Modeling of flow features in burners' air supply channel

Kęstutis Račkaitis, Robertas Poškas, Povilas Poškas

Lithuanian Energy Institute, Nuclear Engineering Laboratory, Breslaujos 3, LT-44403 Kaunas, Lithuania,

Email: kestutis.rackaitis@lei.lt

INTRODUCTION

In industrial facilities, the efficient operation of burners is paramount to ensuring consistent and effective combustion processes. In systems where a single air supply duct serves multiple burners, precise airflow management becomes even more critical. In this paper, we consider a duct with one air entry and two outlets (Fig. 1), each directing air to separate burners. The design of such a system must ensure that each burner receives the same amount of air to maintain proper combustion conditions, minimize emissions, and maximize energy efficiency. Numerical simulations play a vital role in achieving this by enabling engineers to model the airflow dynamics, pressure distribution, and temperature variations within the duct. By simulating different operational conditions, we predict how the air is split between the two outlets and identify potential issues such as possible uneven mass flow distribution. These simulation help refine the design before physical implementation, ensuring the impeccable operation of burners in the system.

METHODOLOGY

During computer model creation, the initial design was simplified. It was decided not to include in the simulations all parts through which the air flows to the fan, including the fan itself. Instead of the fan, a rectangular air duct was bent to replicate the fan's circular shape, and the inlet boundary conditions were set at the beginning of the duct (Fig. 2). Behind the fan, all parts of the duct were left unchanged, except for the outlet. To obtain more accurate modeling results in the absence of the burner geometry, the outlet ducts were extended by more than two hydraulic diameters. In this way, the unwanted influence of the pressure outlet boundary conditions on the solution is avoided. The mesh was prepared taking into account the structure of the duct, specifying one inlet (green plane in Fig. 2) and two pressure outlets (blue planes in Fig. 2). In narrower sectionds of the duct, or locations with finer details (such as guiding plates), the mesh was densified. To verify the reliability of the solution, several meshes were created to ensure sufficient cell density in complex flow areas. The simulations were performed using ANSYS Fluent and ANSYS Discovery programs under steady-state conditions and only the Discovery program for transient conditions. The modeling was performed for four flow rates to reflect different operating conditions, with an air inlet temperature of 20 °C.

