# CFD – Experiments Integration in the Evaluation of Six Turbulence Models for Supersonic Ejectors Modeling

*Y. Bartosiewicz*, P. Desevaux+, Zine Aidoun* and Yves Mercadier‡*

Supersonic ejectors are technological components that have found many applications in engineering. In the aerospace area, they can be used for altitude testing of a propulsion system by reducing the pressure of a test chamber [1]. Moreover, in the same area, the pumping effect is also used to mix the exhaust gases with fresh air in order to reduce the thermal signature [2].

Our primary interest in this paper is the application of supersonic ejectors as means of thermal compression in refrigeration cycles [3, 4]. Ejectors in this case may either totally replace the mechanical compressor or simply be introduced as a means of cycle optimization.

Numerical simulations of these systems are useful to improve their performance and to understand some complex phenomena, such as the condensation front involved during expansion or the shock waves appearance. However, some existing numerical works [5, 6, 7] have demonstrated the difficulty to correctly predict the flow structure and global operation (entrainment rate), even for a single-phase ejector.

The purpose of this study is therefore to correctly validate a numerical model capable of tracking the flow physics, more particularly the strong interaction between compressibility and turbulence, which is responsible for the pressure rise through a shock train region. In this respect, six turbulence models, namely k-epsilon, realizable k-epsilon, RNG-k-epsilon, RSM and two k-omega, with their respective published constants are tested with different corrections for compressible flows.

The results are compared with experimental data and flow visualizations (laser tomography) [7]. The most important parameter for evaluating the shock reflection pattern is generally the axial pressure distribution, which is measured here with a capillary tube crossing the whole flow domain. In particular, the effect of the pressure probe is evaluated through the numerical model, which allows choosing the turbulence model most appropriately. Indeed, the inclusion of the probe in the model shows that the RNG, the realizable k-epsilon and a k-omega model are superior in predicting the shock positions, while the results without the probe do not allow putting forward a more suitable model. This fact shows that one cannot rely on previous validations [8] for only under-expanded jets to choose the most appropriate turbulence model.

Flow visualizations are also used to check the non-mixing length and are compared with a numerical colorant (passive scalar). The results show very good agreements for the non-mixing length, the shock position and the global operation (induced and no induced flux) of the ejector. In addition, this study illustrates the effect of a probe inserted in the supersonic flow, even if it is likely to be small enough so as not to disturb the flow and measurements.

[1] Roshke, E. J., Massier, F. P., and Gier, H. L., Experimental Investigation of Exhaust Diffuser for Rocket Engines, Technical Report 32-210, Jet Propulsion Laboratory, Pasadena, 1962.

[2] Zhou, B, Fleck, B. A., Bouak, F., and Gauthier, J. E. D., Comparison of Turbulence Models for swirling Effects on Ejector Performances, 8th Annual Conference of the CFD Society of Canada, vol. 1, pp. 321-327, Montreal, Canada, 2000.

[3] M. Ouzzane, M. and Z. Aidoun, Model Development and Numerical Procedure for Detailed Ejector Analysis and Design, to be published in Applied Thermal Engineering.

[4] DA-WEN Sun, and Eames, I. W., Recent Development in the Design Theories and Applications of Ejector - a Review, Journal of the Institute of Energy, Vol. 68, pp. 65-79, 1995.

[5] Riffat, S. B., and Gan, G., Smith, S., Computational Fluid Dynamics Applied to Ejector Heat Pump, Applied Thermal Engineering, vol. 16, No. 4, pp. 291-297, 1996.

[6] Riffat, S. B., and Everitt, P., Experimental and CFD Modelling of an Ejector System for Vehicle Air Conditioning, Journal of the Institute of Energy, vol. 72, No. 6, pp. 41-47,1999.

[7] Desevaux, P., and Aeschbacher, O., Numerical and Experimental Flow Visualization of the Mixing Process Inside an Induced Air Ejector, International Journal of Turbo and jet Engines, vol. 19, pp. 71-78, 2002.

[8] Bartosiewicz, Y., Mercadier, Y., and Proulx, P., Numerical Investigations on the Dynamics and heat Transfer in a Turbulent Underexpanded Jet, AIAA Journal, vol. 40, No. 11, pp. 2257-2265, 2002.

[*] CANMET Energy Technology Centre, Varennes (Qc), Canada

[+] CNRS, IGE, Belfort, France

[‡] THERMAUS, Université de Sherbrooke, Sherbrooke (Qc), Canada