Dissertation Research - Computational Fluid Dynamics Uncertainty Analysis for Payload Fairing Spacecraft Environmental Control Systems

by

Curtis E. Groves

Bachelor of Science in Aerospace Engineering, 2008

Bachelor of Science in Mechanical Engineering, 2008

Masters of Science in Aerospace Engineering, 2012

Doctor of Philosophy in Mechanical Engineering, ECD: Spring 2014

A DISSERTATION PROPOSAL

Spring Term

2013

Chair: Marcel Ilie, Ph.D.

Member 1: Tuhin Das, Ph.D.

Member 2: Jeffrey Kauffman, Ph.D.

Outside Member 1: Brian Moore, Ph.D.

ABSTRACT

Spacecraft thermal protection systems are at risk of being damaged due to airflow produced from Environmental Control Systems. There are inherent uncertainties and errors associated with using Computational Fluid Dynamics to predict the airflow field around a spacecraft from the Environmental Control System. This proposal describes an approach to validate the uncertainty in using Computational Fluid Dynamics to predict airflow speeds around an encapsulated spacecraft. The research described here is absolutely cutting edge. Quantifying the uncertainty in analytical predictions is imperative to the success of any simulation-based product. The method could provide an alternative to traditional "validation by test only" mentality. This method could be extended to other disciplines and has potential to provide uncertainty for any numerical simulation, thus lowering the cost of performing these verifications while increasing the confidence in those predictions.

Spacecraft requirements can include a maximum airflow speed to protect delicate instruments during ground processing. Computational Fluid Dynamics can be used to verify these requirements; however, the model must be validated by test data. The proposed research project includes the following three objectives and methods. Objective one is develop, model, and perform a Computational Fluid Dynamics analysis of three (3) generic, non-proprietary, environmental control systems and spacecraft configurations. Several commercially available solvers have the capability to model the turbulent, highly three-

dimensional, incompressible flow regime. The proposed method uses FLUENT and OPENFOAM. Objective two is to perform an uncertainty analysis of the Computational Fluid Dynamics model using the methodology found in "Comprehensive Approach to Verification and Validation of Computational Fluid Dynamics Simulations". This method requires three separate grids and solutions, which quantify the error bars around Computational Fluid Dynamics predictions. The method accounts for all uncertainty terms from both numerical and input variables. Objective three is to compile a table of uncertainty parameters that could be used to estimate the error in a Computational Fluid Dynamics model of the Environmental Control System /spacecraft system.

Previous studies have looked at the uncertainty in a Computational Fluid Dynamics model for a single output variable at a single point, for example the re-attachment length of a backward facing step. To date, the author is the only person to look at the uncertainty in the entire computational domain. For the flow regime being analyzed (turbulent, three-dimensional, incompressible), the error at a single point can propagate into the solution both via flow physics and numerical methods. Calculating the uncertainty in using Computational Fluid Dynamics to accurately predict airflow speeds around encapsulated spacecraft in is imperative to the success of future missions.

Table of Contents

1.0 Introduction:

This proposal will investigate the applicability of the American Society of Mechanical Engineers (ASME) Verification and Validation of Computational Fluid Dynamics (CFD) Simulations applied to spacecraft / fairing Environmental Control Systems (ECS).

Delicate spacecraft instruments will be needed for satellite technology enhancement of agricultural yield, environment sustainability, or telecommunications. Before spacecraft are released into orbit to complete their science goals, the spacecraft must survive the ground and launch environments. ECS systems supply air to keep the spacecraft cool, dry, and clean. Delicate spacecraft instruments are sensitive to high velocity flow from the ECS systems and manufactures set impingement requirements to protect these instruments. CFD is often chosen to complete verifications of the impingement requirements rather than testing. Using CFD to predict the airflow field around a spacecraft enclosed in a fairing has been documented and validated using test data [15, 16].

The problem is there are inherent uncertainties and errors associated with using CFD to predict the airflow field, and there is no standard method for evaluating uncertainty in the CFD community [1]. Some potentials errors include physical approximation error, computer round-off error, iterative convergence error, discretization errors, computer programming errors, and usage errors [4]. An uncertainty, as defined by the American Institute of Aeronautics and Astronautics (AIAA), is a potential deficiency in any phase or activity of modeling and simulation that is due to the lack of knowledge [3]. An example of an uncertainty in performing a CFD analysis is turbulence modeling [4]. There is a lot about turbulence modeling that is not understood [4]. There has been progress in estimating the uncertainty of CFD, but the approaches have not converged [1].

CFD is used primarily for analytical predictions of the velocity, heat transfer coefficient, and pressure. CFD is the current state of the art and industry standard used for spacecraft ECS flow analysis; however CFD has many challenges. The users must select the appropriate models to characterize their specific problem. The proposed research will use different turbulence models as an input uncertainty to help the community evaluate the accuracy of turbulence modeling. There are many other input variables. These include boundary conditions, wall functions, fluid properties, turbulence models, solution schemes, solvers, mesh, and numerical calculations. The current state of the art uncertainty analysis will evaluate each of the error sources and provide the corresponding uncertainty of the velocity around a spacecraft due to the ECS system. No one to date has ever calculated the uncertainty in using CFD to predict the velocity of spacecraft/ECS systems for the entire domain. The benefit to the community will be to prove and document the approach used and provide a table of all uncertainty variables, which can be used to estimate the error in a velocity prediction.

2.0 Biographical Sketch:

The author, Mr. Curtis Groves, is a PhD student at the University of Central Florida. Mr. Groves has worked for NASA at the Kennedy Space Center in the Launch Services Program since 2006 where he performs independent verifications of NASA's science payload requirements. Mr. Groves has performed ECS impingement verifications for the following missions: GLORY, MSL, TDRSS-K/L, and IRIS and external aerodynamics verification on the Atlas V vehicle. Mr. Groves completed dual Bachelor's Degrees in aerospace engineering and mechanical engineering from West Virginia University and graduated Summa Cum Laude. Mr. Groves has graduated from the University of Central Florida with a master's in aerospace engineering in May 2012 and is working to complete a PhD in May 2014. A summary of Mr. Groves' background is provided in Table 1. Mr. Groves has research interests in Computational Fluid Dynamics, Turbulence Modeling, Uncertainty Analysis, External Aerodynamics, Spacecraft Venting, Environmental Control Systems, and Heat Transfer.

The author has recently published a "Comprehensive Approach to Verification and Validation of CFD Simulations Applied to Backward Facing Step – Application of CFD Uncertainty Analysis", AIAA-2013-0258. This document lays out the literature review, state of the art, and the proposed method that will be applied to the spacecraft / fairing ECS problem.

Table 1 – Summary of Authors Prior Work and Background

Curtis E. Groves
(321) 274-2758, Curtis.E.Groves@nasa.gov

Research Interests:

Computational Fluid Dynamics, Turbulence Modeling, Uncertainty Analysis, External Aerodynamics, Spacecraft Venting, Environmental Control Systems, Heat Transfer.

Degrees:

- Ph.D., Mechanical Engineering, University of Central Florida, Expected Completion Date 2014.
- M.S., Aerospace Engineering, University of Central Florida, May 2012.
- B.S., Aerospace Engineering, West Virginia University, May 2008.
- B.S., Mechanical Engineering, West Virginia University, May 2008.

