ulm university universität ulm

Institute of Chemical Engineering

Masterthesis

Theoretical analysis of the radiation field of a mercury vapor lamp with OpenFOAM®

Prof. Dr. Dirk Ziegenbalg
Albert-Einstein-Allee 11
89081 Ulm

Contact person:
Florian Gaulhofer
April 22, 2020

Motivation:

The development of highly efficient photoreactors takes great interest in the current research of chemical engineering. A profound understanding for the relevant physical effects occuring in photoreactors to derive general design criteria for photochemical processes is crucial.
The numerical analysis of continuum mechanics problems in science and engineering represents a vital research field and is called Computational Fluid Dynamics (CFD). The current challenge of modeling photochemical processes is the coupling of physical effects at the same time and space. OpenFOAM® offers a toolbox to solve multi-physics problems.

Current scientific work:

The aim of this thesis is the analysis of the radiation field of an mercury vapor immersion lamp with OpenFOAM®. Therefore a calculation grid in 2D and 3D has to be created. Afterwards the radiation field is calculated with OpenFOAM® for different geometries with variable boundary conditions. Finally, the simulations are compared with experimental data.

This work includes:

- Familiarisation with the OpenFOAM® environment
- Creation of the 2D/3D calculation grid
- Simulation of the radiation field with OpenFOAM® solver

Qualification

The work requires handling of the Unix operating system, programming skills (C++) and fun in solving mathematical-physical problems.

References

[1] Orlando M. Alfano and Roberto L. Romero and Alberto E. Cassano. Radiation field modelling in photoreactors—I. homogeneous media. Chemical Engineering Science, 1986.
DOI:10.1016/0009-2509(86)87025-7

[2] José Moreno and Cintia Casado and Javier Marugán. Improved discrete ordinate method for accurate simulation radiation transport using solar and LED light sources. Chemical Engineering Science, 2019.
DOI:10.1016/j.ces.2019.04.034

[3] OpenFOAM - The OpenFOAM Foundation - User Guide, version 7.