https://www.hpc.mil/images/hpcdocs/newsroom/ugc_2011_proceedings_small.pdf

Computational Analysis for Air/Ship Integration: 2nd Year Report

Susan A. Polsky
US Naval Air Warfare Center, Aircraft Division (NAWCAD), Patuxent River, MD
susan.polsky@navy.mil

Abstract

This paper documents the accomplishments from the second year of a three-year Grand Challenge Project focusing on the application of computational fluid dynamics to predict coupled ship and aircraft aerodynamics. Unstructured chimera techniques were used to simulate the coupled ship and aircraft systems. Dynamic aircraft maneuvers were prescribed with the intention of building simulations with an auto-pilot-in-the-loop. All simulations were computed in a time-accurate fashion due to the unsteady nature of the flowfield, and used the commercial flow-solver Cobalt. Analyses for both vertical shipboard landings of the Joint Strike Fighter (JSF) and rotary-wing aircraft are discussed. Internal components of the JSF lift-fan were added to the model to increase the solution fidelity.

1. Introduction

While many aspects must be taken into consideration to ensure safe shipboard flight operations, a primary factor is evaluation of turbulent air-wake effects on aircraft performance and pilot workload. The air-wake is a product of wind passing over ship structures creating non-uniform, turbulent air flow. The US Navy conducts shipboard dynamic interface (DI) testing to evaluate ship air-wake effects on aircraft operations. These tests result in wind-over-deck (WOD) flight envelopes that prescribe in what wind conditions an aircraft can or cannot fly. The WOD flight envelopes are part of the operating procedures for all ship-based aircraft. Testing is required to generate WOD envelopes for each model of air vehicle operating from a given ship. The DI tests are performed at-sea, typically over the course of several weeks.

Application of computational fluid dynamics (CFD) methods to predict the turbulent ship air-wake has been studied in the past with considerable success[1–4]. This has resulted in the use of CFD as an analysis tool to "diagnose" air-wake structures that may impact air operations for both current and future ship designs. These diagnoses are accomplished by linking stored CFD-generated air-wake data with offline aircraft models, controlled by either a pilot model or some other autonomous controller. For this "one-way coupled" approach, the air-wake data is imposed on the aircraft model; however, the presence of the aircraft does not feed back into the air-wake data. While this approach has proven very useful, there are limitations to its applicability. When employing CFD data generated from a ship in isolation, the underlying assumption is that the presence of the aircraft will not affect the air-wake from the ship structures. In the case of a small uninhabited air vehicle (UAV), this is likely a valid assumption; however, for an aircraft that produces a large wake of its own (such as a helicopter), this assumption becomes less-and-less valid as the aircraft comes in closer proximity to ship structures. The interaction of the ship air-wake and the aircraft wake is generally referred to as ship/aircraft coupling. Aerodynamic coupling is a concern for both fixed-wing and rotary-wing shipboard operations.

As mentioned above, past research developed methods to accurately predict ship air-wake and laid the groundwork for prediction of coupled ship & aircraft predictions. Research executed in the 2006–2008 time-frame demonstrated the feasibility of modeling both stationary aircraft and aircraft with prescribed-motion immersed in ship air-wake. The aircraft types examined included fixed-wing (F-18) and rotary-wing (V-22, H-60). The present work builds upon past research in coupled ship/aircraft modeling through support from the Office of Naval Research "Coupled Aircraft Ship Simulation for Improved Acquisition" (CASSIA) program and the Joint Strike Fighter (JSF) program.

The goal of the CASSIA program is to understand the physical and numerical modeling deficiencies that prevent the application of current dynamic interface simulations for flight-envelope prediction. Analysis of an H-60 helicopter with a DDG (destroyer)-class ship was continued in the 2nd year (Figure 1). The motivation for the DDG/H-60 analysis is to understand where (in regards to proximity to a ship) aerodynamic coupling becomes important for rotary-wing vehicles. This knowledge, along with the coupled DDG/H-60 CFD data will be used to develop methods to account for aerodynamic coupling suitable for man-in-the-loop simulations (i.e., methods that run in real-time).