Recent Work:

- Mars Science Laboratory Mission, NASA, Environmental Control System Impingement Analysis
- Mars Science Laboratory Mission, NASA, Nuclear Launch Approval Processes
- NASA, Source Evaluation Board
- Tracking and Data Relay Satellites (TDRS K/L), NASA, Environmental Control System Impingement Analysis
- Tracking and Data Relay Satellites (TDRS K/L) Missions, NASA, Spacecraft Venting Analysis
- Interface Region Imaging Spectrograph (IRIS), NASA, Environmental Control System Impingement Analysis
- GLORY Mission, NASA, Environmental Control System Impingement Analysis, Testing, Uncertainty Analysis

Honors and Awards:

- Group Achievement Award, ELVIS 2 Source Evaluation Board Team, NASA Kennedy Space Center, 2013.
- Group Achievement Award, Mars Science Lab, NASA Kennedy Space Center, 2012.
- Space Flight Awareness Award, NASA Kennedy Space Center, 2012.
- Certificate of Appreciation, NASA Kennedy Space Center, 2011.
- Distinguished Performance Rating, NASA, Launch Services Program, 2011.
- Completion of Accelerated Training Program, NASA Kennedy Space Center, 2008.
- NASA Cooperative Education Program, NASA Kennedy Space Center, 2006-2008.
- NASA West Virginia Space Grant Scholar, 2004-2008.
- Summa Cum Laude, West Virginia University, 2008.
- Ralph. M Barnes Senior Scholastic Achievement Award, West Virginia University (Graduate with highest GPA during the junior and senior years), 2008.
- Promise Scholar, West Virginia University, 2003-2008.
- President's List, West Virginia University (4.0 GPA), 2003-2008.

Publications:

1. Brink, J., Godfrey, G., Wittenborn, D., O'Keefe, D., **Groves. C**. (2005). Orbiter-External Tank (ET) Mate Simulation. National Aeronautics and Space Administration: John F Kennedy Space Center 2005 Technology, Development and Application Annual Report. 129-130.
2. Godfrey, G., O'Keefe, D., Whittenborn, D., **Groves, C**., & Kapr, F. (2006, December) Orbiter to External Fuel Tank Mating Simulation. Paper present at Engineering Design Graphics Division Mid-Year Meeting in Ft. Lauderdale, Florida.
3. **Groves, C**., Ilie, M., Schallhorn, Paul. "Comprehensive Approach to Verification and Validation of CFD Simulations Applied to Backward Facing Step," AIAA-2013-0258, 2013.

3.0 Literature Review:

A literature review was performed to determine the "State of the Art" method for calculating CFD uncertainties. CFD is extensively used in industry, government, and academia to design, investigate, operate, and improve understanding of fluid physics [3]. The rate of growth in using CFD as a research and engineering tool will be directly proportional to the level of credibility that the simulation can produce [3]. One needs to evaluate the uncertainty in the results of a CFD simulation to postulate a level of credibility. In 1986, The American Society of Mechanical Engineers (ASME) Journal of Fluids Engineering published a policy statement stating the need for quantification of numerical accuracy [1]. Other journals have issued similar statements [7]. These statements lead to research on the best method to determine numerical uncertainty. In 1995, Celik and Zhang published "Calculation of Numerical Uncertainty Using Richardson Extrapolation: Application to Some Turbulent Flow Calculations" which used Richardson's Extrapolation method to estimate the uncertainty in CFD [8]. In 1997, Roache published "Quantification of Uncertainty in Computational Fluid Dynamics" [7]. Roaches research also used the Richardson Extrapolation method to quantify CFD uncertainties.

In 1998, the AIAA has published a "Guide for the Verification and Validation of Computational Fluid Dynamics Simulations" [3]. This document provides guidelines for assessing credibility via verification and validation [3]. The document does not recommend standards due to issues not yet resolved, but defines several terms [3]. "Uncertainty is defined as a potential deficiency in any phase or activity of the modeling process that is due to lack of knowledge [3]." "Error is defined as a recognizable deficiency in any phase or activity of modeling and simulation that is not due to lack of knowledge [3]." "Prediction is defined as the use of a CFD model to foretell the state of a physical system under conditions for which the CFD model has not been validated [3]." Uncertainty and error are normally linked to accuracy in modeling and simulation[3]. The guide defines four predominate error sources: insufficient spatial discretization convergence, insufficient temporal discretization convergence, lack of iterative convergence, and computer programming, but does not make claims about the accuracy of predictions [3]. The guide emphasizes that systematically refining the grid size and time step is the most important activity in verification [3]. Once the grid has been refined such that the discretization error is in the asymptotic region, Richardson's extrapolation can be used to estimate zero-grid spacing [3]. A sensitivity analysis and uncertainty analysis are two methods for determining the uncertainty in CFD [3]. The validation test compares a CFD solution to experimental data [3]. The guide has outlined the terms and an overall structure to performing validation, but does not offer a quantitative method.

In 1999, Stern, Wilson, Coleman, and Paterson, E. G., published Iowa Institute of Hydraulic Research (IIHR) Report No. 407 titled "Verification and Validation of CFD Simulations" [9]. In 2001, the American Society of Mechanical Engineers (ASME) Journal of Fluids Engineering published a "Comprehensive Approach to Verification and Validation of CFD Simulations" in an attempt to provide a comprehensive framework for overall procedures and methodology [6]. Two papers were published on the subject in Parts I [6] and Parts II [10] and used the methodology documented in IIHR Report 407. Numerical errors and uncertainties in CFD can be estimated using iterative and parameter convergence studies [6]. The method uses three convergence conditions as possible in estimating uncertainties; (1) monotonic convergence which uses Richardson's extrapolation, (2) oscillatory convergence which uses the upper and lower bounds to estimate uncertainty, (3) divergence in which errors and uncertainties cannot be estimated [6]. The literature provides an approach for estimating errors and uncertainties in CFD simulations for each of the three cases [9, 6, 10]. The approach uses Richardson's extrapolation, which is not new, however; the method has been extended to use input parameters and correction factors to estimate errors and uncertainties [9, 6, 10]. The method examines two sources for error and uncertainty: modeling and simulation. Examples of modeling errors include geometry, mathematical equations, boundary conditions, turbulence models, etc. [IVII]. Examples of numerical errors include discretization, artificial dissipations, incomplete iterative and grid convergence, lack of conservation of mass, momentum, energy, internal and external boundary non-continuity, computer round-off etc. [4]. The method lacks correlations among errors and assumes these are negligible, which may be inappropriate for some circumstances [6]. Additionally, the method provides a quantitative approach for determining the iterative convergence uncertainty [6]. Iterative Convergence must be evaluated and is typically done by monitoring the residuals order of magnitude drop graphically [6]. For oscillatory convergence, the deviation of a residual from the mean provides estimates of the iterative convergence [6]. This is based on the range of the maximum S_U and minimum S_L values [6]. For convergent iterative convergence, a curve-fit is used [6]. For a mixed convergent/oscillatory, iterative convergence is estimated using the amplitude and the maximum and minimum values [6]. A method for confirming validation is presented as compared to experimental data [6].

In 2008, the International Towing Tank Conference (ITTC) has published "Recommended Procedures and Guidelines – Uncertainty Analysis in CFD Verification and Validation Methodology and Procedures" [11]. The ITTC guide was largely based off of the methodology and procedures presented in the ASME Journal of Fluids Engineering a "Comprehensive Approach to Verification and Validation of CFD Simulations" [11]. Also in 2008, the ASME Journal of Fluids Engineering published a "Procedure for Estimating and Reporting of Uncertainty Due to Discretization in CFD Applications" [12].

