CFD MODELING OF NASA'S ARES PLATFORM. J. J. Coltrane[1,3] and A. S. Arena[2,3]. [1]Dept. of Mechanical and Aerospace Engineering, North Carolina State University, Raleigh, NC 27695, (jjcoltr2@ncsu.edu); [2]Dept. of Mechanical and Aerospace Engineering, Oklahoma State University, Stillwater, OK 74078; [3]Arkansas-Oklahoma Center for Space and Planetary Sciences, Oklahoma State University, Stillwater, OK 74078.

Introduction: Computational Fluid Dynamics has evolved into a powerful tool for aerodynamicists with the continuous increase in processor speed. Intensive computations are now being completed on simple desktop computers, allowing meaningful research to be done by students new to the field of computational fluid dynamics.

This project's aim is to model NASA's platform for aerial exploration of Mars, A.R.E.S., or Aerial Regional-scale Environmental Survey. This Mars Scout Mission candidate did not initially win the bidding for the Scout Mission to fly in 2007, losing to the lander Phoenix. It is still in competition however for the next Mars Scout Mission and many of the specifics on the plane design are competition sensitive and cannot be discussed openly. This constraint leads to difficulty in modeling the plane, but a suitable model has been constructed using the FELISA grid generator.

FELISA (Finite Element Langley Imperial Swansea Ames) is a grid generator developed at the previously listed institutions and is part of the STARS package that OSU uses to model Aeroelasticity for NASA Dryden. Three input files must be constructed for the FELISA grid generator to run, these are the surfaces, boundary conditions and background files. FELISA's grid output from these three files and a controls input file allow the fluid solver Euler3d to simulate a variety of atmospheres, free stream speeds, and directions of air flow over an aircraft.

Figure 1. Orthogonal view of ARES Platform from NASA Langley's Proposed Mars Scout Mission website, http://marsairplane.larc.nasa.gov/multimedia.html

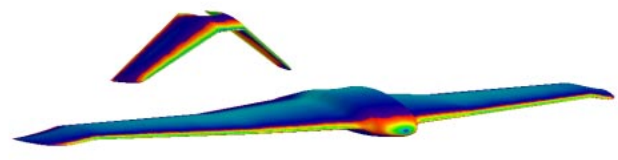

Figure 2. Colorbars view of different Mach speeds over ARES. Image generated with Glplot3d, part of the STARS package.

Methodology: The most difficult part of the CFD process is the programming of the plane's surfaces to be analyzed. This modeling is done through the surfaces file and is a list of x, y and z-coordinates that comprise 74 segments and 36 surfaces. Each surface is identified by the segments that comprise it in a "right hand rule" fashion. The surface is broken into parallel segements and each segment is listed from right to left as to orient the surface "out." The surfaces file for ARES is copied from an Excel spreadsheet that is linked to another sheet dictating placement of the 33 airfoils in the wing and 13 airfoils in the stabilizer. Each airfoil exists as a set of 98 x,y-coordinate points from the UIUC airfoil database. The collection of airfoils, given the added dimension of depth in the spreadsheet, make the surfaces file over 7000 lines long.

The boundary conditions file needed by FELISA indicates the sharpness of certain segments, such as the leading and trailing edges. The background file indicates the density of nodes where measurements are taken on and around the plane. Areas of denser measurements produce a finer grid over the plane, but neccessites a longer run time for computing (Figure 3). The last input file, the controls file, provides information to the Euler3d solver as to the free stream Mach speed and the direction of this stream flowing over the plane.

A pullout scenario of the ARES deploying straight down was ran in Euler3d. The computed lift coefficients, along with constants such as ARES's weight on Mars, gave a height requirement for deployment. This scenario also provided a glide distance, if for some reason ARES's liquid rocket propulsion system malfunctioned [1].

Results and Conclusions: The CFD solver Euler3d outputs the total forces on the plane in the x,y and z dimensions and the total moments on the plane in the x, y and z dimensions. These forces and moments along with the variables in the control file can be used to compute coefficients of lift and drag.

Some steady data was taken for ARES using the makecut3d utility. It allows the Mach numbers and coeffiecients of pressure to be taken for each coordinate point along a surface of the plane (Figure 4). A similar utility in the Glplot3d program plots the calculated values along with ARES (Figure 5). The Glplot3d program allows a number of interesting visuals to be constructed that provide greater insight into where problems of turbulance arise.