Figure 1. An H-60 on approach to DDG-Class ship

Analyses of the short takeoff vertical landing (STOVL) JSF on L-class US Navy ships (Figure 2) were also conducted. The JSF CFD analysis was required to prepare the ship for JSF testing. In particular, analyses were required to examine whether JSF outwash during vertical landings (VL) would damage critical (and costly) ship systems, either due to exposure to hot jet exhaust or due to the force of the jet exhaust. During an approach to a landing spot, the aircraft core nozzle passes near many large hull-mounted systems, such as radars and weaponry, and passes directly over the catwalk and flight deck. Although sub-scale experimental studies of the STOVL outwash field were conducted, these experiments were for zero ambient wind-speed with exhaust impingement on flat plates. It was recognized that the air-flow patterns would likely be affected by surrounding ship structures and prevailing wind-speed and direction. Therefore, the CFD analyses included the ship hull, island superstructure, and many of the larger ship systems in the area of concern (Figure 2). The CFD analyses were used to help determine whether steps should be taken to move or shield deck-edge equipment and replace it with instrumentation to gather data during flight-test at-sea. In the 2nd year of this project, short takeoff (STO) scenarios were also examined. In addition, the fidelity of the JSF model was significantly increased to include most of the internal lift-fan structure.

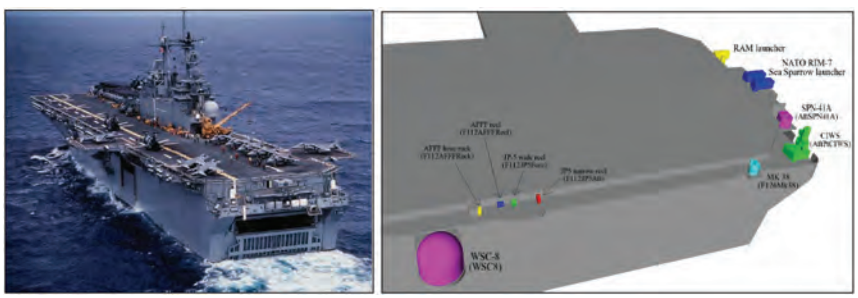

Figure 2. Left: A typical L-Class ship; Right: Stern- & port-mounted ship systems modeled in initial CFD model

2. Technical Approach

The commercial CFD solver Cobalt, which was initially developed by the US Air Force under a CHSSI project, was used for this work. The solver employs a finite-volume, cell-centered scheme with flux vector-splitting and unstructured, overset-grid technology for bodies in relative motion[5]. All cases were run with second-order spatial and temporal accuracy. Past work has established that employing a monotone large-eddy simulation (MILES) turbulence modeling approach is well-suited for the prediction of the large-scale turbulence found in ship air-wake flowfields. This approach was applied for the DDG/H-60 analysis discussed in this work. However, the MILES approach is not well-suited for JSF analysis due to the fine-scale turbulence which dominates the jet-flow. Previous to this work, a validation study was completed comparing core-nozzle-exit-flow to sub-scale experimental temperature and pressure measurements at various distances below the nozzle-exit. The results of this study indicated that applying a shear stress transport (SST) model was required to capture the core-nozzle exhaust-flow with required accuracy. This analysis approach was extended to prediction of the lift-fan-flow in the 2nd year and was found to accurately predict the thrust for the lift-fan. For coupled ship cases with no wind, using an SST model is not an issue. However, application of an SST model to aircraft/ship analyses with ambient wind presents a dilemma as it has been established that application of turbulence models to ship air-wake flows damps-out much of the higher-frequency turbulence[2]. While it is thought that this higher-frequency turbulence will not significantly affect the JSF outwash; more analysis is needed to confirm this.

The gridding and turbulence modeling approaches established by the core-nozzle-validation study were applied when modeling the complete JSF configuration including the core-nozzle, lift-fan and roll-nozzles. In the first year of this project, the lift-fan and roll-jets were modeled using user-prescribed boundary-conditions at the nozzle-exits. The boundary-conditions were derived from high-fidelity CFD solutions of the isolated nozzles provided by Lockheed Martin. Verification of the multiple nozzle-flow interaction against sub-scale experimental data using this configuration is complete. While the comparisons demonstrated that the CFD was in the "ballpark" of the sub-scale data, unsteady characteristics in the CFD analysis pointed to issues with application of the user-prescribed boundary-conditions for the lift-fan in particular. The internal geometry of the lift-fan was recently modeled using Cobalt and gave very good results for predicted thrust. As such, the lift-fan geometry has been added to the full-JSF model and test cases are currently underway.