In 2011, the National Energy Technology Laboratory (NETL) conference proceedings held a major section related to CFD Uncertainty Calculation [13]. Celik presented "Critical Issues with Quantification of Discretization Uncertainty in CFD" [13]. The proceedings were based off of the ASME "Comprehensive Approach to Verification and Validation of CFD Simulations" [6].

In 2009, the American Society of Mechanical Engineers published "Standard for Verification and Validation in Computational Fluid Dynamics and Heat Transfer", ASME V&V 20-2009 [14]. The standard provides a procedure for estimating the uncertainty and is based off of the literature presented above.

Summary of Literature Review: A thorough literature review has been performed to determine the best method to evaluate the uncertainty in CFD predictions. Both major journals in mechanical and aerospace engineering, AIAA and ASME, have published articles on this subject. The ASME Standard methodology has been adopted by many researchers and provides a detailed approach to calculate uncertainty in CFD from different levels of grid refinement. The method published by the ASME Journal of Fluids Engineering (ASME V&V 20-2009 [14]) is the state of the art for determining the uncertainty in CFD predictions and was used for the completed research problem.

4.0 Summary of State of the Art CFD Uncertainty Analysis:

A summary of the ASME V&V 20-2009 **"Standard for Verification and Validation in Computational Fluid Dynamics and Heat Transfer"** is provided in this section. An overview of the validation process is shown in Figure 1.

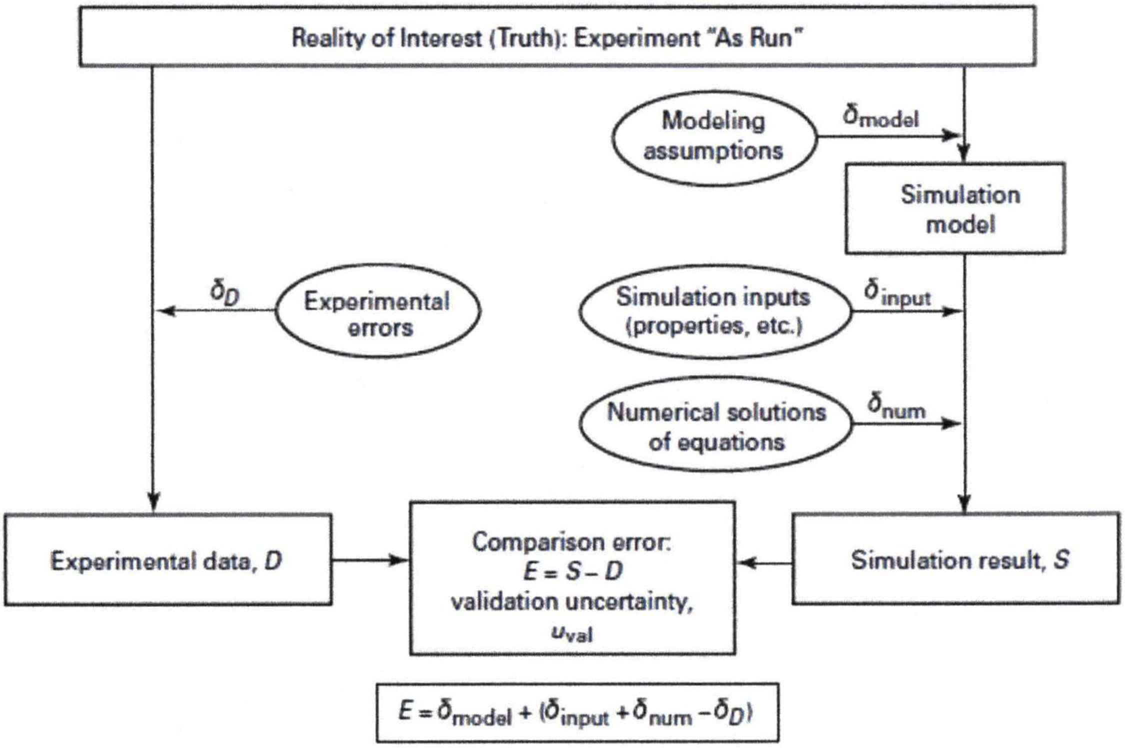


Figure 1 – Overview of Validation Process [14]

To estimate the interval within which δmodel falls with a given degree of confidence, the following error sources are used (U_{num},U_{input},U_D). The resulting uncertainty equation is shown in equation1.

$$U_{Val} = \sqrt{U_{num}^{\ 2} + U_{input}^{\ 2} - U_D^{\ 2}} \tag{1}$$

Numerical, U_{num}

The uncertainties of the variables with monotonic convergence (numerical) are calculated using Richardson's extrapolation as outlines by ASME V&V-2009 [14]. This is accomplished through the five-step procedure. Step 1, calculate representative grid size, ***h*** as shown in equation 2.

$$h_1 = \left(\frac{Total\ Volume}{total\ number\ of\ cells\ in\ fine\ grid}\right)^{\frac{1}{3}}$$

$$h_2 = \left(\frac{Total\ Volume}{total\ number\ of\ cells\ in\ medium\ grid}\right)^{\frac{1}{3}}$$

$$h_3 = \left(\frac{Total\ Volume}{total\ number\ of\ cells\ in\ coarse\ grid}\right)^{\frac{1}{3}} \tag{2}$$

Step 2 is to select three significantly ($r>1.3$) grid sizes and computer the ratio as shown in equation 3.

$$r_{21} = \frac{h_2}{h_1}$$

$$r_{32} = \frac{h_3}{h_2} \tag{3}$$

Step 3 is to calculate the observed order, p, as shown in equation 4. This equation must be solved iteratively.

$$p = \left[\frac{1}{\ln(r_{21})}\right] * [\ln\left(\frac{\varepsilon_{32}}{\varepsilon_{21}}\right) + ln\left(\frac{r_{21}{}^{p} - sign\left(\frac{\varepsilon_{32}}{\varepsilon_{21}}\right)}{r_{32}{}^{p} - sign\left(\frac{\varepsilon_{32}}{\varepsilon_{21}}\right)}\right) \qquad (4)$$

Step 4 is to calculate the extrapolated values as shown in equation 5.

$$S_{ext}{}^{21} = \frac{(r_{21}{}^{p} * S_{k1} - S_{k2})}{(r_{21}{}^{p} - 1)}$$

$$e_a{}^{21} = \frac{(S_{k1} - S_{k2})}{(S_{k1})} \qquad (5)$$

Step 5 is to calculate the fine grid convergence index and numerical uncertainty as shown in equation 6. This approached used a factor of safety of 1.25 and assumed that the distribution is Gaussian about the fine grid, 90 % confidence.

$$GCI_{fine}{}^{21} = \frac{1.25 * e_a{}^{21}}{(r_{21}{}^{p} - 1)}$$

$$U_{monotonic} = \frac{GCI_{fine}{}^{21}}{1.65} \qquad (6)$$

Input, U_{inpt}

The uncertainty associated with the CFD calculation is the compilation of the elemental errors associated with each of the numerical, input, and solver errors. This uncertainty can be calculated using a Data Reduction equation the form $r = r(X_1, X_2, \ldots X_J)$ is shown in equation 7, below.

$$U_{CFD} = \left(\sum_{i=1}^{J}\left\{\left(\frac{\partial r}{\partial X_i}\right)^2 B_i^2\right\} + 2\sum_{i=1}^{J}\sum_{k=i+1}^{J}\left\{\left(\frac{\partial r}{\partial X_i}\right)\left(\frac{\partial r}{\partial X_k}\right)[B_i B_k]_{correlated}\right\} + \sum_{i=1}^{J}\left\{\left(\frac{\partial r}{\partial X_i}\right)^2 P_i^2\right\}\right)^{1/2} \qquad (7)$$

Where,

B_i = the systematic (bias) error associated with variable X_i,

$(B_i B_k)_{correlated}$ = the correlated systematic error between variables X_i and X_k,

and P_i = the random error associated with variable X_i.

