069-CFD

Numerical Simulation of A Rocket Propulsion Nozzle During the Startup Condition

Ahmed M. Elattar, Ahmed W. Shahin,

Alexandria University, Egypt, el.attar@zoho.com, ahmedshahin71@outlook.com

Supervisor: Yehia A. Eldrainy, Assistant Professor at Mechanical Engineering Department
Faculty of engineering Alexandria University, Egypt, yeldrainy@yahoo.com

In the present study a numerical investigation of transient flows in a rocket convergent-divergent bell nozzle was performed. A two-dimensional unsteady numerical simulation has been carried out over a convergent-divergent nozzle in order to investigate the transient flow characteristics which represent the normal shock wave propagation during rocket start-up process. ANSYS Fluent 17.2 CFD software was used in this simulation. The gas flow through the nozzle is assumed to be isentropic and follows the ideal gas law and the flow domain is bounded by nozzle wall, pressure inlet and outlet boundaries. The results of the pressure ratio distribution along nozzle axis and nozzle side wall with the shock position had been demonstrated. During startup process there is a transition from FSS to RSS pattern as be shown in the simulation results which lead to fluctuation in wall pressure values. It can be also observed that there is an end-effect at the higher pressure ratio when the recirculation bubble opens to the atmosphere.