SPACE SYSTEMS SYMPOSIUM (D1)
System Engineering - Methods, Processes and Tools (2) (6)

Author: Mr. Abhishek Sharma
Indian Space Research Organization (ISRO), India

Mr. Deepak K. Agarwal
Indian Space Research Organization (ISRO), India
Dr. Sunil Kumar S.
Indian Space Research Organization (ISRO), India

ANALYSIS OF FLAME DEFLECTOR PIT SIMULATOR FOR SEMI-CRYOGENIC ENGINE TEST FACILITY

Abstract

An engine or a rocket stage is tested by mounting the engine/stage vertically on test stand, where the combustion gases exhaust down into a plume deflector pit. The deflector pit helps to redirect the exhaust flow safely away from the test facility. The major challenge of conducting long duration static engine test is to protect the flame deflector pit walls from harsh thermal environments generated by plume gases. In a typical high thrust Semi-cryogenic engine facility, the engine is test fired in open 'J' type cross-section deflector pit in which the deflector pit walls are water cooled with a number of holes on the plates through which water flows out and protects the deflector plates from high thermal loads due to impinging hot gases. It is mandatory to assess water cooling requirements and heating rate distribution over deflector plates. In this regard a detailed CFD simulation of the entire test facility considering the rocket plume impingement and the water cooled deflector plates is made. A simulated water-cooled flame deflector pit is designed to study effectiveness of multi orifice cooling for validation of the CFD model.

In the present study, a numerical methodology is developed to capture non-equilibrium multi-phase flow of water sprays in supersonic rocket plume environment.A unified tightly coupled two-phase methodology is developed to analyze multi orifice plate cooling for deflector pit simulator. An Eulerian two phase formulation of ANSYS FLUENT is used to model disperse phase and is coupled to continuous-phase equations through momentum and heat transfer exchange. The mass transfer phenomenon based on Lee Model is used to capture evaporation-condensation physics. Various test cases with varying plume impingement angle and depth are studied to verify the validity of numerical framework. The model is able to accurately predict thermal load on the flame deflector plate and is demonstrated through validation with experimental data from simulator tests. The CFD simulation was able to reproduce results of simulator test-firing for different configurations and could capture orifice water cooling and formation of non-uniform heating zones over the deflector plate.