For the calculation, the correlated errors and random errors are neglected and the data reduction equation reduces to the following, as shown in equation 8.

$$U_{CFD} = \left(\sum_{i=1}^{J}\left\{\left(\frac{\partial r}{\partial X_i}\right)^2 B_i^2\right\}\right)^{1/2} \tag{8}$$

Experimental, U_D

The effect of experimental data on the uncertainty is described in the standard. The proposed project will not include experimental data and therefore, the reference to the standard is suggested.

4.1 Discussion of State of the Art CFD Uncertainty Analysis:

There are a few items to note from the summary of the ASME standard. The summary assumes that there are no random errors and that none of the input variables are correlated. Additionally, the standard states that the numerical error can be calculated by the 5-step procedure, which is essentially Richardson's Extrapolation Method. There are additional assumptions to Richardson's Extrapolation. To apply this method, the variable must be monotonically increasing (ie In the extrapolated Region). The input variables are assumed to be oscillatory convergence. A convergence study can be calculated to determine if the grid is monotonic, oscillatory, or divergence.

Convergence studies require a minimum of three solutions to evaluate convergence with respect to an input parameter [2]. Consider the situation for 3 solutions corresponding to fine S_{k1}, medium S_{k2}, and coarse S_{k3} values for the *kth* input parameter [2]. Solution changes **ε** for medium-fine and coarse-medium solutions and their ratio R_k are defined by [2]:

$$\varepsilon_{21} = S_{k2} - S_{k1}$$

$$\varepsilon_{32} = S_{k3} - S_{k2}$$

$$R_k = \varepsilon_{21} / \varepsilon_{32} \quad (9)$$

Three convergence conditions are possible[2]:

(i) Monotonic convergence: $0 < R_k < 1$

(ii) Oscillatory convergence: $R_k < 0^i$

(iii) Divergence: $R_k > 1$

The quantity of interest for the ECS/ spacecraft system is velocity magnitude. Three grids can be compared, and the convergence conditions determined for every point in the computational domain. This is accomplished through interpolation between the medium to coarse grid and the fine to coarse grid. The velocity magnitude from the medium and fine grids are interpolated on to the coarse grid. Then the solutions changes, ε_{21}, ε_{32}, R_k, and convergence conditions are calculated for every point in the domain.

This interpolation can induce errors in the solution, which has been seen by the author in recent publication. The method that was used in the backward facing step used a 'zeroth' order interpolation scheme in FLUENT. The proposal would like to find a higher order interpolation scheme and plot the three different convergence conditions. Treating the grid as a monotonically increasing parameter in the entire domain may be inappropriate. Additionally for an oscillatory convergence parameter, Stern, Wilson, Coleman, and Paterson recommend the following [2]. ***S*** is the simulated result. For this case it is the upper velocity S_U and the lower velocity S_L.

$$U_{Oscillatory} = \frac{1}{2}(S_U - S_L) \quad (10)$$

It is hypothesized that treating the grid as a oscillatory input parameter might provide a faster, more accurate method to estimate the uncertainty in the numeric's.

4.2 Applying the of State of the Art CFD Uncertainty Analysis to a Backward Facing Step:

The author applied the ASME standard to a backward facing step in AIAA-2013-0258 [13]. A summary of this paper is included here. The proposed problem is to apply this method to the spacecraft / ECS system problem.

The quantity of interest for the backward facing setup is velocity magnitude. Three grids were compared, and the convergence conditions were determined for every point in the computational domain. This is accomplished through interpolation between the medium to coarse grid and the fine to coarse grid. The velocity magnitude from the medium and fine grids are interpolated on to the coarse grid. Then the solutions changes, $\boldsymbol{\varepsilon_{21}}$, $\boldsymbol{\varepsilon_{32}}$, $\mathbf{R_k}$, and convergence conditions are calculated for every point in the domain. Figure 2 shows the different convergence conditions inside the computational domain for the grid refinement study.

