11th. World Congress on Computational Mechanics (WCCM XI)
5th. European Conference on Computational Mechanics (ECCM V)
6th. European Conference on Computational Fluid Dynamics (ECFD VI)
July 20 - 25, 2014, Barcelona, Spain

EVALUATION OF AN IMMERSED BOUNDARY METHOD FOR SOLVING THE FLUID STRUCTURE INTERACTION PROBLEM IN REFRIGERATION COMPRESSOR VALVES

José L. GASCHE[1*], Franco BARBI[2]

[1*] UNESP-Univ. Estadual Paulista, Department of Mechanical Engineering, Av. Brasil Centro 56, 15385-000, Ilha Solteira, SP, Brazil, gasche@dem.feis.unesp.br, www.dem.feis.unesp.br
[2] UFU-Federal University of Uberlândia, Department of Mechanical Engineering, Campus Santa Mônica – Bloco M1, 34800-902, Uberlândia-MG, Brazil, frcbarbi@gmail.com, www.mecanica.ufu.br/laboratorios/mflab

Key Words: *Immersed Boundary Method, Fluid-Structure Interaction, Compressor, Valve*

In refrigeration compressors, the suction and discharge valves (reed valves), as shown in Fig. 1, are responsible for the retention of the refrigerant from the suction chamber to the cylinder and passage from the cylinder to the discharge chamber.

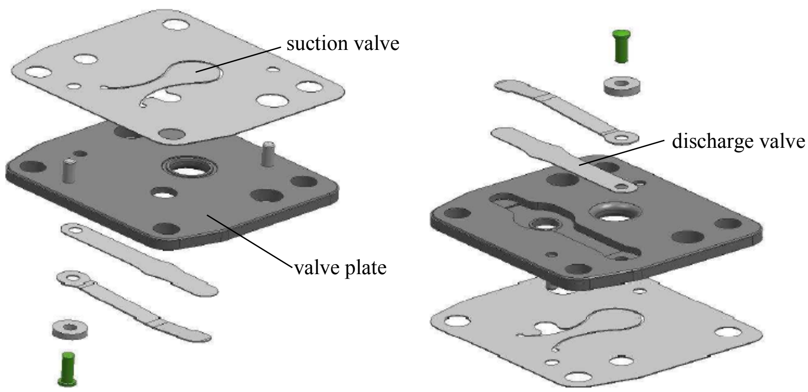

Figure 1 – Refrigeration compressor valves

As the opening and closing of the valves are caused by the forces produced by the refrigerant flow itself, the understanding of this fluid-structure interaction problem which characterizes the valve dynamics is of fundamental importance in order to enhance the efficiency of the valve system. The numerical simulation of the flow has been used as an efficient method to perform this task. Due to the complex geometry usually found in this type of valve, simplified geometries have been much used to represent the valve, particularly the radial diffuser. A very few researchers have solved this problem by considering real geometries for modeling the valves. This work presents a numerical procedure to simulate the unsteady flow through a more realistic model for the suction valve, including the movement of the reed. In order to evaluate the numerical procedure, the movement of the reed was given by imposing an angular velocity to the reed. An Immersed Boundary Method (IBM) with the Multi-Direct Forcing Scheme [1] was used to represent the valve geometry. An adaptive mesh dynamically refined was used for representing the flow domain. The governing equations were solved by a projection method, using a semi-implicit second-order scheme for time integration [2]. The systems of algebraic equations were solved by a Multigrid-Multilevel technique. Results for pressure (Fig. 2) and velocity fields and for pressure profiles on the reed surface were obtained for Reynolds number varying from 1,000 to 8,000. The results show that the IBM is a very good alternative for solving the flow through reed type valves with complex geometry.

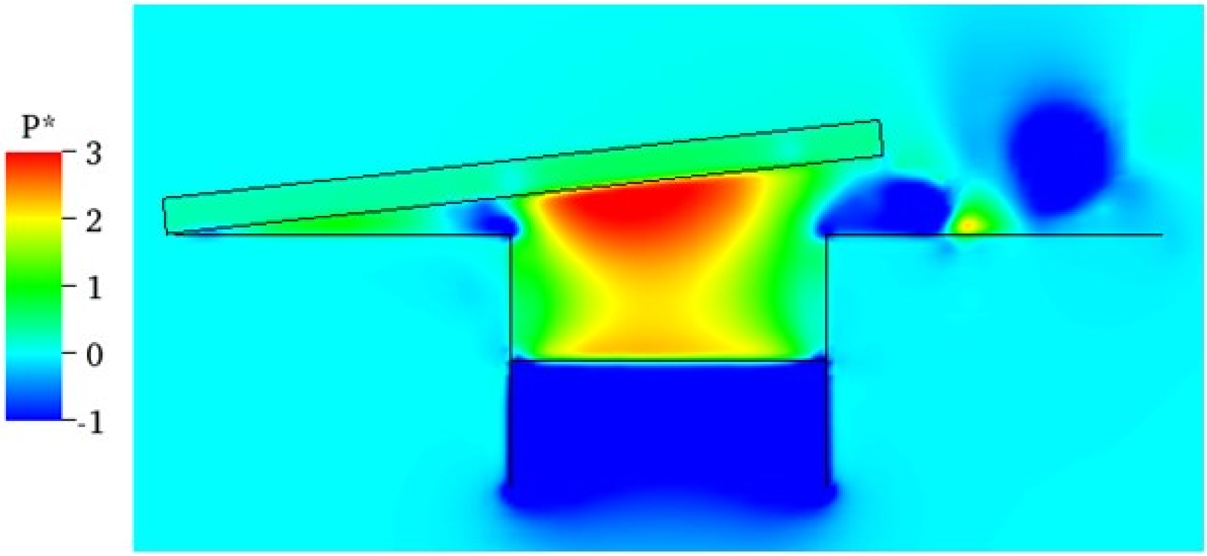

Figure 2 – Pressure field for Reynolds number (based on the inlet parameters) equal to 3,000.

REFERENCES

[1] Z. Wang, J. Fan, and K. Luo, Combined multi-direct forcing and immersed boundary for simulating flows with moving particles. *International Journal of Multiphase Flow*, Vol. **34**, pp. 283–302, 2008.

[2] U.M. Ascher, S.J. Ruuth, and B. Weeton, Implicit-explicit methods for time-dependent PDE´s, *SIAM, Journal Numerical Analysis*, Vol. **32(3)**, p. 797-823, 1995.