COMPUTATIONAL AND EXPERIMENTAL STUDY OF SHOCK TUBE EXIT JET FOR NON-IDEAL BLAST SIMULATION

GUIDOS,B.J.; SCHRAML,S.J.; MULLER,P.C.

Nonideal blast effects must be considered in the prediction of vehicle response and survivability to tactical nuclear weapon detonations. A nonideal blast environment of particular interest here is created by the formation of a thermal layer adjacent to the ground. The resulting environment consists of an accelerated flow field that possesses a significantly elevated and sustained dynamic pressure following the initial blast wave. The U.S. Army Research Laboratory (ARL) has contributed to the development of capabilities to simulate and characterize ideal and nonideal blast environment. These computational and experimental capabilities provide the necessary tools to accurately predict blast environments and to protect personnel and equipment.

As part of a nonideal blast program, this paper documents computational fluid dynamics (CFD) and experimental results of the exit jet of ARL's 1.68-m diameter shock tube. The shock tube exit j et can be used to assess vehicle response to blast as well as to provide experimental validation for more complex CFD simulations. In the present study, Navier-Stokes solutions are generated to simulate a shock tube exit j et case from an M 113 armored personnel carrier test in which pressure probe measurements were obtained. The major objectives of the study are to compare the computational results to analytical and experimental results, as well as to assess the computational accuracy and requirements.

The shock tube flow field is modeled using the urified solution algorithm for real gas in two dimensions (USA-RG2) CFD code, developed at the Rockwell International Science Center (RISC), Thousand Oaks, CA. The USA-RG2 code is used here to solve the axisymmetrical form of the Reynolds-averaged Navier-Stokes equations cast in generalized coordinates and strong conservation law form. The multiple zone capability of the code is used. Details of the technique and its application to the shock tube exit j et problem are discussed in the full paper.

The computations were performed remotely on a Cray Y-MP computer located at the U.S. Army Corps of Engineers Waterways Experimental Station, Vicksburg, Mississippi, using a single processor. CFD solutions were generated using two different grids which differ by their respective number of grid points and spatial resolutions. The initial grid solutions used approximately 100 central processing unit (CPU) hours and 2.3 million words of memory each, while the refined grid solution used approximately 200 CPU hours and 3.6 million words of memory.

Riemann solutions and boundary layer analytical solutions are used to validate the CFD solutions before shock exit. Experimental data from in-tube and out-of-tube probes are used to validate the CFD solutions for the duration of the event. An examination is made of the jet flow field structure, and comparisons are made with other empirical data. Specific flow field structures of interest include the brief appearance of a ring vortex, the chain-like structure of expansions and recompressions. and the primary Mach disc. The study demonstrates that the CFD model can accurately simulate important aspects of the dynamic flow associated with the shock tube exit jet. The details will contribute to the formulation of CFD simulations that model target vehicle response to nonideal blast environments.