Verification against full-scale shipboard data will take place in the 4th quarter of CY 2011. Shipboard and land-based testing will provide a wealth of verification data for the modeling approach applied here. Verification against full-scale data will be the major thrust of the 3rd year of this project.

Both the H-60 and JSF calculations employed overset grids. All grids were generated using the NASA GridTool/VGRID system. Grid sizes for the DDG/H-60 analysis were on the order of 11.4 million and 3 million cells, respectively. The rotors (main and tail) were modeled as actuator disks, and in some cases as blade elements. In the first year, grid sizes for the LHD/JSF outwash analysis were on the order of 29 million and 3 million cells, respectively. With the addition of the lift-fan internal geometry, the grid sizes for the JSF have grown to over 15 million cells. Because aircraft performance predictions were not required, a relatively simple fuselage model was used for the JSF aircraft. This allowed for construction of much smaller grids than would be required if the detailed surface-geometry was modeled. The JSF analysis modeled motion of the aircraft relative to the ship, while the H-60 analysis employed overset grids to integrate individual blade-segment aerodynamic information generated by an external blade-element model.

3. Progress to Date

3.1 LHD/JSF

As in the first year, all dynamic approach simulations were conducted with prescribed approach paths and included a 90° approach (shown in red in Figure 3) and a 45° approach and a short takeoff. Two landing spots were examined: spot 7 (forward spot shown in Figure 3) and spot 9. In the first year of the project, a generic LHD ship was used (Figure 3). However, in the 2nd-year, a ship surface model specific to LHD-1 was created and considerable detail added to the port-side deck-edge to gather data from STO analysis (Figure 4). A typical STO consists of three distinct phases: deck-roll, rotation, and fly-away. Currently, only the deck-roll portion is modeled because the core-nozzle and lift-fan conditions and orientations stay fairly constant. This is in contrast to both the rotation and fly-away portions where significant movement of both nozzles occurs. Modeling movements of the nozzles would require significant increase in complexity of the CFD model and is not currently planned.

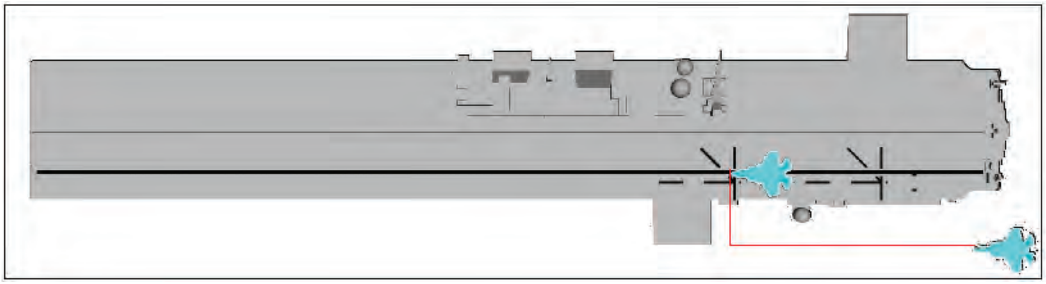

Figure 3. Example of a vertical landing approach path (red) on generic LHD ship model

The length of the deck-roll is illustrated in Figures 4 and 5. The motion is again prescribed and models the typical aircraft accelerations and speeds. To date, only zero atmospheric wind conditions have been modeled. Non-zero wind cases will be completed later this year. A snapshot from a zero-wind STO case is shown in Figure 6. The jet exhaust is visualized by surfaces of constant velocity.

Figure 4. Extent of deck-roll for short takeoff (dashed red line)

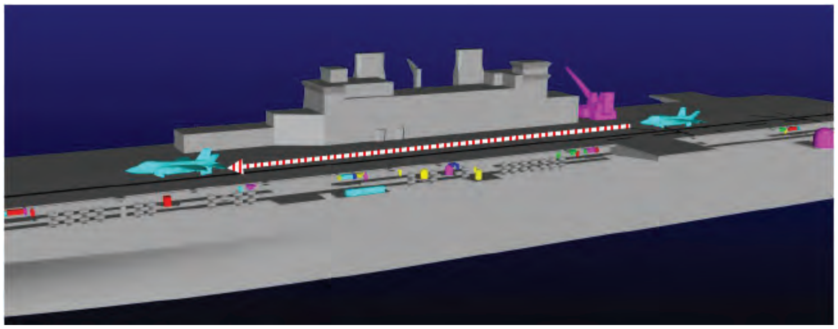

Figure 5. Perspective view of detailed deck-edge equipment modeled for STO analysis

