MISSILE DIVERT SYSTEM OPERATING PHENOMENONOLOGY

K. D. Kennedy*, C. D. Mikkelsen, and B.J. Walker
US RDE Command
Redstone Arsenal, Alabama 35898

1. INTRODUCTION

The Atmospheric Interceptor Technologies (AIT) missile airframe uses a solid propellant Divert and Attitude Control System (DACS) to maneuver the interceptor and close with the target. Precise knowledge of the aerodynamic forces and moments is required to insure hit-to-kill (HTK) aim point accuracies required for successful operation of the interceptor. The accuracy of missile interceptors is critically dependent on the fidelity of the DACS system. The fidelity of the DACS is dependent on the accuracy of the force amplification factor due to the flow separation caused by the divert jet interacting with the free stream flow. Predictions of the behavior of a DACS system at medium altitude (45-50 km) endo-atmospheric operating conditions has shown a very large separated region flow field upstream of the divert jet, a massive separated extent of the flow field around the missile, and very large transient changes in the behavior of this separated region. This behavior has been corroborated with tests of full-scale missile hardware at duplicated flight conditions in Aero-Thermal/Aero-Optics Evaluation Center (AAEC) tests. Surprisingly, lower altitude operations of the DACS (30-35km altitude) has led to the conclusion that jet induced flow separation is an inherently unsteady event at both altitudes, resulting in a variable amplification factor. Comparisons between experiments run in a shock tunnel at duplicated run conditions using a real solid propellant thruster as well as computational fluid dynamic predictions of a powered missile in flight with plume induced flow separation support the conclusions reached above.

2. TECHNICAL CHALLENGE

The AIT flight environment is technically challenging for computational fluid dynamics models because of the complex nature of the external flowfield and particularly so during DACS operation.

Even without jet interaction effects, adequate modeling of the external hypersonic flowfield is difficult because the high flight Mach number produces large gradients of pressure and density and a shock structure close to the body interacting with the vehicle boundary layer. This shock/boundary layer interaction is of sufficient energy to produce aerothermochemical reactions in which the air becomes a reacting fluid. This external vehicle flowfield, when reacting to the presence of the divert jet, also experiences strong compressibility effects that are difficult to model. Furthermore, the boundary layer interacts with the divert jet to produce a large-scale flow separation event in front of the jet. This separation, depending on the geometry of the vehicle and the transition of the boundary layer, may completely separate over the window.

3. AIT CONFIGURATION

The AIT wind tunnel model is composed of a fore cone followed by a conical frustum, a cylindrical section, and a flow stabilizing afterbody flare. The fore cone is a blunted tetracone with four evenly spaced elliptical shaped flats, one of which contains the sensor window. There are no vanes, fins, or other external control surfaces on this configuration save for the divert jet thruster located in the cylindrical section at the missile center of gravity. The AIT vehicle installed in the AAEC facility was full scale. The use of a full-scale model means that no scaling of the measured data is needed, a must for this experimental work, since the scaling of a chemical reactive divert jet is nonlinear.

Flow from the divert jet nozzle was generated by a solid propellant rocket motor. This rocket motor was custom designed for the tests to provide thrust with minimal ignition rise time using an existing AP/HTPB propellant producing no solid particulates in the exhaust. The shock tunnel flow condition was set to a Mach number of 8.5 and the equivalent altitude by pressure of 31 km (102 kft).

A series of static pressure ports were placed at various locations on the body within the symmetrical quarter plane adjacent to the divert jet nozzle. Pressure ports were also located around the sensor window (forebody). The pressure ports along the centerline bisecting the divert jet were capable of a 20 kHz response. The pressure ports off the centerline were adequate for measurement of a 4 kHz response. The 20 kHz static pressure gauges were added to resolve perceived surface pressure fluctuations.

4. COMPUTATIONAL CONDITIONS

AMRDEC has developed a CFD code which models the full Navier-Stokes (FNS) equation set to provide an aerothermochemical plume / missile / airframe, unsteady-flow, predictive capability. The code numerics include a 1D/2D/AXI/3D finite-volume discretization with an

Report Documentation Page

Form Approved
OMB No. 0704-0188

1. REPORT DATE **00 DEC 2004**	2. REPORT TYPE **N/A**	3. DATES COVERED **-**

4. TITLE AND SUBTITLE **Missile Divert System Operating Phenomenonology**	5a. CONTRACT NUMBER
	5b. GRANT NUMBER
	5c. PROGRAM ELEMENT NUMBER
6. AUTHOR(S)	5d. PROJECT NUMBER
	5e. TASK NUMBER
	5f. WORK UNIT NUMBER
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) **US RDE Command Redstone Arsenal, Alabama 35898**	8. PERFORMING ORGANIZATION REPORT NUMBER
9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)	10. SPONSOR/MONITOR’S ACRONYM(S)
	11. SPONSOR/MONITOR’S REPORT NUMBER(S)

12. DISTRIBUTION/AVAILABILITY STATEMENT
Approved for public release, distribution unlimited

13. SUPPLEMENTARY NOTES
See also ADM001736, Proceedings for the Army Science Conference (24th) Held on 29 November - 2 December 2005 in Orlando, Florida.

