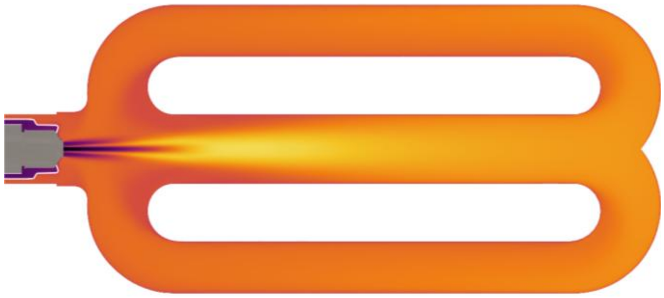
Bachelor-/Masterthesis

Experimental investigation and numerical modeling of cold-flow recirculation in a PP-type radiant tube

Indirect heating systems are required in industrial heating processes where the product shall be protected from combustion flue gases. In continuous annealing lines this is typically assured by gas burners firing in radiant tubes, where the heat is transferred from tube to furnace and product primarily via thermal radiation. These systems are the subject of several research projects at the Department for Industrial Furnaces and Heat Engineering. Besides life time predictions based on creep deformation investigations, an ongoing topic is the decarbonization of the combustion process by substituting natural gas by potentially carbon-neutral fuels as hydrogen or ammonia.

In PP-type radiant tubes, the combustion flue gases are recirculated to dilute the reaction zone, promoting temperature homogenity, reducing peak temperatures, and thereby mitigating the formation of NOx emissions, which are harmful to the environment and are regulated by the authorities. Consequently, an accurate prediction of the recirculation is crucial to obtain precise predictions for the NOx formation from numerical simulations of such system.



The student thesis shall improve the recirculation prediction in computational fluid dynamics (CFD) simulations of a PP-type radiant tube using the Reynolds-Averaged Navier-Stokes approach. Recirculation is driven by the suction of flue gases into the flame, where the interaction of phenomena as aerodynamics, chemical reactions, and heat transfer is complex in the physical system as well as in the numerical simulation. To isolate the effects, cold-flow experiments will be conducted at the departments technical center, where air is fed to the radiant tube system without combustion and recirculation is measured in the radiant tube's lower leg. A CFD simulation will then be set up in the commercial software Ansys Fluent to validate the computational mesh and modeling of turbulence. The scope of the work is adapted to the working hours stipulated in the respective examination regulations.

Your profile:

- Good English skills
- Reliable and independent way of working
- Basic knowledge of fluid dynamics
- Experience in CAD and CFD are advantageous

Duration: 3 – 6 Month

Start: from now

Questions and further information:

Johannes Losacker, M.Sc.
Department for Industrial Furnaces and Heat Engineering
Group: Combustion
Office 01-205
Phone: +49 241 / 80 26052
E-Mail: losacker@iob.rwth-aachen.de

Further information and thesis at
www.iob.rwth-aachen.de