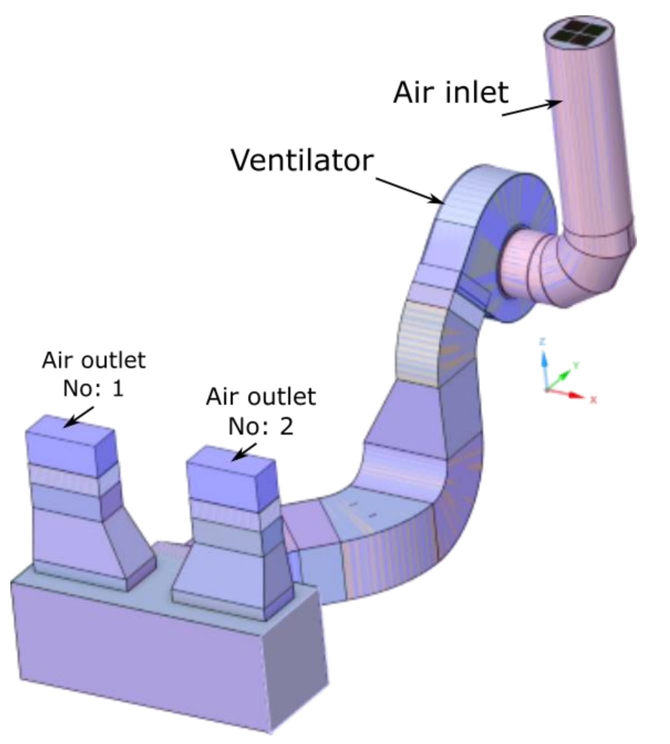

Fig. 1. Real design of air supply duct

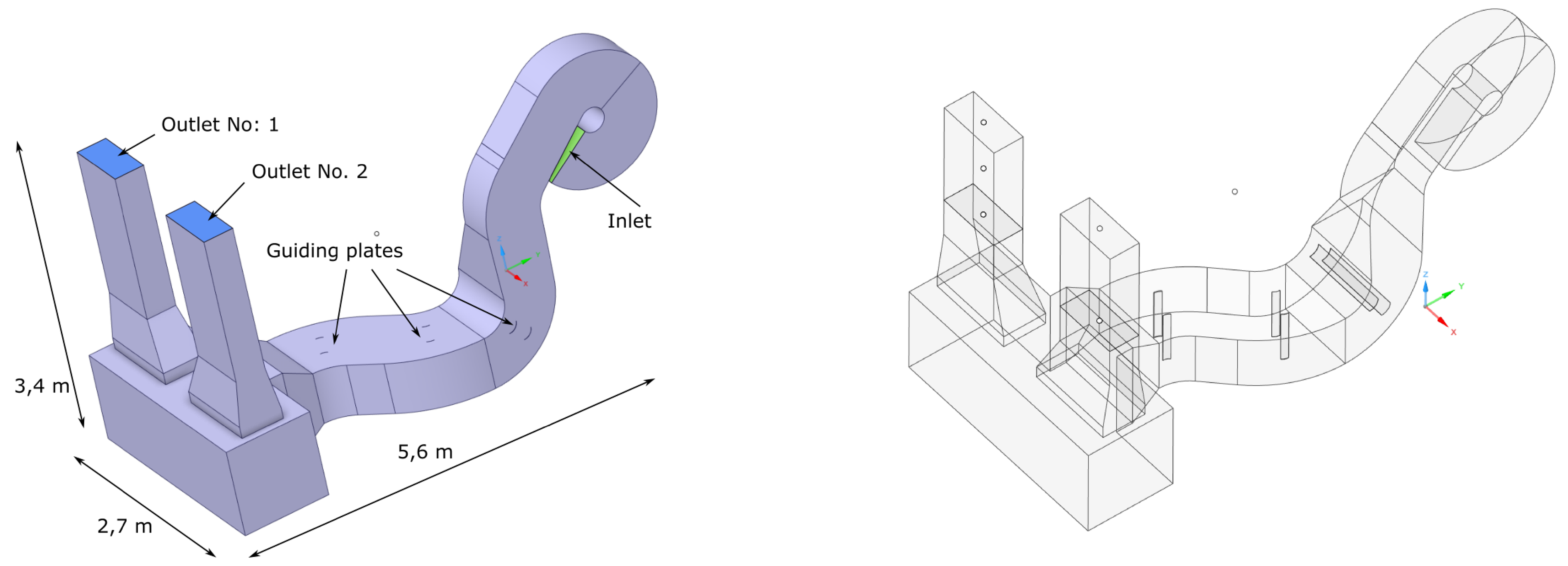

Fig. 2. Model for numerical simulations