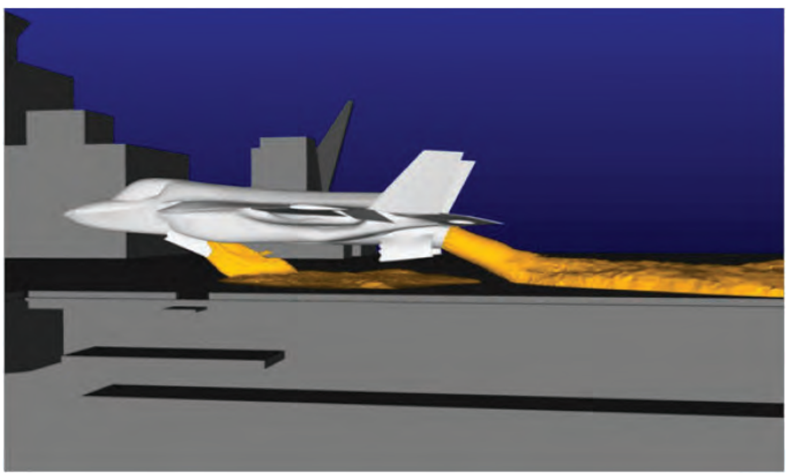

Figure 6. A snapshot-in-time of short take-off simulation. Jet exhaust and outwash visualized by velocity iso-surface.

JSF simulations were run on AFRL Hawk, MHPCC Mana, and ARL Harold. Calculations typically utilized 256 processors and required approximately 30,000–50,000 CPU-hours each, depending on the grid-fidelity and length of dynamic maneuver. Cases incorporating the internal geometry of the lift-fan are currently running. CPU requirements from these preliminary cases will be reported in the oral presentation.

3.2 DDG/H-60

In the first year, analysis of the coupled ship/helicopter flowfields used actuator disk models for the main- and tail-rotors of an H-60 helicopter. A total of 90 cases were completed with the helicopter in different hover positions around a DDG destroyer. This data is being used to create a correction methodology to incorporate first-order effects of ship/aircraft coupling into real-time simulations by modifying the ship-alone CFD air-wake datasets. Several approaches in this regard are under examination including the application of proper orthogonal decomposition (POD) analysis in conjunction with neural network algorithms.

In the 2nd year, emphasis has transitioned from actuator-disk models to incorporation of blade-element models. While Cobalt had a previously existing blade-element modeling capability, the code is being tailored to support interfacing with a blade-element model running as a separate executable outside of the Cobalt code. The motivation for this work is to enable aircraft real-time simulation models used by the Manned Flight Simulator (MFS) at Patuxent River Naval Air Station to be "plugged" into Cobalt for coupled ship/aircraft CFD analysis. This allows models such as engine performance, landing gear response, actuator response, etc., to be incorporated into the analysis while the CFD code is used to predict the high-fidelity aerodynamics. A block diagram of the data passing from Cobalt (red box) to the external aircraft model (green box) is depicted in Figure 7. The blade-element model being linked with Cobalt is a generic "experimental" helicopter model called ExHel developed by the Air Vehicle Modeling and Simulation Branch at NAVAIR Patuxent River. The ExHel model was designed to run in a real-time, man-in-the-loop simulation.

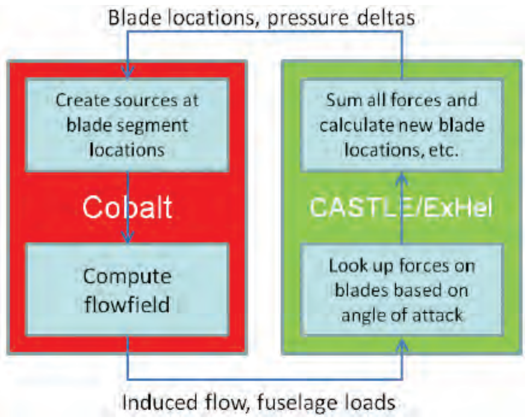

Figure 7. Block diagram showing Cobalt and CASTLE data paths

All aircraft simulation models at the MFS run within an operating system called CASTLE. As part of this project, CASTLE/ExHel were ported to an HPC system and successfully run in batch-mode. At the MFS, CASTLE communicates with outside models (such as CFD ship air-wake databases) through TCP/IP calls. However, due to security issues, using TCP/IP is not allowed on HPC systems. Therefore, data exchanges between Cobalt and CASTLE/ExHel will be accomplished using a file-in-file-out (FIFO) approach such that Cobalt will write a text file for CASTLE/ExHel input and vice-versa. Dependency on text files for data-transfer will, unfortunately, significantly hinder the execution and scalability performance of the analysis. It is hoped that a method to more-efficiently accomplish the data-transfer will be found in the future.

