Investigation of Flow Dynamics and Combustion Characteristics in Shower and Rocket Type Burners Using CFD Simulations

Arthit Sangsopha *

Combustion and Jet Application Research Laboratory (CJARL) Department of Mechanical Engineering, Faculty of Engineering, Ubon Ratchathani University, Warinchamrap, Ubon Ratchathani, Thailand

Anirut Matthujak

Combustion and Jet Application Research Laboratory (CJARL) Department of Mechanical Engineering, Faculty of Engineering, Ubon Ratchathani University, Warinchamrap, Ubon Ratchathani, Thailand

Thanarath Sriveerakul

Combustion and Jet Application Research Laboratory (CJARL) Department of Mechanical Engineering, Faculty of Engineering, Ubon Ratchathani University, Warinchamrap, Ubon Ratchathani, Thailand

Sutthisak Phongthanapanich

Department of Mechanical Engineering TechnologyKing Mongkut's University of Technology North Bangkok, Bangsue, Bangkok, Thailand

Abstract:

This study addresses the need for energy-efficient and low-emission combustion technologies by investigating the flow dynamics and combustion characteristics of shower and rocket-type ceramic burners. Traditional burners face challenges with inefficient air-fuel mixing, leading to higher fuel consumption and emissions. Using Computational Fluid Dynamics (CFD) simulations, the study analyzes fluid velocity, temperature distribution, and combustion efficiency under varying fuel power inputs for both burner types. The simulations revealed that the rocket burner outperformed the shower burner, achieving a maximum velocity of 5.6 m/s and a peak temperature of 1,396.75 K, compared to the shower burner's 4.4 m/s and 1,361.75 K. These results suggest that the rocket burner's design promotes better air-fuel mixing and higher combustion efficiency, particularly at elevated power inputs. Validation against experimental data showed minimal errors—6.52% for the shower burner and 5.7% for the rocket burner—indicating the reliability of the CFD models. The findings highlight the rocket burner's suitability for high-temperature applications, where sustained combustion intensity is essential. In contrast, the shower burner exhibited lower performance, limiting its use in high-thermal-output processes. This research underscores the significance of burner design in optimizing combustion performance, offering valuable insights for developing energy-efficient burners. The study also demonstrates the effectiveness of CFD simulations as a cost-effective tool for burner optimization, contributing to reduced fuel consumption and emissions in industrial combustion systems.

Keywords:

Ceramic burners, Computational Fluid Dynamics (CFD), Fluid velocity, Temperature distribution, Rocket burner, Shower burner.