ASIA-PACIFIC
SOLAR RESEARCH CONFERENCE

Experimental Investigation of the Interaction of the Flow inside and Outside a Cavity Receiver

E. Alipourtarzanagh, A. Chinnici, Z. Tian G. Nathan and B. Dally

Centre for Energy Technology, School of Mechanical Engineering
The University of Adelaide, South Australia 5005, Australia

The objective of this paper is to investigate the flow behaviour inside and outside a Hybrid Solar Receiver Combustor (HSRC) by conducting a systematic experiment and numerical modelling. The HSRC operates under three different modes namely; solar only, combustion only and mixed mode of operation. In the mixed mode of operation, energy is supplied by combustion inside the cavity while the aperture is open to receive the solar radiation. Under this mode, ingress/egress into and from the solar cavity receiver due to the pressure difference between the inside of the cavity and the ambient, leads to convection losses, particularly under high wind velocities. In the experimental part of the study, to evaluate the interaction of the flows through the aperture of the cavity and the flow structure induced by the wind, a simplified scaled down geometry of the solar receiver cavity was constructed. The cylindrical cavity model was 0.074m internal diameter and 0.225 m long with an annular gap as the outlet of the cavity. The inlet to the cavity was blocked with a straight desk which had an aperture of 0.024 m diameter. Four equi-spaced jets were included on the aperture side to model the combustion burners. Two models were investigated, both had jets with an inclination angle of α_{jet}=25° but each had a different azimuth angle, γ_{jet}=0° and 5°. The model was placed in the water channel, under isothermal conditions and velocity of water in the channel aimed to emulate the wind conditions external to the HSRC. Water was also used for the jets in the model cavity. The systematic investigations include different both closed and opened aperture with free stream velocities of 0.0, 0.08, 0.16, 0.24 m/s. The velocity of the jets flow inside the cavity were fixed at 2.6 m/s to secure the aerodynamics similarity between the jets and wind speed in real-world condition and to ensure that the jet flow is in the fully turbulent regime (Re=10,500). The cavity is made of acrylic material that makes it possible to visualise flow behaviour using the Particle Image Velocimetry (PIV) technique. In the numerical part of the study, a Computational Fluid Dynamic (CFD) package is used to investigate the effect of aperture opening, wind speed, jet inclination and azimuth angles on the flow patterns through the cavity opening. The results show that the flow behaviour through the aperture depends on the flow fields outside the cavity in the vicinity of the aperture and that changing the flow pattern inside the cavity through the variation of the jet azimuth angle, changes the flow characteristics inside the cavity. These results point to a complex interaction of the external and internal flows and highlight the need to better understand these interactions. Based on the obtained results the need for the development of a fluidic barrier to de-couple the flow inside and outside the cavity and decrease the convection losses is essential.