EXPERIMENTAL AND NUMERICAL STUDY OF FLOW IN MULTI-DRIVER LARGE BLAST SIMULATORS

SCHRAML,S.; MERGNAT,D.

The U.S. Army Research Laboratory (ARL) and its predecessor organization, the U.S. Army Ballistic Research Laboratory, have played a significant role in the technology development of the Large Blast/Thermal Simulator (LB/TS), a Defense Nuclear Agency (DNA) testing facility located at the White Sands Missile Range, New Mexico. During the concept and early design phases, the operational characteristics and the primary dimensions of the facility were determined through a combination of small-scale experiments, Computational Fluid Dynamics (CFD) simulations, and a transfer of technology employed at the Centre d'Etudes de Grarnat (CEG). In order to perform this function in a timely and costeffective manner, the experimental work was primarily performed in small, cylindrical shock tubes. The accompanying numerical simulations were performed in a two-dimensional, axisymmetric representation.

With the knowledge that three-dimensional (3-D) numerical simulations would be required for the characterization of the LB/TS, it became necessary to validate some candidate 3-D CFD codes to model the facility in full geometric detail. In order to accomplish this task, the ARL formed a cooperative research agreement with the CEG in which the flow characteristics of the Large Blast Simulator at CEG would be mapped and used to validate the candidate CFD codes.

France performed a series of experiments in the Large Blast Simulator (SSGG) located at the CEG. This facility has been employed for 15 years to simulate shock and blast waves from nuclear explosions. The simulator has a test section area of 70 m2' and can produce a maximum static overpressure of 140 kPa.
The flow characteristics of static and stagnation overpressure were measured at two longitudinal positions in the expansion tunnel of the simulator, one in the test section at 67.5m downstream from the beginning of the expansion tunnel, the other at a position 22.5m downstream from the beginning of the expansion tunnel. Steel instrumentation rakes were designed and fabricated to support a total of 19 pairs of static and Stagnation pressure transducers which were distributed across the expansion tunnel cross section at each of the longitudinal positions. Flow measurements were recorded for static overpressure levels of 20 kPa, 80 kPa, and 120 kPa.

Two candidate CFD codes were used to simulate the experiments performed in the SSGG facility. The first code, SHARC, is an explicit Euler equation solver which has been used in the blast simulation community for many years. The 3-D SHARC calculations performed in the validation tests employed approximately 4 million grid points in a rectangular mesh coordinate system. USA-RG3 is the other CFD code which was validated against the experimental data. The USA code was developed by the Rockwell International Science Center and has been used primarily for aerospace applications. This code is an implicit Navier-Stokes solver which supports a variety of turbulence models. The USA-RG3 model of the SSGG facility contains approximately 1 million grid points in a multiple-block, generalized, curvilinear coordinate system. Each SHARC and USA calculation performed in this validation series required more than 100 CPU hours to complete on a Cray vector supercomputer.

The static and Stagnation overpressure history data measured during the experiments were used to determine the mach number and dynamic pressure histories and the static, Stagnation and dynamic pressure impulses for each experiment. The data from all 19 measurement positions were used to study the uniformity of each measured parameter at the given longitudinal position. Each of the calculations was set up with data gathering stations corresponding to the transducer locations in the expansion tunnel of the SSGG. The experimentally derived flow characteristics were calculated by the simulations, and comparisons are made between the experiments and the predictions of the two CFD codes.