigeration has not been able to penetrate the market because of the low coefficient of performance and because of the high influence of ejector performance on the efficiency of the refrigeration system. Improving the performance of ejector refrigeration systems relies on the understanding of the fluid dynamic phenomena, which can be obtained through computational fluid-dynamics approaches. Unfortunately, no agreement has been reached yet on the closures for the turbulence modeling in the Reynolds Averaged Navier-Stokes (RANS) approach. This paper contributes to the existing discussion by presenting a numerical study of the turbulent compressible fluid in a supersonic ejector. Seven RANS turbulence closures have been compared and, in addition, the turbulence models were tested under different near-wall modelling options in order to investigate the wall treatment effect on the numerical results. The numerical results have been validated by literature data consisting in entrainment ratio and wall static pressure measurements, for different ejector geometries and operating conditions. As a result, the k–omega model shows better performance in terms of global and local flow phenomena predictions for the different ejector geometries and operating conditions: (a) on the global point of view, the entrainment ratio is well predicted with a maximum relative error equal to about 10%, (b) on the local point of view, the shock wave position, the pressure recovery and the wall static pressure values are well predicted. The results, taking into account previous numerical studies, suggest the use of the k–omega model to simulate ejector fluid dynamics.

Keywords: *CFD; Ejector refrigeration; Computational Fluid Dynamics; Turbulence models; Modeling*