Computational fluid dynamics is allowing planes like NASA's ARES to be designed for optimum flight in alien atmospheres. Atmospheres one-hundredth the density of our own and consisting nearly entirely of carbon dioxide [2]. These properties combined with strange temperatures in the negative fifties give Mars a low Mach number velocity, 241 m/s [2]. This furthur complicates flight with the intoduction of shocks at slower speeds. The most accurate way to replicate all of these interesting flight dynamics is through CFD.

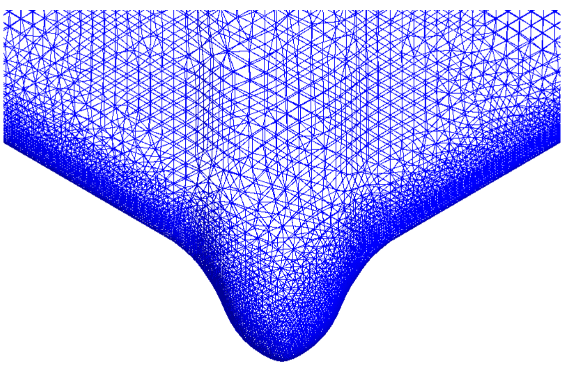

Figure 3. Top view of ARES showing the background file grid and the density of nodes along the leading edge. Image generated using GLplot3d ver. 6.55, part of the STARS package.

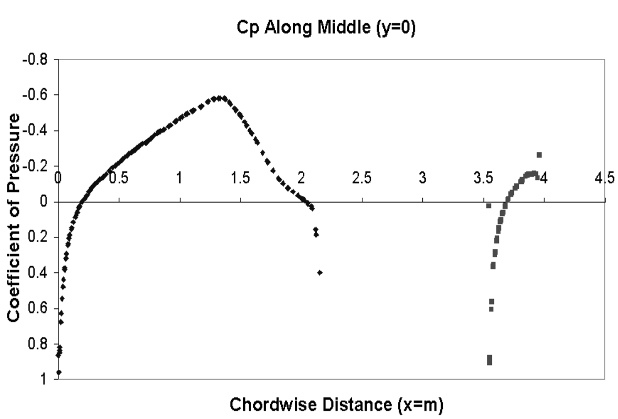

Figure 4. Graph of coeffiecient of pressure along the line of symmetry of ARES.

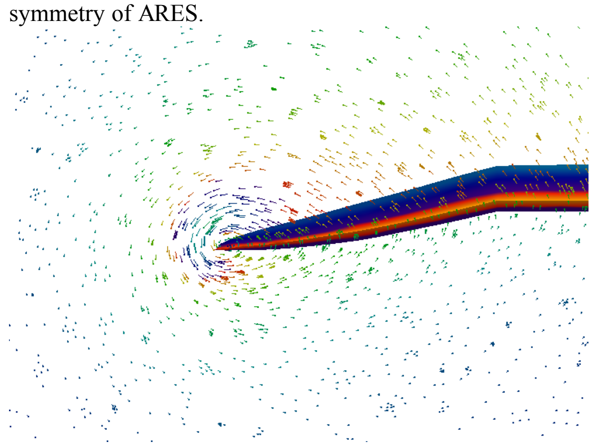

Figure 5. Velocity vectors cut made two centimeters behind trailing edge of wing. At this distance vortices can be seen to be forming.

References: [1] Guynn M. D. et al (2003) *Evolution Of A Mars Airplane Concept For The ARES Mars Scout Mission,* AIAA-2003-6578. [2] Kenney P. S. and Croom, M.A. (2003) *Simulating the ARES Aircraft in the Mars Environment*, AIAA-2003-6579. [3] Abbott I. H. and Von Doenhoff A.E. (1959) *Theory of Wing Sections,* 3-9. [4] Anderson J. D. (2001) *Fundamentals of Aerodynamics.* [5] Cowan T. J. (2003) *Finite Element CFD Analysis.* [6] Morgan K. (1993) *FELISA Ref. Manual.*

Acknowledgments: I thank Charles O'Neill and Nick Moffit for their help in learning the CFD software. I would also like to thank the Arkansas-Oklahoma Center for Space and Planetary Science for hosting the NSF REU program.