RESULTS AND DISCUSSION

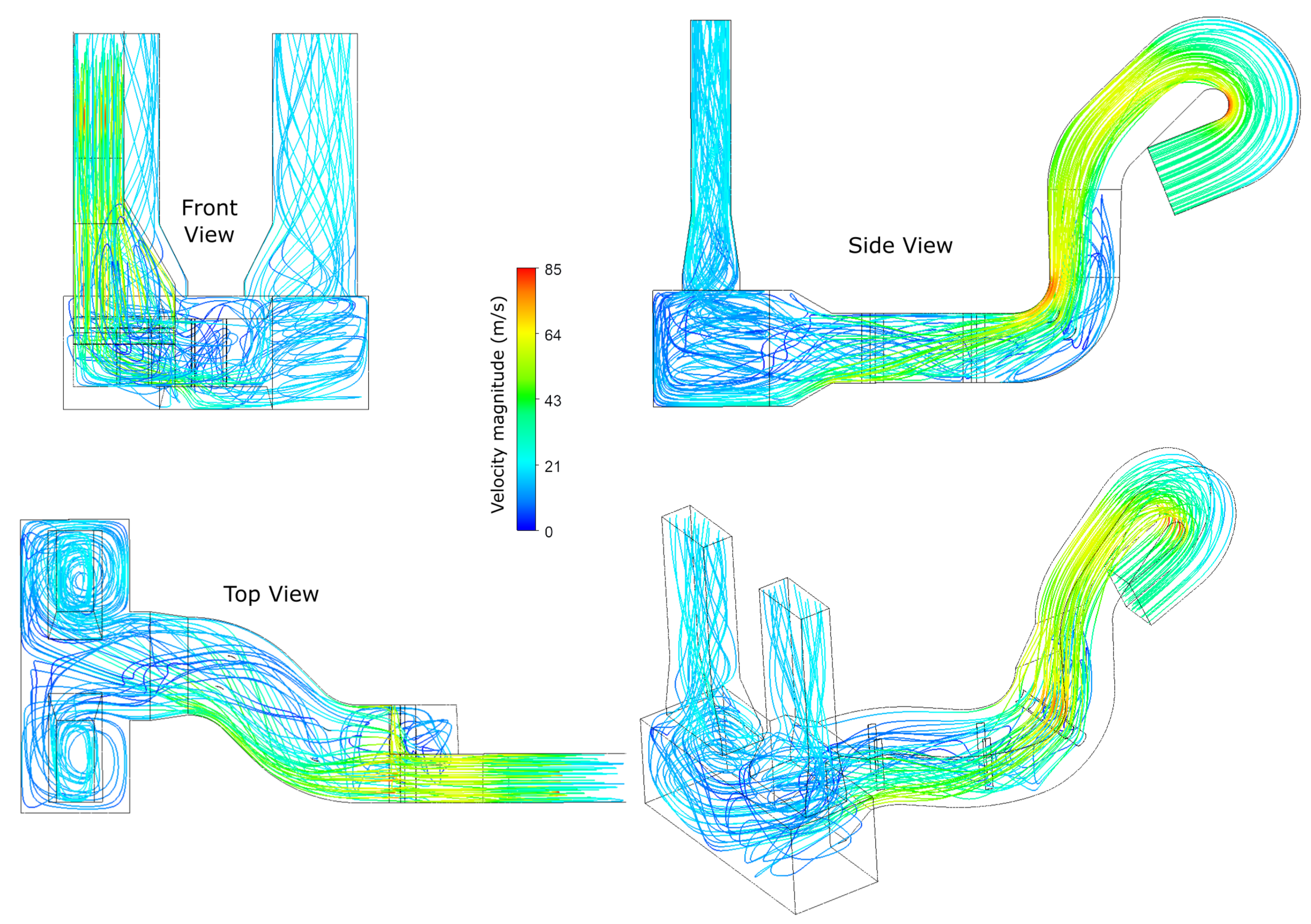

Fig. 3. Velocity streamlines from steady state simulation, air flow rate 34900 m³/h;

