Chair of Renewable and Sustainable Energy Systems
TUM School of Engineering and Design
Technical University of Munich

TUM

Research Internship / Forschungspraxis

CFD simulation and optimization within the framework Ansys optiSLang

Background

Computational Fluid Dynamics (CFD) tools are used in numerous research and industrial areas to analyze natural phenomena or technical systems. Ansys Fluent* is one of such tools for simulating fluids, such as air flow in a room (Figure 1). CFD simulations provide results about the parameters of interest (e.g. air velocities) for a given set of design/input parameters (e.g. window size). However, to find the best design, an optimization procedure has to be conducted. Ansys optiSLang** is an optimization tool that is compatible with Ansys Fluent and offers many different functionalities. The main motivation behind this project is to test these functionalities in order to check whether optiSLang is suitable for finding optimal designs of indoor (vertical) farming units (boxes) in the future.

Goals

The goal of the internship is to implement a simple model of a rectangular box (including heat and air sources/sinks) in Ansys Fluent, in order to enable the analysis of optimization capabilities of Ansys optiSLang. This includes following tasks:

Figure 1

- Familiarizing with Ansys Fluent / optiSLang
- Implementing the CFD box model in Ansys with proper boundary conditions
- Defining different optimization problems and solution strategies in optiSLang
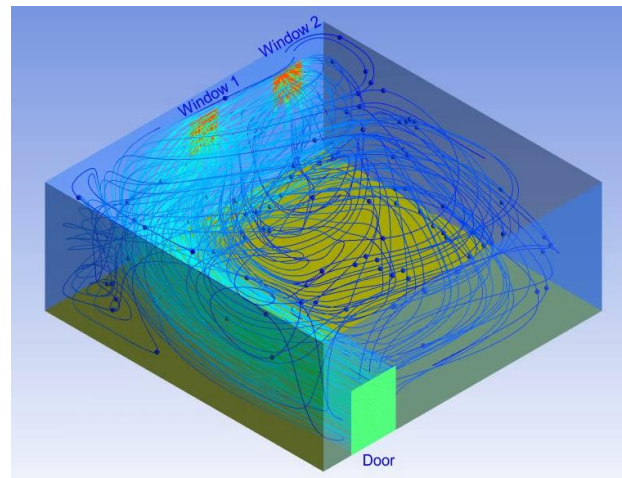- Testing the optimization capabilities

Requirements

- Interest in CFD modeling and simulation
- Interest in numerical optimization
- Software skills: Ansys or similar CFD tools
- Knowledge of fluid flow and heat transport modeling / optimization methods (is a plus)
- CV and grade report

Contact

Smajil Halilovic, M.Sc.
Lichtenbergstr. 4a, 85748 Garching b. München, Raum 2016
E-Mail smajil.halilovic@tum.de

* https://www.ansys.com/en-gb/products/fluids/ansys-fluent ** https://www.ansys.com/products/platform/ansys-optislang