Proceedings of CHT-17
ICHMT International Symposium on Advances in Computational Heat Transfer

May 28-June 1, 2017, Napoli, Italy

CHT-17-087

REDUCED MODELS FOR CALCULATION OF RADIATION-TUBE BURNERS - THE DEFLECTING ZONE

Aline Jünger[*,§], Ingo Riehl[*], Ulrich Gross[*] and Tobias Fieback[*]
[*]Chair of Technical Thermodynamics, TU Bergakademie Freiberg, Freiberg, Germany
[§]Correspondence author. Fax: +49 3731 39 2161 Email: aline.juenger@ttd.tu-freiberg.de

Gas-fired radiation tube burners are used in process, chemical and blast furnace engineering to ensure the high quality of products. They have a closed outer casing manufactured of SiSiC-ceramics, which separates exhaust gases from the process room. This special design consists of a two-stage natural gas burner with internal exhaust gas recirculation and a counter-current flow coaxial heat exchanger to preheat air.

The radiation-tube burner, illustrated in Figure 1, consists of two consecutive sections: The first one is built from three concentric tubes: the inner one is the burning tube, which acts as open combustion chamber, a finned tube (heat exchanger) separating air flow (inside) from the counter-current exhaust gas flow (outside) and the radiation tube acting as the outer casing, which continues to the second section. It contains the coaxial flame tube as the second-stage burning-zone, where the recirculation of the exhaust gases takes place. The outer casing is closed at its end, where the deflecting zone forces back flow of the exhaust gases through the outer annulus. Figure 1 is demonstrating schematically a typical temperature distribution on the outer surface of the radiation tube. There exist three temperature maxima: a mean peak at the deflecting zone and two local maxima located at the second stage burning zone, at the beginning of the flame tube. This inhomogeneous distribution causes thermal stresses and could lead to damage of the outer casing. Steady state 3D-CFD methods using FVM were made to study the involved flow dynamic and thermal phenomena to improve the temperature distribution and to increase the heat transfer at the same time.

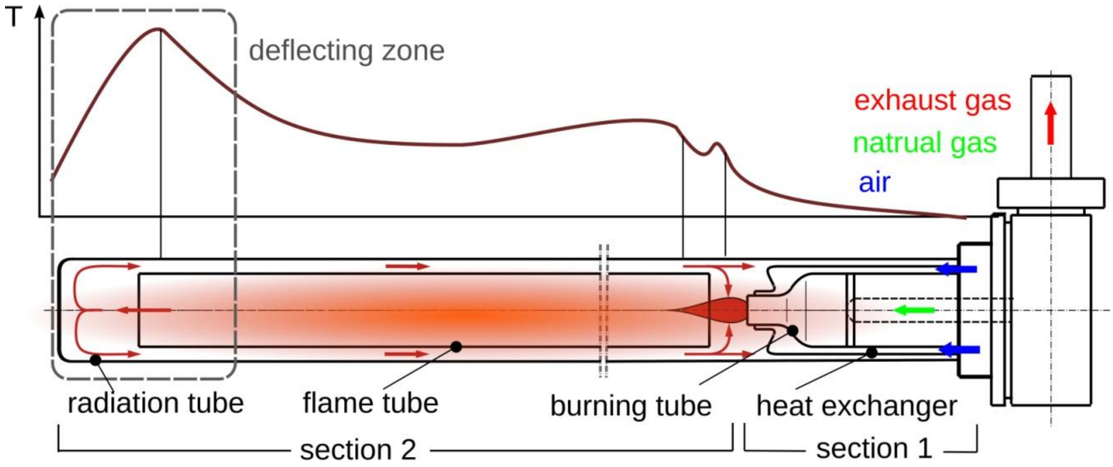

Figure 1. Components of a radiation-tube burner and a typical temperature distribution on the surface of the outer casing

THE DEFLECTING ZONE

By 3D-CFD modelling the initial situation has been analysed and after that a basic study of the influences of a reconstruction of the deflecting zone has been performed as a parameter study. As the result of the optimization process a well-shaped stagnation-point shape followed by a 180 ° bend has been obtained leading to a homogeneous temperature distribution on the outer surface of the radiation tube. To validate the stagnation point flow, 3D-CFD simulations for the Reynolds number of 50.000 and 100.000 are tested using four RANS turbulence models to predict the heat transfer and pressure loss measured by Pape [2008, 2004]: ke, lke, V2F and AKN. The ke-model with adapted coefficients by Pape [2004] is used to study influences of geometric parameters for the guided 180 ° bend. A resulting prototype based on this improved geometry of the radiation and flame tube was made of steel. Surface temperature on the outer casing and static pressure were measured under real operating conditions of a radiation-tube burner. Due to the strong interactions between the deflecting zone and the mass flow of recirculating exhaust gases, a reduced model of the deflecting zone with respect to changing inlet conditions is developed. For the improved geometry of the guided 180 ° bend pressure loss and heat transfer coefficients are predicted using the reduced model. The pressure loss is additionally been calculated by using the correct steady-state solution of the Navier-Stokes equations and the assumption of a rotational symmetric stagnation point flow. The heat transfer coefficient is predicted by using empirical Nusselt formulations for nozzles, which were compared with numerical results of the validation process.

Pape, D., 2004. Influence of the 180° Bend Geometry on the Pressure Loss and Heat Transfer in a High Aspect Ratio Rectangular Smooth Channel. *In*: ASME Turbo Expo 2004, Vol.3*, 14-17June 2004 Vienna, Austria*, 685-695.

Pape, D. [2009], Experimentelle Untersuchung der Strömung und der Wärmeübertragung in 180°-Umlenkungen, *PhD Thesis,* Institute of Aerospace Thermodynamics, Universität Stuttgart.