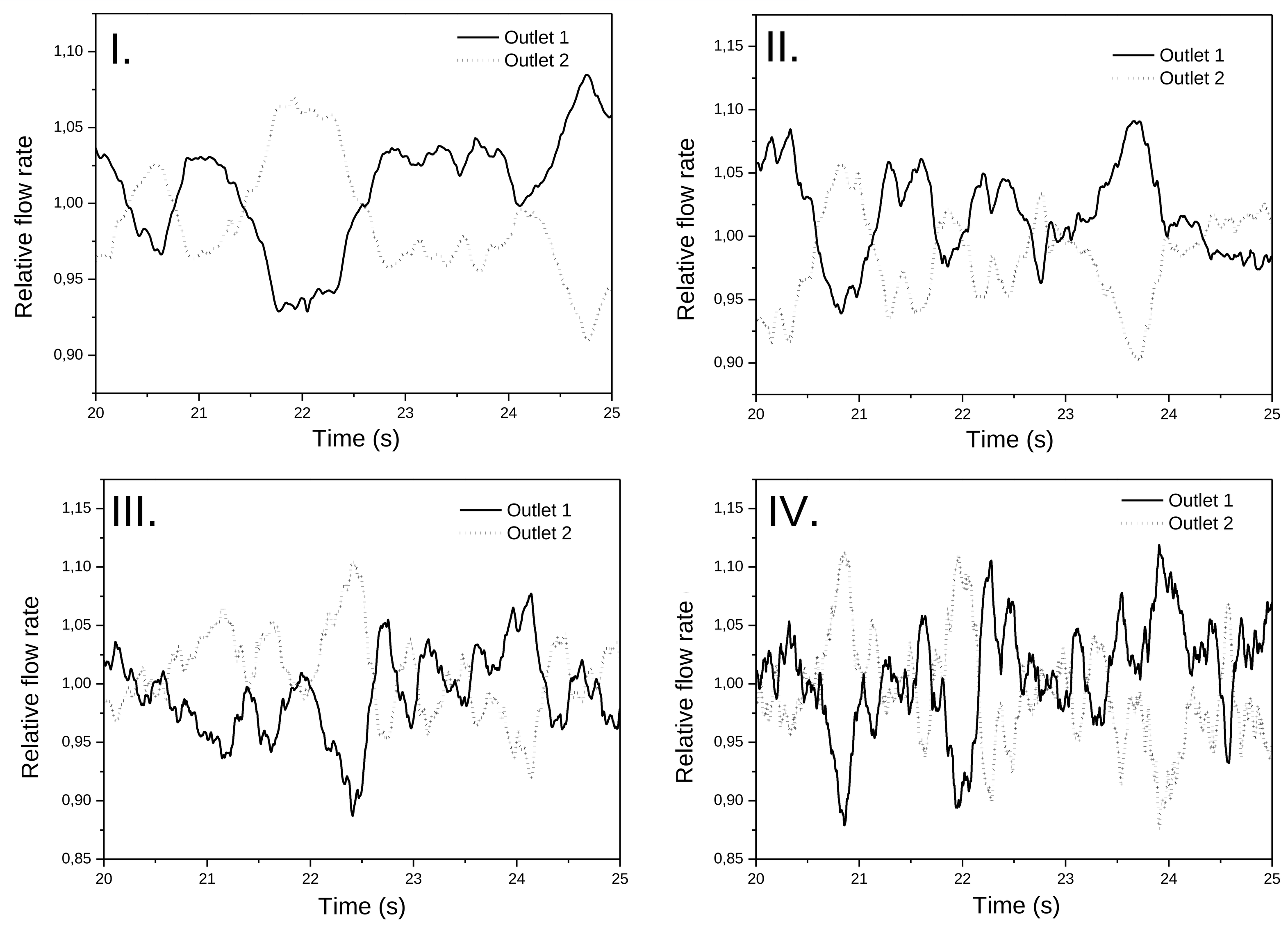

Fig. 4. Transient simulations. Relative flow rate pulsations at channel outlets when flow rates are: I – 6580 m3/h; II – 11850 m³/h; III – 18520 m³/h ; IV – 34900 m³/h;

Steady-state simulations (Fig. 3) showed that the air supply channel was appropriately designed as it distributes air equally through two outlets. However, transient simulations showed some flow rate pulsations between the outlets. The frequency of these pulsations depended on the flow rate. At lower flow rates, the frequency is about 1 Hz, while at the maximum flow rate, it increased to 4 Hz. The amplitude of the pulsations could reach up to 10%.

LITHUANIAN ENERGY INSTITUTE

Contact information:
Kęstutis Račkaitis
Researcher,
Lithuanian Energy Institute,
Breslaujos st. 3, LT-44403, Kaunas
kestutis.rackaitis@lei.lt

RAD CONFERENCE
THIRTEENTH INTERNATIONAL CONFERENCE ON RADIATION, NATURAL SCIENCES, MEDICINE, ENGINEERING, TECHNOLOGY AND ECOLOGY
JUNE 16-20, 2025
HUNGUEST HOTEL SUN RESORT, HERCEG NOVI, MONTENEGRO