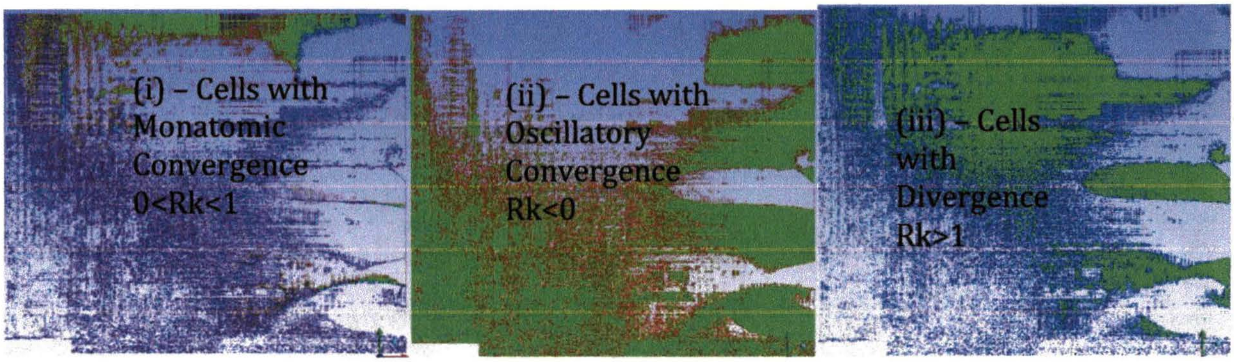


Figure 2: Convergence conditions for a Flat plate – Grid refinement 1

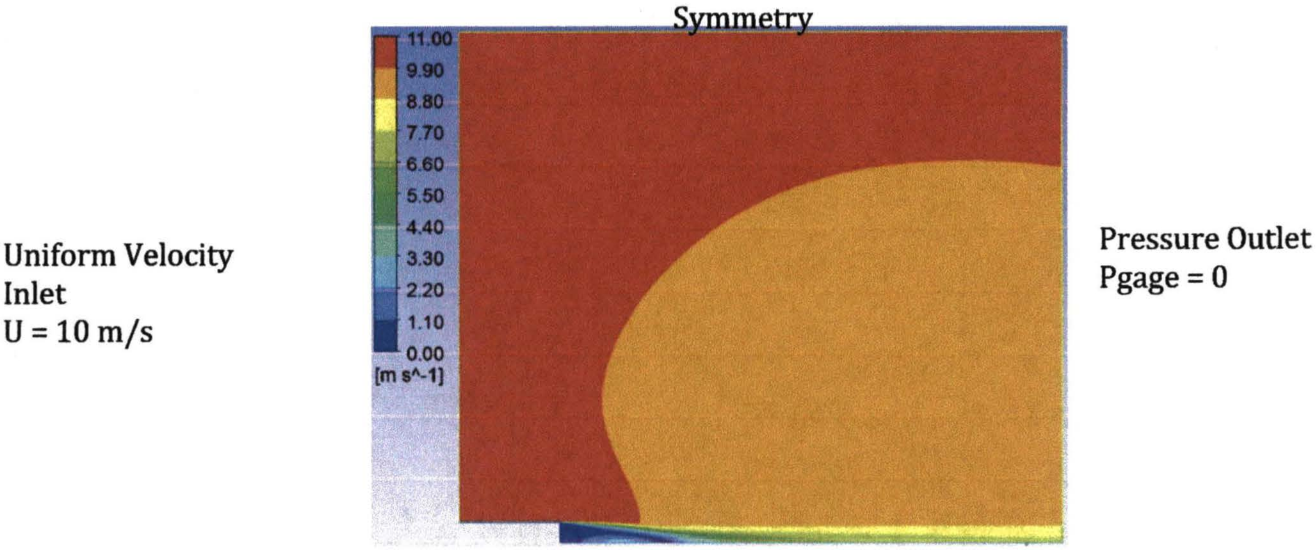


Figure 3: Velocity Magnitude for Flow over Backward facing step – Coarsest Grid (Structured 1,192,000 cells)

Numerical Results for Backward Facing Step

Input Variables:

$$U_{CFD} = \left(\sum_{i=1}^{J}\left\{\left(\frac{\partial r}{\partial X_i}\right)^2 B_i^2\right\}\right)^{1/2}$$

A list of variables for the k-e-realizable turbulence model analyzed is listed in Table 1.

Table 1: Uncertainty Variables, Xi

Type of Variable	Variables Xi	Value	Bias Error
Boundary Conditions	epsilion turbulent mixing length dissipation rate inlet (m2/s3)	0.5	0.5
	k turbulent intensity kinetic energy inlet (m2/s2)	0.05	0.05
	pressure outlet (Pa)	101325	2%
	velocity inlet (m/s)	10	0.5
Fluid Properties	kinematic viscosity nu represents air [0-50-100] deg C	1.79E-06	[13.6e-06 -> 23.06e-06]
Grid Size	Method - Uses Oscillatory Uncertainty	1,192,000 1,862,500 3,311,689	
Numerical	Method - Uses Richardson's Extrapolation (ASME 5 Step Procedure) – Calculated for Velocity at each Cell		
Solver	OpenFOAM (SimpleFoam) vs. Fluent		
Turbulence Models	ke-realiable, kwSST, and SpalartAllmaras		

Expanding the data reduction equation for the listed variables as shown in order from top to bottom.

$$\begin{aligned} U_{CFD-Velocity} = \Bigg(& \left(\left(\frac{\partial V}{\partial e}\right)^2 B_e^2\right) + \left(\left(\frac{\partial V}{\partial k}\right)^2 B_k^2\right) + \left(\left(\frac{\partial V}{\partial p}\right)^2 B_p^2\right) + \left(\left(\frac{\partial V}{\partial U}\right)^2 B_u^2\right) \\ & + \left(\left(\frac{\partial V}{\partial nu}\right)^2 B_{nu}^2\right) + \left(\left(\frac{\partial V}{\partial g}\right)^2 B_g^2\right) + \left(\left(\frac{\partial V}{\partial num}\right)^2 B_{num}^2\right) + \left(\left(\frac{\partial V}{\partial solver}\right)^2 B_{solver}^2\right) \\ & + \left(\left(\frac{\partial V}{\partial turb}\right)^2 B_{turb}^2\right)\Bigg)^{1/2} \end{aligned}$$

Each of the variables was analyzed separately for their elemental error sources. The following plots show the each variables and their corresponding uncertainty plot as a function of the percent uncertainty in the CFD Velocity prediction. The percent uncertainty is calculated by dividing by the local velocity (ie the uncertainty velocity in each cell divided by the velocity in each cell). There may be a more appropriate way to non-dimensionalize , such as using the average inlet velocity.

The uncertainty for each of the following was calculated as shown below for each cell using the following method outlined by Stern, Wilson, Coleman, and Paterson [2]. ***S*** is the simulated result. For this case it is the upper velocity $\boldsymbol{S_U}$ and the lower velocity $\boldsymbol{S_L}$.

$$U_{Oscillatory} = \frac{1}{2}(S_U - S_L)$$

epsilion turbulent mixing length dissipation rate inlet (m^2/s^3)

For a value of 0.5 +/- 0.5 m^2/s^3, the uncertainty in the velocity prediction was 0 – 1.155 percent as shown in Figure 4.

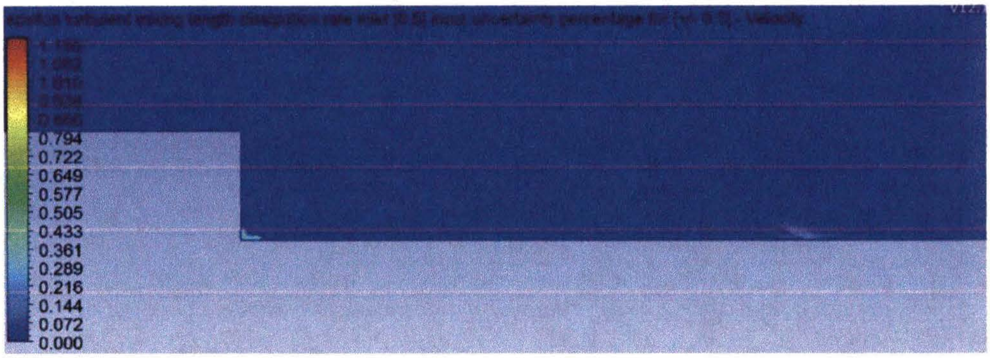


Figure 4: Epsilon Turbulent Mixing Length Dissipation Rate Inlet – Velocity Uncertainty Percentage

k turbulent intensity kinetic energy inlet (m^2/s^2)

For a value of 0.05 +/- 0.05 m^2/s^2, the uncertainty in the velocity prediction was 0 – 0.785 percent as shown in Figure 5.

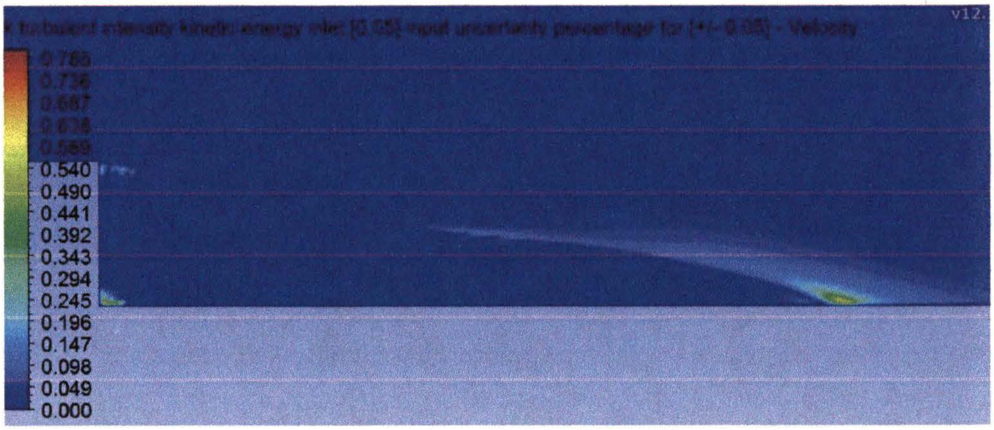


Figure 5: k Turbulent Intensity Kinetic Energy Inlet – Velocity Uncertainty Percentage

Pressure outlet (Pa)

For a value of 101325 +/- 2% Pa, the uncertainty in the velocity prediction was 0 – 20 percent as shown in Figure 6.

