Hessisches Kompetenzzentrum
für Hochleistungsrechnen

Optimisation of the Control of Waste Incineration Plants Through Dynamic Process Modelling and the Use of Innovative Monitoring Methods

Project Manager
Prof. Dr.-Ing. Bernd Epple

Principal Investigator
Yannik Lichtmannegger

Project Term
2024 - 2025

Clusters
Lichtenberg II Cluster Darmstadt

Software
ANSYS

Institute
Institut Energiesysteme und Energietechnik

University
Technische Universität Darmstadt

Introduction

The efficient and environmentally friendly operation of waste incineration plants is becoming increasingly important. In order to enhance the performance of these plants and optimize their components, it is essential to comprehend the flow of gases and the heat exchange processes occurring within the system. Experimental investigations frequently incur significant costs and require considerable time. In addition, such investigations are not always feasible due to technical constraints. Consequently, numerical simulations, particularly Computational Fluid Dynamics (CFD), present a viable alternative. CFD facilitates the examination of the impact of design modifications or operational parameters on the performance of a plant, obviating the necessity for physical prototypes. In the context of complex simulations, the tube bundle heat exchanger represents a significant computational challenge. The complex geometry of these bundles requires a high computing power, as the mesh needs to be refined around the geometry. In order to reduce the effort involved, the tubes can be modelled as a porous medium, a simplified approach that approximates the effect of the tubes on the flow without representing each tube individually. In order to ensure that the simplification still accurately reflects the system, this approach necessitates validation. In this context, the use of a high-performance computer (HPC) is essential, since even simplified simulations require millions of calculations that cannot be performed efficiently on standard computers.

Methods

The study is based on CFD simulations performed using Ansys Fluent. Two approaches were compared: a detailed model representing every tube in the bundle, and a simplified model representing the tubes as a porous zone. The porous zone was defined using custom functions to account for flow resistance and heat transfer. Simulations were performed for a grate-fired waste incineration plant. To focus on the effects of the tube bundles, the influence of other factors was systematically excluded. The main parameters analyzed were the distribution of flow velocity and pressure drop across the bundle. By comparing the simplified and detailed models, the accuracy of the reduced approach could be evaluated.

Results

The comparison demonstrates that the simplified porous model can reproduce the main flow characteristics of the detailed tube bundle. A comparative analysis was conducted to ascertain the extent of any discrepancies in pressure loss and flow distribution. The findings showed that these variations were small and fell within the acceptable limits. The simplified model resulted in a substantial reduction in the required computational time, thereby enabling faster simulations of the entire plant. This facilitates the rapid testing of numerous design variations and operating conditions, thereby reducing the time required for a comprehensive model by a significant margin.

Discussion

The results show that modelling tube bundles as a porous medium is an effective method for conducting CFD simulations of waste incineration plants. This simplification reduces computational costs while maintaining the level of predictive accuracy required for engineering decisions. The use of HPC resources was essential to handle the large number of calculations and to explore multiple scenarios efficiently. In the future, this approach can be extended to other plant components, thereby supporting the optimization of new and existing facilities and helping to improve energy efficiency and reduce emissions.

Last Update: 2025-10-22 09:13