14. ABSTRACT

15. SUBJECT TERMS

16. SECURITY CLASSIFICATION OF:			17. LIMITATION OF ABSTRACT	18. NUMBER OF PAGES	19a. NAME OF RESPONSIBLE PERSON
a. REPORT **unclassified**	b. ABSTRACT **unclassified**	c. THIS PAGE **unclassified**	**UU**	**2**	

implicit, higher-order upwind (Roe/TVD) formulation, which is highly accurate for convective wave tracking. The thermochemistry includes real gas mixtures (calorically and thermally imperfect) and finite rate chemistry with an arbitrary number of species and reactions. The turbulence model includes a K-ε formulation with compressibility/vortical upgrades and several low R_e near wall formulations. Both the chemistry and turbulent equations are fully implicit. Boundary conditions are applied along the outer computational boundaries and embedded surfaces.

Computational boundary conditions for this high speed jet interaction problem were specified supersonic freestream conditions at the inflow boundary with gradient extrapolation procedures used at the outflow boundary. Surface wall conditions were viscous no-slip flow with imposed isothermal temperature (296.2 K). Divert jet nozzle exit plane conditions were held fixed with a variable supersonic flow profile, which was generated as a separate problem. The flowfield was treated as laminar flow from the nose tip to the end of the sensor window. After the window the flowfield was numerically tripped to turbulent flow (no transition model was used). The entire flowfield was modeled as a fully reacting chemical system using a one step, three species model (fuel, oxidizer, product). All species were tracked separately and had variable thermodynamic properties.

5. RESULTS AND DISCUSSION

Calculations were made at a Mach number of 8.5 and 31 km altitude and were fully time-dependent. The boundary layer was tripped at a fixed location. A total run time of 13.5 ms was compared to the CFD predictions. Pressure ports were located on the centerline of the rear section of the double cone and located on the centerline of the main body upstream of the divert jet. Pressure measurements as a function of time for several centerline pressure ports are shown in Figure 1.

The shock tunnel is started in near vacuum at time zero. As the tunnel is started and the flow of air encases the model, the nearest pressure port to the nose shows a rise in pressure first, followed by the remaining centerline pressure ports respectively. The last two pressure ports (closer to the divert jet) show a larger delay in time before the pressure begins to rise. This rise in pressure correlates to the rise in combustor pressure. This implies that the freestream flow had not fully encased the body before the pressure effects of the combustor were felt. Several ports were influenced by the divert jet periodically during the experiment. Each time the pressure rose and fell back down to a local minimum. This indicates that the divert jet plume induced separation bubble passed over the pressure ports several times during the run. The last two pressure ports closest to the divert jet were in the separation bubble at all times during the run and are strongly influenced by the divert jet.

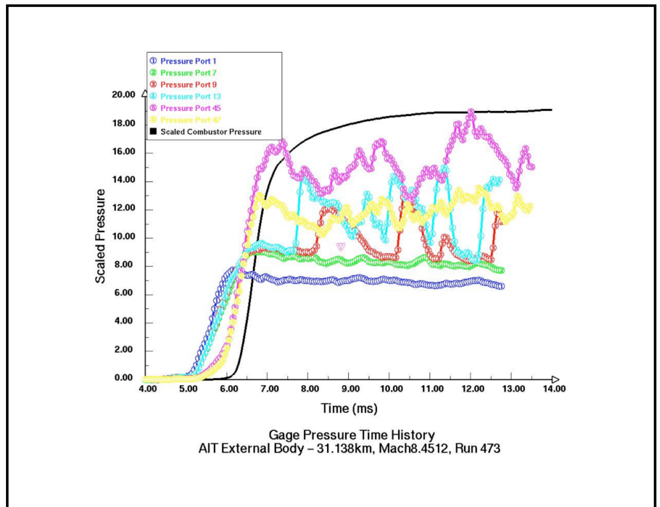

Fig. 1 Centerline Pressure Measurements

CONCLUSIONS

The predicted/measured separation distance location of the divert jet differed by 8%. Separation was predicted larger than shown in the experiment. The model predictions all show a higher frequency oscillation than shown in the measured data. The model predicted surface recovery pressures (pressure before the influence of divert jet) for all ports are higher than measured data. This higher pressure is in the range of 17% compared to the measured pressures. This higher calculated pressure may be a result of turbulence/chemistry interactions.

It was concluded that future analysis of this problem should model the full 360-degree flowfield with no symmetry planes. The separation region most likely moves side-to-side as it advances forward. This would supply some relief to the pressure within the separation bubble and result in a smaller bubble. Additionally, the divert jet nozzle flowfield should be directly coupled with the external flowfield. Lastly, the problem should be run with a fully coupled, multi-species, finite rate chemistry model.