The ExHel blade-element model provides blade positions (including flap-angle and lead-lag positions) along with the aerodynamic variables for each blade-segment on each blade. Since the blade aerodynamic properties are provided by ExHel, it is not necessary to create body-fitted grids to model the blades within Cobalt. This significantly reduces required grid sizes. However, in order to ensure adequate grid quality always exists at the blade locations, it was decided to use overset (non-body-fitted) grids for each blade. With this approach, the background ship grid can be made fairly generic, avoiding the need to tailor a ship grid to support a particular aircraft maneuver. Examples of the blade-element grids are shown in Figure 8 along with the blue background grid. The blade locations provided by ExHel are shown as solid gray. The overset grids used for each of the four blades are shown in color (yellow, green cyan, red). Within each overset grid, cells on the blade segments do not change, ensuring consistent model-fidelity throughout the analysis.

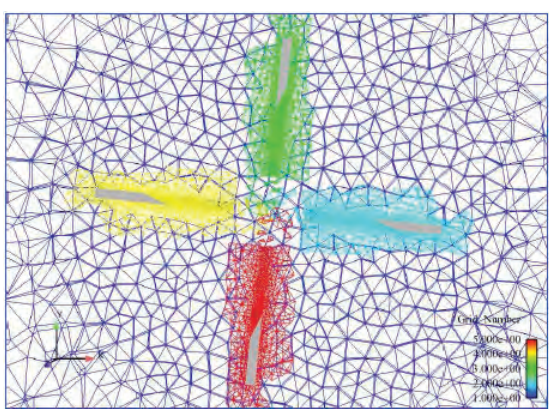

Figure 8. Blade-element overset grids (red, cyan, green, yellow) with blue background grid

Before tackling, getting Cobalt and CASTLE/ExHel to run in concert, a set of test-data was generated using ExHel running in its native environment and provided to the Cobalt developers as a text file. In this scenario, Cobalt does not talk back to ExHel, but simply reads a set of time history blade position and pressure-data and applies that data within the overset blade-grids. The intricacies associated with multiple coordinate transformations and interpolations make this step, which is still on-going, an essential part of the model development. Results from one of these preliminary calculations are shown in Figures 9 and 10. These figures depict iso-surface of vorticity colored by pressure at a very early time-step (Figure 9) and at a time-step ~5 seconds later (Figure 10) showing how the flowfield develops. Lessons-learned from this preliminary analysis are being used to better-define the data needed from ExHel and to establish a verification method for the CFD predictions.

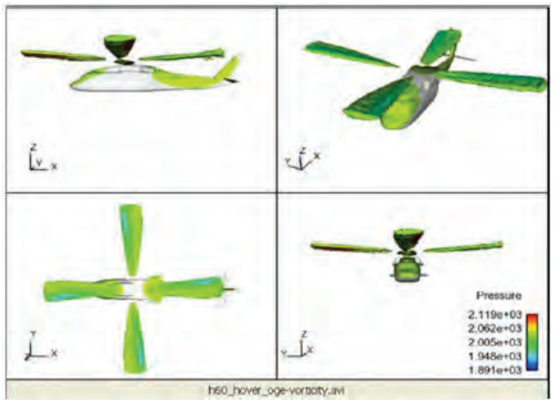

Figure 9. Iso-surface of vorticiy at an early time-step of Cobalt analysis using ExHel blade-element data

