Proceedings of the 8th World Congress on Mechanical, Chemical, and Material Engineering (MCM'22)
Prague, Czech Republic – July 31, 2022 - August 02, 2022
Paper No. HTFF 137
DOI: 10.11159/htff22.137

Numerical Simulation through Fluent Of a Cold, Swirling Particle Flow in a Combustion Chamber

Wronski Tomek[1], Zouaoui-Mahzoul Nabila[1], Schönnenbeck Cornelius[1], Brillard Alain[1]
[1]Laboratory Gestion des Risques et Environnement, Université de Haute-Alsace
2bis rue des Frères Lumière, 68093 Mulhouse, France

Extended Abstract

The study aims to first model the cold, confined and swirling flow in a magnesium burner. One of the most important features of the experimental burner is the presence of a significant recirculation zone, which is crucial for stabilizing the flame in the combustion chamber, [1]. Before modelling the combustion reaction and the flame, a first step is the simulation of the cold monophasic airflow and the accurate simulation of the recirculation zone. To validate the numerical simulations performed with the ANSYS Fluent software, experimental velocity measurements were first made in a 1:1 scale PMMA replica of the experimental burner. A constant temperature hot-wire anemometer was used to determine radial profiles of axial velocity. A low swirl case (S=0.13) was first considered because of its apparent simplicity. Simulation results obtained using different eddy viscosity models were compared to the experimental data and the Standard k-ε model proved to predict the velocity profiles and the central recirculation zone with the most accuracy. A high swirl case was then studied (S=2.94), corresponding to the conditions occurring in the experimental burner. In this case, the turbulence and its anisotropy appeared to be too strong for the eddy viscosity models previously used, and they failed to provide a converging solution. The RSM model was better suited for the task and could predict the position, size and shape of the central toroidal recirculation zone with acceptable accuracy, although important errors were still observed for the velocity values. Part of the inaccuracies could be explained by the usage of first-order discretization schemes. Higher discretization orders (second or higher order) could not be used in this case due to induced numerical instabilities, which could not be reduced despite several attempts, [2].

After the modelling of the monophasic airflow, the second step is the addition of the magnesium particles to simulate the diphasic cold flow. The Discrete Phase Model, which uses a Lagrangian approach to predict the path of the particles, appears to be the most appropriate. Knowing the volume fraction of particles in the airflow, a two-way interaction between the continuous and the discreet phases is to be considered. While the airflow remained stationary, an unsteady treatment of the particles was applied. No experimental measurements could be done to validate the numerical results, but these could still provide useful insights on the particle flow inside the burner. It was shown that smaller particles are strongly influenced by the airflow: they can be trapped in the recirculation zone or be ejected towards the walls of the chamber. The heavier particles are more influenced by gravity and tend to drift towards the bottom of the chamber. The choice of the size distribution of the particles thus has a great impact on the local air/fuel ratio.

References

[1] D. Laraqui, G. Leyssens, C. Schonnenbeck, O. Allgaier, R. Lomba, C. Dumand, J.-F. Brilhac, *Heat recovery and metal oxide particles trapping in a power generation system using a swirl-stabilized metal-air burner*. Appl. Energy. 264 114691, 2020.

[2] T. Wronski, N. Zouaoui-Mahzoul, C. Schönnenbeck and A. Brillard, *Numerical simulation through Fluent of a cold, confined and swirling airflow in a combustion chamber*. Paper submitted to publication, 2022.