Launch Environment Water Flow Simulations

Provasi Alberto (1), Bruno Ludovico (2)

(1) Telematic Solutions
Via Gallarate, 205 – 20151 Milano Italy
EMail: alberto.provasi@telematicsolutions.it

(2) Telematic Solutions
Via Gallarate, 205 – 20151 Milano Italy
EMail: ludovico.bruno@telematicsolutions.it

ABSTRACT

This paper describes the use of CFD to simulate the water flow from the nozzle system used in the sound suppression system during pad abort and nominal launch.

The simulations help determine if water from nozzles will impinge on the rocket nozzles and other sensitive ground support elements. One of the crucial ground structures employed at the launch pad is the nozzle system, whose primary objective is to suppress acoustic energy generated by the launch vehicle during pad abort and nominal operations.

The simulations help determine the projectile motion for different water flow rates employed at the pad, as it is critical to know if water will splash on the first-stage rocket engine during liftoff.

The problem considers the transient injection of water into an open space filled with air.

It provides a less conservative prediction as compared to 1-D projectile analysis, thus allowing design margins to be reduced. In general, there is good agreement between the two analyses for the short throw distances. The discrepancies are greater for long throw distances, in which air resistance and drag dominated flows are more significant.

Final step is a dedicated nozzle and feeder tanks design section to increase system performance.