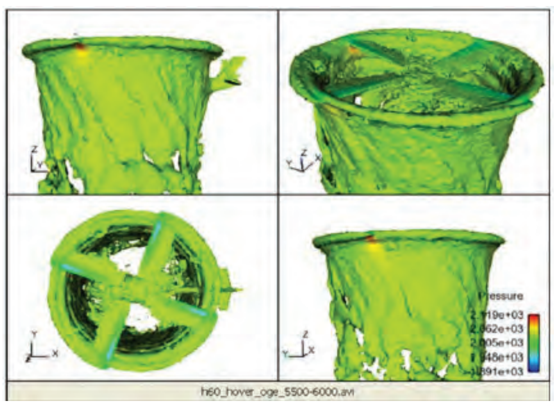

Figure10. Iso-surface of vorticiy at a later time-step of Cobalt analysis using ExHel blade-element data

Although the generic ExHel model is being used as the demonstration case for Cobalt/CASTLE integration, the ultimate goal is to use the integrated approach for a real air vehicle. A major difficulty in linking an aircraft like an H-60 in a non-real-time simulation is the need for a pilot to fly the simulation model. This requirement also exists for ExHel. While a pilot model (auto-pilot) has been developed to “fly” ExHel during a Cobalt run, it will never quite represent how an actual pilot would fly an aircraft. In the near-term, the pilot model will be programmed to maintain a stationary hover within a ship air-wake. Upon successful completion of hover analyses, the pilot model will be enhanced to include approach to the ship. A hover pilot model is complete and runs in its native CASTLE environment linked to offline CFD ship air-wake databases for DDG. The pilot model will be ported to an HPC system in the future.

The Fire Scout VTUAV provides a unique opportunity because the same autonomous control system that flies the aircraft can be integrated in the simulation, obviating the need for a pilot model. Because a pilot is not required in-the-loop, the simulation need not run in real-time. Development of a Fire Scout simulation model tailored to link with Cobalt is underway. A Fire Scout CFD fuselage surface model has been created, and initial calculation of fuselage drag has been calculated and compared to available wind-tunnel data. It is hoped that before the end of the 3rd year of this project, a demonstration case of Cobalt linked with a Fire Scout simulation model will be completed.

4. Significance to the DoD

Rotary-wing ship integration modeling and simulation research funded through the Office of Naval Research CASSIA program is paving the future in dynamic interface modeling and simulation. Results from this work will directly contribute to improving ship design, ship-based aircraft control system design, and flight-control design for piloted and autonomous air vehicles. Numerical algorithms and physical models developed through this program will be transitioned to the CREATE program to benefit the CFD community throughout the DoD.

Techniques and methodologies developed through this project will be directly applicable to the JSF program during developmental testing (DT), operational testing (OT), and fleet support to help ensure safe operation of the aircraft aboard ship. Successful application will require the development, validation, and application of CFD techniques related to coupled aircraft and ship air-wake prediction. Inherent in the development and validation of this technology are important science and technology areas such as vortex modeling, large-eddy simulation, grid-generation, parallel-code development, CFD boundary-condition development and temporal-accuracy issues. These developments will also benefit other DoD aircraft and ship programs, such as V-22, VTUAV, CVN-78, and DDG-1000.

The military advantages gained by exploiting HPC capability are related to design and testing. This analysis will enable ship integration of the JSF system in such a way that minimizes potential damage to ship structure and systems, and helps to ensure safety of personnel.

Acknowledgements

The author would like to thank: 1) the Office of Naval Research and the Joint Strike Fighter Program for their financial and professional support, 2) the High Performance Computing Modernization Program for their generous allocation of computer time, without which this work would not have been possible, and 3) Cobalt Solutions LLC for Figures 7–10.

References

1. Polsky, S. and Bruner, C.W., "Time-Accurate Computational Simulations of an LHA Ship Air-wake." *AIAA Paper 2000-4126*, 2000.

2. Polsky, S., "Computational Study of Unsteady Ship Wake." *AIAA Paper 2002-1002*, January 2002.

3. Polsky, S., "CFD Prediction of Air-wake Flowfields for Ship Experiencing Beam Winds." AIAA Paper 2003-3657, August 2003.

4. Woodson, S. and Ghee, T., "A Computational and Experimental Determination of the Air Flow Around the Landing Deck of a U.S. Navy Destroyer (DDG)", *AIAA Paper AIAA-2005-4958*, June 2005.

5. Strang, W.Z., Tomaro, R.F., and Grismer, M.J., "The Defining Methods of Cobalt60: A Parallel, Implicit, Unstructured Euler/Navier-Stokes Flow Solver", *AIAA-99-0786*, January 1999.