실제기체모델이 적용된 OpenFOAM 기반 연소 해석 솔버 개발

Danh Nam Nguyen, 정기성, 심재원, 유춘상

A new method to implement real gas models for thermophysicalmodels library in OpenFOAM

Danh Nam Nguyen, Ki Sung Jung, Jae Won Shim, Chun Sang Yoo

OpenFOAM [1], an open-source computational fluid dynamics (CFD) tool, has been widely used to analyze multi-dimensional problems with complex physical models in continuum mechanics. In combustion applications, sophisticated models for real gas effect consideration are frequently required to numerically investigate the fundamental combustion characteristics of modern combustion devices such as gas turbine engines and rocket engines [2]. However, there is no set of real gas models available for combustion solvers in the *thermophysicalModels* library yet. Here we proposes a novel approach to implement real gas models in OpenFOAM 6.0, especially for combustion simulations at high pressures. A set of models including modified Soave-Redlick-Kwong (SRK) equation of state (EoS), Chung model for dynamic viscosity and thermal conductivity and Takahashi for binary diffusion coefficients is adopted because of their validity over a wide range of fluid state and its easy implementation [3]. The implementation of real gas models in OpenFOAM is challenging not only because the mixing rules of evaluating thermophysical properties of a mixture differ from model-by-model for most sets of real gas models, while the *thermophysicalModels* library in OpenFOAM only provide a simple mixing rule based on mass fraction weighted average. In OpenFOAM, the thermophysical properties of a mixture are obtained by employing overloading operator functions "*" and "+=". However, these functions are not efficient for complicated mixing rules such as those in Chung's model. To resolve the problem, we propose an alternative approach by adopting the methods of *Void* type to update coefficients for mixture instead of using the existing operator functions. In addition, to avoid the dependency of updated coefficients for a mixture on temperature and pressure, those in an EoS model should be decomposed into temperature and pressure independent coefficients. By applying this approach, a new real-fluid based *thermophysicalModels* library including a set of the above real gas models is implemented into OpenFOAM, which is then validated against the data of NIST [4] and real-fluid based OPPDIF [5, 6]. The present results show a quantitatively good agreement with the benchmark data over a wide range of temperature and pressure (1 – 300 atm), demonstrating the outstanding performance of the new library and the applicability of the proposed approach.

Acknowledgement

This study was supported by Basic Science Research Program through the National Research Foundation of Korea (NRF) funded by the Ministry of Science and ICT (NRF-2021R1A2C2005606).

References

[1] The OpenFOAM Foundation. URL https://openfoam.org/

[2] 2010, J. Foster, R. S. Miller, *High Pressure Processes in Chemical Engineering*, pp. 53–75.

[3] 2008, G. Ribert, N. Zong, V. Yang, L. Pons, N. Darabiha, S. Candel, *Combustion and Flame* Vol. 154, pp. 319–330.

[4] 2017, E. Lemmon, M. McLinden, D. Friend, *NIST Chemistry WebBook, NIST Standard Reference Database.*

[5] A. E. Lutz, R. J. Kee, J. F. Grcar, F. M. Rupley, OPPDIF: A Fortran program for computing opposed-flow diffusion flames, United States.

[6] 2021, K. S. Jung, H. S. Bak, D. N. Nguyen, B. Lee, C. S. Yoo, *Fuel*, (accepted)

울산과학기술원 기계공학과 * Corresponding author E-mail: csyoo@unist.ac.kr, Tel : 052) 217-2322