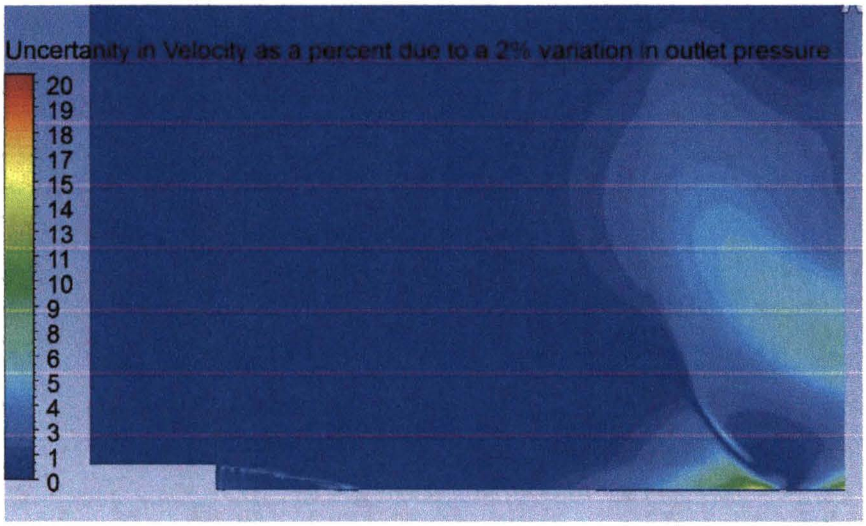


Figure 6: Pressure Outlet – Velocity Uncertainty Percentage

Velocity Inlet (m/s)

For a value of 10 +/- 0.5 m/s, the uncertainty in the velocity prediction was 0 – 6.558 percent as shown in Figure 7.

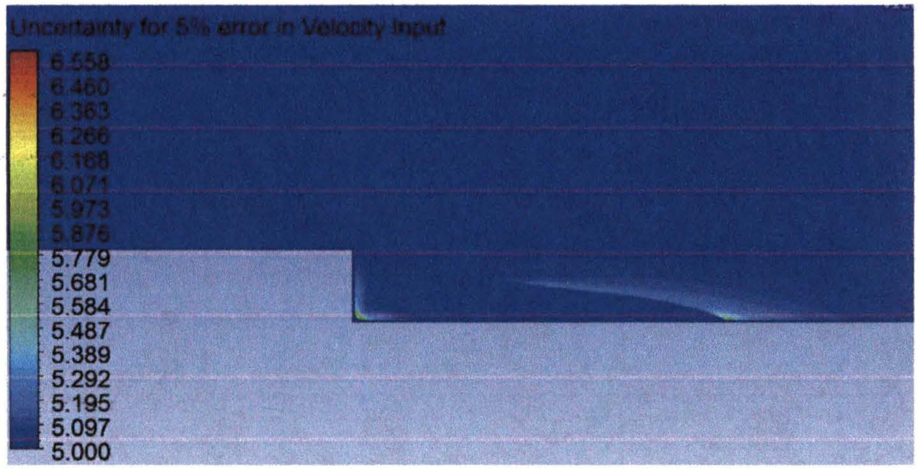


Figure 7: Velocity Inlet – Velocity Uncertainty Percentage

Kinematic viscosity nu=17.06e-06 [13.6e-06 -> 23.06e-06] (m^2/s) represents air [0-50-100] degrees C

For a value of nu=17.06e-06 [13.6e-06 -> 23.06e-06] (m^2/s), the uncertainty in the velocity prediction was 0 – 27.727 percent as shown in Figure 8.

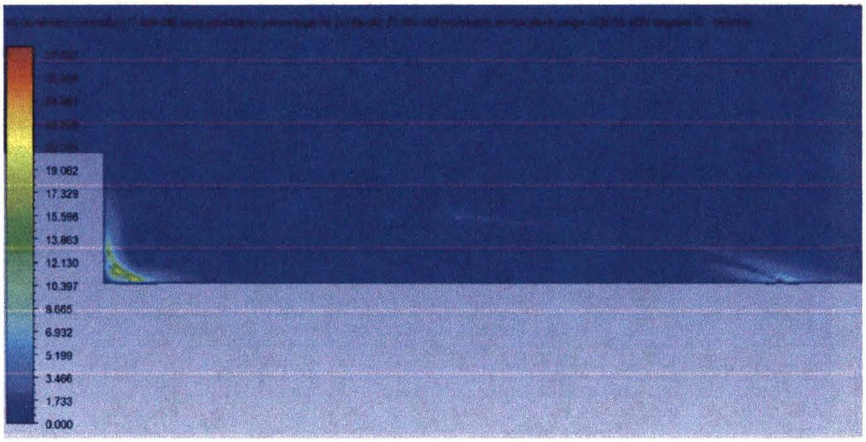


Figure 8: Kinematic Viscosity – Velocity Uncertainty Percentage

Grid size

For a grid size of 1,192,000 cells [grid 2 -1,862,500 cells], [grid3 - 3,311,689 cells], the uncertainty in the velocity prediction was 0 – 698 percent as shown in Figure 9.

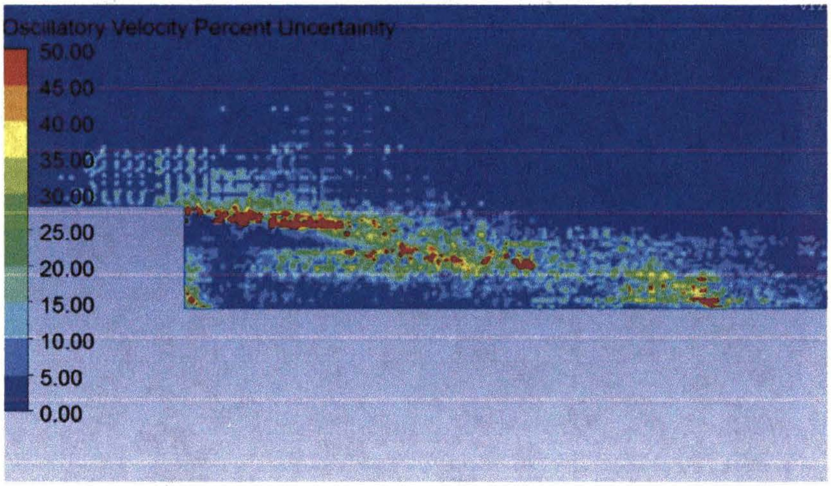


Figure 9: Grid Size – Velocity Uncertainty Percentage

Turbulence Models

The ke-realiable, kwSST, and SpalartAllmaras turbulence models converged using OpenFoam and the uncertainty was calculated as an oscillatory input parameter as shown in Figure 10.

