Proceedings of the 4th International Conference on Fluid Flow and Thermal Science (ICFFTS'23)
Lisbon, Portugal- December 07 - 09, 2023
Paper No. 138
DOI: 10.11159/icffts23.138

CFD Analysis of a Two-Phase Fluid Flow Compressor

Wenjing Lyu[1], Giovanni Luzi[1], Bernhard Gatternig[2], Antonio Delgado[1]
[1]LSTME Busan,
1276 Jisa-Dong, Gangseo-Gu, Busan, Republic of Korea
wenjing.lyu@lstme.org; giovanni.luzi@lstme.org; antonio.delgado@lstme.org
[2]Hochschule Weihenstephan-Triesdorf,
Markgrafenstraße 12, Weidenbach, Germany
bernhard.gatternig@lstme.org

Extended Abstract

The goal of the REGEN-BY-2 Horizon 2020 project is to develop a lab-scale prototype of a highly efficient thermodynamic cycle and related plant for the exploitation of renewable thermal energy sources. The whole thermodynamic cycle incorporates Carnot cycles operating with fluid in two-phase flow conditions that circulate through expanders and compressors. The aim of this work is to numerically investigate the two-phase fluid flow in a newly designed two-phase scroll compressor. Numerical simulations represent a reliable tool for the design of Positive Displacement (PD) machines. Therefore, a detailed numerical analysis of a two-phase scroll compressor is necessary for an accurate investigation of the multiphase flow phenomena that occur while the machine is in operation and the finalization of the machine design. To this end, we developed an in-house Cavitating Dynamic Mesh Solver (CDMS) that can predict the phase change phenomena inside the chambers of the compressor. It is a transient cavitation solver based on the Homogeneous Equilibrium Model (HEM) [1] with barotropic closure [2]. The solver considers mesh motion and allows for mesh topology changes. The dynamic grids of the orbiting turbomachine are generated using the OpenFOAM® utility *blockMesh* which allows for creating meshes with multiple blocks. The Arbitrary Coupling Mesh Interface (ACMI) is utilized to connect adjacent mesh domains and it is particularly suitable for rotating geometries. The key numerical assessments of the two-phase compressor in the current investigations are the pressure and velocity distribution, liquid evaporation and vapor condensation phenomena, the average inlet and outlet mass flow rates, and the isentropic and volumetric efficiencies. Those are necessary to identify whether the machine is working under overloaded or overheated conditions, or the working fluid causes severe vibrations. The average inlet and outlet mass flow rates, as well as the isentropic and volumetric efficiencies obtained numerically, are in good accordance with the results of a deterministic model (DM) available in the literature [3]. Besides, the distribution of pressure and velocity inside the compressor agrees qualitatively well with the results of numerical studies of scroll compressors available in the literature [4, 5]. The high isentropic and volumetric efficiencies indicate that the investigated compressor type has been well-designed and is capable of efficiently compressing the refrigerant while delivering the desired mass flow rate at the specified pressure and temperature.

Ackowledgements

This work was supported by the National Research Foundation of Korea (NRF) grant funded by the Korean government (MSIT) (No. 2022K1A3A1A7809107511)

References

[1] B. Biçer and A. Sou, "Application of the improved cavitation model to turbulent cavitating flow in fuel injector nozzle," *Applied Mathematical Modelling*, vol. 40, 7-8, pp. 4712–4726, 2016.
[2] F. J. Salvador, J.-V. Romero, M.-D. Roselló, and J. Martínez-López, "Validation of a code for modeling cavitation phenomena in Diesel injector nozzles," *Mathematical and Computer Modelling*, vol. 52, 7-8, pp. 1123–1132, 2010.
[3] N. Leclerq, V. Lemort "Modeling and Simulation of a Two-phase Scroll Compressor," *26th International Compressor Engineering Conference at Purdue*, Purdue, IN, 2022, vol. 2545, p. 10.

[4] G. Cavazzini, F. Giacomel, A. Benato, F. Nascimben, and G. Ardizzon, "Analysis of the Inner Fluid-Dynamics of Scroll Compressors and Comparison between CFD Numerical and Modelling Approaches," *Energies*, vol. 14, no. 4, p. 1158, 2021.
[5] H. Gao, H. Ding, and Y. Jiang, "3D Transient CFD Simulation of Scroll Compressors with the Tip Seal," *IOP Conf. Ser.: Mater. Sci., Eng.,* Macau, China, 2015, vol. 90, p. 12034.