05/01/98 FRI 14:49 FAX 3123532094 ONR-CHICAGO 002

A University • Industry • Government Consortium

OAI OHIO AEROSPACE INSTITUTE

22800 Cedar Point Road • Cleveland, OH 44142
Phone: (216) 962-3000 • Fax: (216) 962-3120

7N-34-CR P2
114850

NASA/CR- -97- 207809

December 31, 1997

Ms. Saundra Gage
Grants Officer
NASA Lewis Research Center
21000 Brookpark Road
Mail Stop 500-315
Cleveland, Ohio 44135

Reference: NCC3-550 — "Application of Chimera Navier-Stokes Code for High Speed Flows"

Dear Ms. Gage:

Please find enclosed the final Technical Report for the NASA cooperative agreement referenced above. Please contact me at (440) 962-3010 if you have questions on the enclosed.

Sincerely,

Eileen Pickett
Vice President, Operations

Enclosure

cc: K. Balog - OAI
C. DiCicco - ONR (cover only)
M. Liou - NASA

FINAL TECH
NCC3-550

"Application of Chimera Navier-Stokes Code for High Speed Flows"

Kumud Ajmani

The primary task for this year was performed in support of the "Trailblazer" project at NASA Lewis (under the auspices of the Lewis Hypersonics Program). The purpose of the task was to perform an extensive CFD study of the shock boundary-layer interaction between the engine-diverters and the primary body surfaces of the Trailblazer vehicle. Information gathered from this study would be used to determine the effectiveness of the diverters in preventing the boundary-layer coming off of the vehicle forebody from entering the main engines.

The first step in the CFD analysis involved the generation of individual, overlapping, three-dimensional grids around the surfaces of interest. The surface definitions provided by the Trailblazer team were processed through the ICEM-CFD program, in order to generate surface definitions which would be acceptable to the grid-generation code(s). Two grid-generation programs - GRIDGEN (for H-grids) and HYPGEN (for C-grids around conical surfaces) were used to generate a total of 6 volume-grids for the vehicle forebody, engine-pods and diverter surfaces.

The grid-generation stage was followed by the grid "assimilation" stage. The PEGSUS code was used to define the "holes" and "boundaries" for each grid. The information provided by the PEGSUS code was used as an input into the CHIMERA-based, grid-overset, upwind, finite-volume CFD code to perform the CFD simulations. The code uses the Baldwin-Lomax turbulence model in order to model turbulent regions of the flow.

Two sets of CFD calculations corresponding to Mach 2.5 and Mach 6.0 were performed. The angle-of-attack (AOA) was varied between -3, 0, and +3 degrees (for the purpose of a parametric study) for the Mach 2.5 case ; the Mach 6.0 case was studied at 0 degrees AOA. A grid-refinement study was performed for the Mach 2.5, AOA = 3 degrees, and the grid densities for the grid-converged solution were then maintained in all the other cases. This was done to ensure that all the CFD results obtained would be grid-independent.

Extensive post-processing of the results (particle-traces, contour plots) was performed for each test case. The results revealed the existence of a glancing side-wall boundary-layer interaction between the shock wave off of the engine-cones and the "diverted" boundary-layer between the pods. The CFD results indicated that the turning of the boundary-layer due to the pressure exerted by the engine-cone shock(s) might be sufficient enough that the "physical" diverters might be eliminated from the configuration, without any loss in vehicle performance. However, this phenomenon needs to be studied further, particularly since the elimination of the "physical" diverters would considerably simplify the overall design of the Trailblazer vehicle. Detailed CFD results of the computations performed in this task will be made available in an ICOMP report (to be published).

This work was done in collaboration and with the assistance of NASA colleagues Meng-S. Liou, Charles J. Trefny, Joe Roche, John Slater and Kai-H. Kao.