World Academy of Science, Engineering and Technology
International Journal of Energy and Power Engineering
Vol:13, No:09, 2019

Analysis of Flow Dynamics of Heated and Cooled Pylon Upstream to the Cavity past Supersonic Flow with Wall Heating and Cooling

Authors : Vishnu Asokan, Zaid M. Paloba

Abstract : Flow over cavities is an important area of research due to the significant change in flow physics caused by cavity aspect ratio, free stream Mach number and the nature of upstream boundary layer approaching the cavity leading edge. Cavity flow finds application in aircraft wheel well, weapons bay, combustion chamber of scramjet engines, etc. These flows are highly unsteady, compressible and turbulent and it involves mass entrainment coupled with acoustics phenomenon. Variation of flow dynamics in an angled cavity with a heated and cooled pylon upstream to the cavity with spatial combinations of heat flux addition and removal to the wall studied numerically. The goal of study is to investigate the effect of energy addition, removal to the cavity walls and pylon cavity flow dynamics. Preliminary steady state numerical simulations on inclined cavities with heat addition have shown that wall pressure profiles, as well as the recirculation, are influenced by heat transfer to the compressible fluid medium. Such a hybrid control of cavity flow dynamics in the form of heat transfer and pylon geometry can open out greater opportunities in enhancement of mixing and flame holding requirements of supersonic combustors. Addition of pylon upstream to the cavity reduces the acoustic oscillations emanating from the geometry. A numerical unsteady analysis of supersonic flow past cavities exposed to cavity wall heating and cooling with heated and cooled pylon helps to get a clear idea about the oscillation suppression in the cavity. A Cavity of L/D 4 and aft wall angle 22 degree with an upstream pylon of h/D=1.5 mm with a wall angle 29 degree exposed to supersonic flow of Mach number 2 and heat flux of 40 W/cm² and -40 W/cm² modeled for the above study. In the preliminary study, the domain is modeled and validated numerically with a turbulence model of SST k-ω using an HLLC implicit scheme. Both qualitative and quantitative flow data extracted and analyzed using advanced CFD tools. Flow visualization is done using numerical Schlieren method as the fluid medium gives the density variation. The heat flux addition to the wall increases the secondary vortex size of the cavity and removal of energy leads to the reduction in vortex size. The flow field turbulence seems to be increasing at higher heat flux. The shear layer thickness increases as heat flux increases. The steady state analysis of wall pressure shows that there is variation on wall pressure as heat flux increases. Shift in frequency of unsteady wall pressure analysis is an interesting observation for the above study. The time averaged skin friction seems to be reducing at higher heat flux due to the variation in viscosity of fluid inside the cavity.

Keywords : energy addition, frequency shift, Numerical Schlieren, shear layer, vortex evolution
Conference Title : ICHTE 2019 : International Conference on Heat and Thermal Energy
Conference Location : London, United Kingdom
Conference Dates : September 25-26, 2019