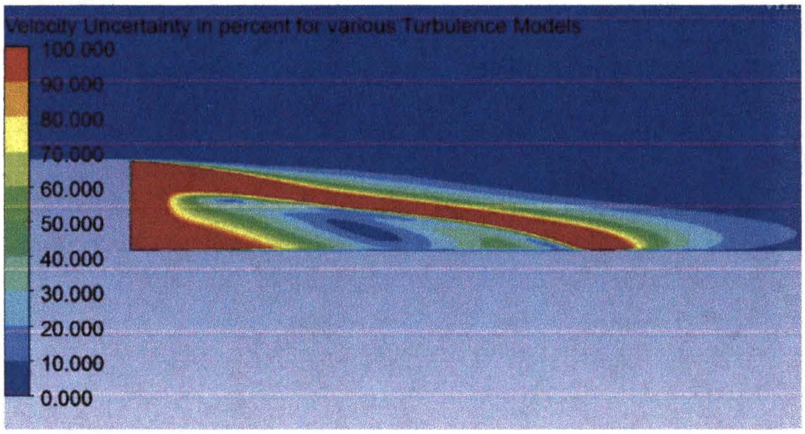


Figure 10: Turbulence Models – Velocity Uncertainty Percentage

Solver

OpenFoam and Fluent were used to calculate the velocity distribution on the backward facing step and the uncertainty was calculated as an oscillatory input parameter as shown in Figure 11.

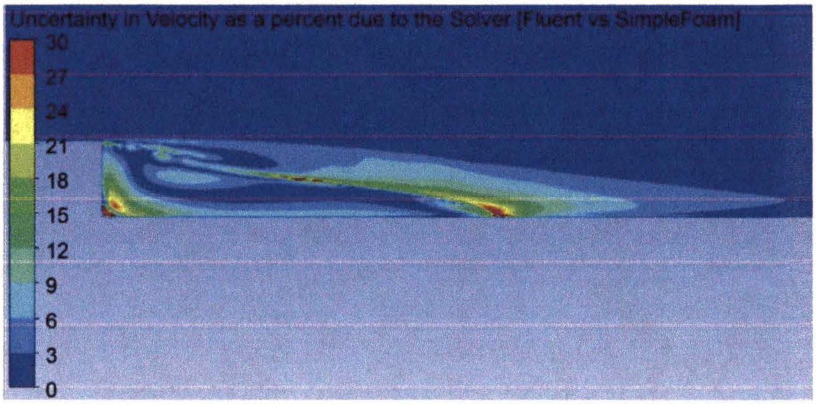


Figure 11: Solver – Velocity Uncertainty Percentage

The uncertainties of the variables with monotonic convergence (numerical) are calculated using Richardson's extrapolation as outlines by ASME V&V-2009 [14]. This is accomplished through the five-step procedure. Step 1, calculate representative grid size, ***h*** as shown.

$$h_1 = \left(\frac{Total\ Volume}{total\ number\ of\ cells\ in\ fine\ grid}\right)^{\frac{1}{3}}$$

$$h_2 = \left(\frac{Total\ Volume}{total\ number\ of\ cells\ in\ medium\ grid}\right)^{\frac{1}{3}}$$

$$h_3 = \left(\frac{Total\ Volume}{total\ number\ of\ cells\ in\ coarse\ grid}\right)^{\frac{1}{3}}$$

Step 2 is to select three significantly (r>1.3) grid sizes and computer the ratio as shown in equation 5.

$$r_{21} = \frac{h_2}{h_1}$$

$$r_{32} = \frac{h_3}{h_2}$$

Step 3 is to calculate the observed order, p, as shown in equation 6. This equation must be solved iteratively.

$$p = \left[\frac{1}{\ln(r_{21})}\right] * [\ln\left(\frac{\varepsilon_{32}}{\varepsilon_{21}}\right) + ln\left(\frac{{r_{21}}^p - sign\left(\frac{\varepsilon_{32}}{\varepsilon_{21}}\right)}{{r_{32}}^p - sign\left(\frac{\varepsilon_{32}}{\varepsilon_{21}}\right)}\right)$$

Step 4 is to calculate the extrapolated values as shown in equation 7.

$${S_{ext}}^{21} = \frac{({r_{21}}^p * S_{k1} - S_{k2})}{({r_{21}}^p - 1)}$$

$${e_a}^{21} = \frac{(S_{k1} - S_{k2})}{(S_{k1})}$$

Step 5 is to calculate the fine grid convergence index and numerical uncertainty as shown. This approached used a factor of safety of 1.25 and assumed that the distribution is Gaussian about the fine grid, 90 % confidence.

$$GCI_{fine}^{21} = \frac{1.25 * e_a^{21}}{(r_{21}^{p} - 1)}$$

$$U_{monotonic} = \frac{GCI_{fine}^{21}}{1.65}$$

Numerical

For a grid size of 1,192,000 cells [grid 2 -1,862,500 cells], [grid3 - 3,311,689 cells], the uncertainty in the velocity prediction was 0 – 5300 percent as shown in Figure 12 as estimated by Richardson's extrapolation method.

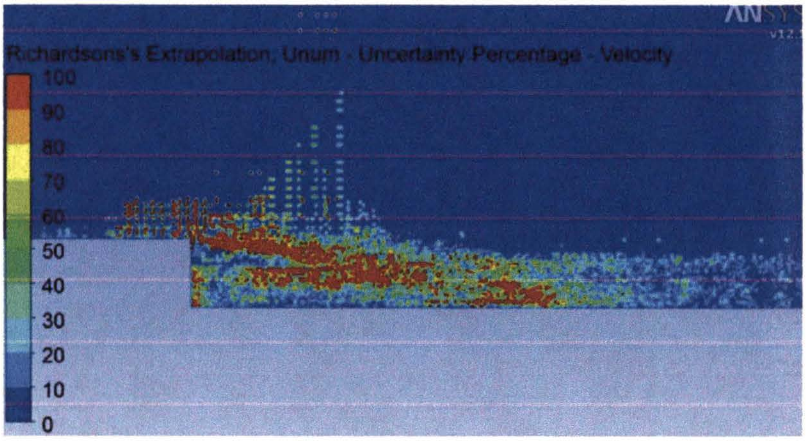


Figure 12: Numerical – Velocity Uncertainty Percentage

A root-sum-squared (rss) of the uncertainty variables was calculated (omitting Richardson's Extrapolation – see Discussion) and the velocity magnitude is shown in figure 13 with the corresponding uncertainty.

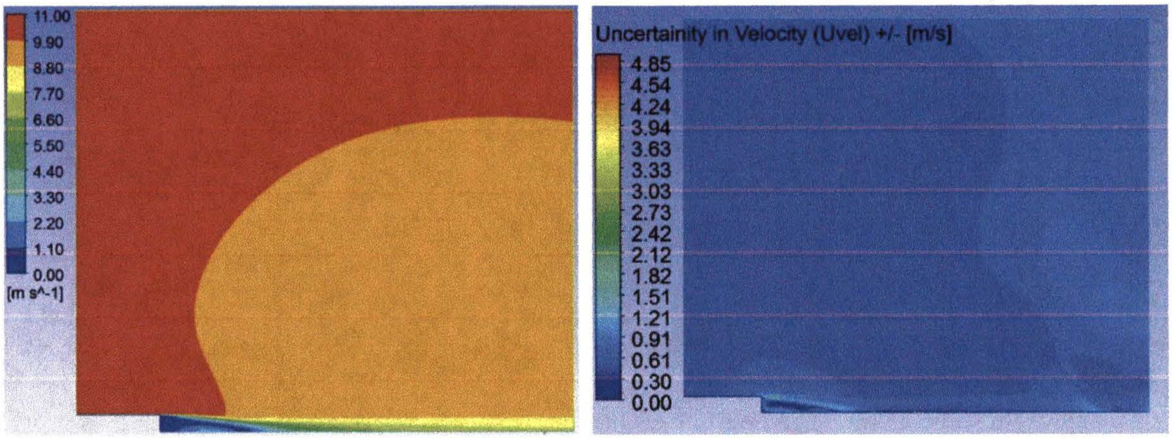


Figure 13: Velocity Prediction and Uncertainty Plot for ke-realizable Turbulence Model

The highest uncertainty is +/- 4.85 m/s. This occurs in the region shown in Figure 14 in red. Figure 14 is the same data presented on the right hand side of Figure 14, except zoomed in to the region near the backward step and a smaller scale is used.

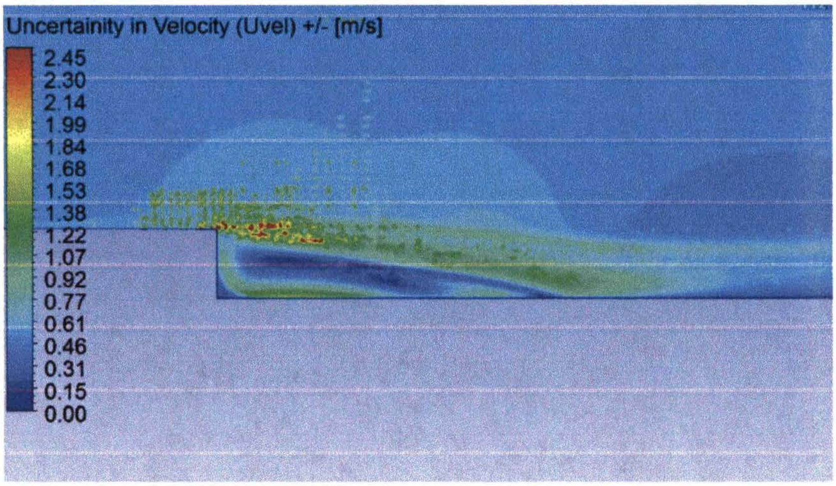


Figure 14: Velocity Uncertainty Plot for ke-realizable Turbulence Model

Discussion

The monotonic convergence uncertainty calculation was omitted in the rss uncertainty plot due to the values that were produced by using this method. The method produced uncertainty values that were on the order of 5000 percent of the localized velocity in the region near the backward step. It is believed this is due to the turbulence and/or the interpolation between the 3 grids. Turbulence is calculated as a steady state value and fluctuations about that steady state. The fluctuations are inducing a non-linear result between the three grids and providing very large uncertainty bands in the localized region near the backward step. However, once you move approximately 5 lengths downstream of the backward step, the method begins producing reasonable results of 0 – 30 percent of the localized velocity. Treating the highly turbulent region behind the backward step as a monotonic case is inappropriate. It is believed that treating the grid as an input parameter with oscillatory convergence provides better results for a steady state, turbulent CFD simulation. This is evident in the R_k values shown in Figure 2. Most of the cells are exhibiting oscillatory convergence. It is believed all cells are exhibiting oscillatory convergence, however depending on when the sample takes place, one could misrule the results as monotonic or divergent. The interpolation between the three grids could also be inducing this non-linear result. The current method for interpolation is using FLUENT to write out an interpolation file, then reading that file back into FLUENT onto a different grid. This method will be investigated further and other interpolations methods considered in the future.

5.0 Proposed Problem:

Prior to launch, cold air (air conditioning) flows downward around the spacecraft after it has been encapsulated in the Payload Fairing [16]. The cold air is delivered through an air-conditioning (AC) pipe, which intersects the fairing and flows past a diffuser located at the pipe/fairing interface [16]. After passing over the spacecraft, it is finally discharged through vents [16]. The Payload Fairing air conditioning is cut off at lift off [16]. An overview the geometry for an Environmental Control System (ECS) along with the swirled airflow is shown in figures 15 and 16, respectively.

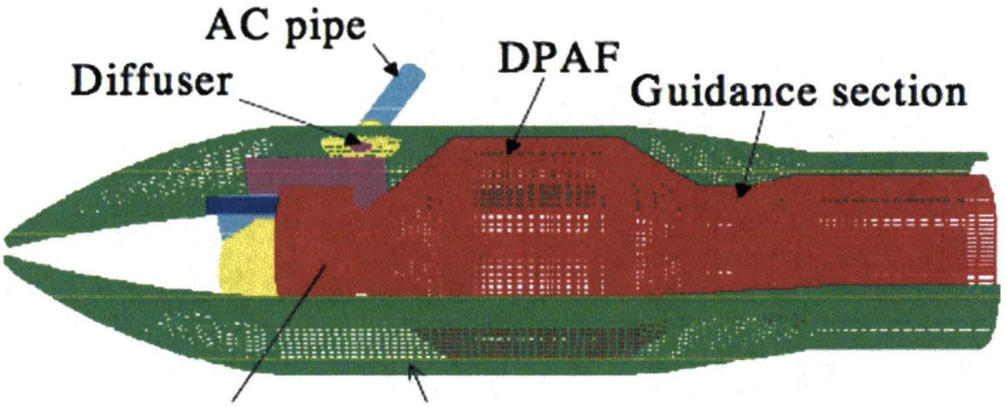


Figure 15 – Environmental Control System (ECS) Overview [16]

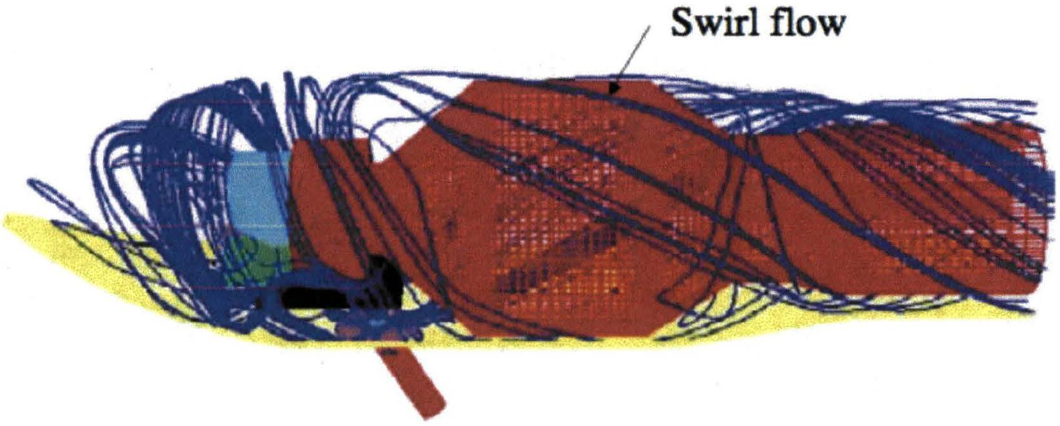


Figure 16 – ECS Airflow Swirl [16]

This problem has been previously solved using overset grids and compared to laser doppler test data as described in AIAA-2005-4910 [18]. An example of the airflow testing performed is shown in figure 17.

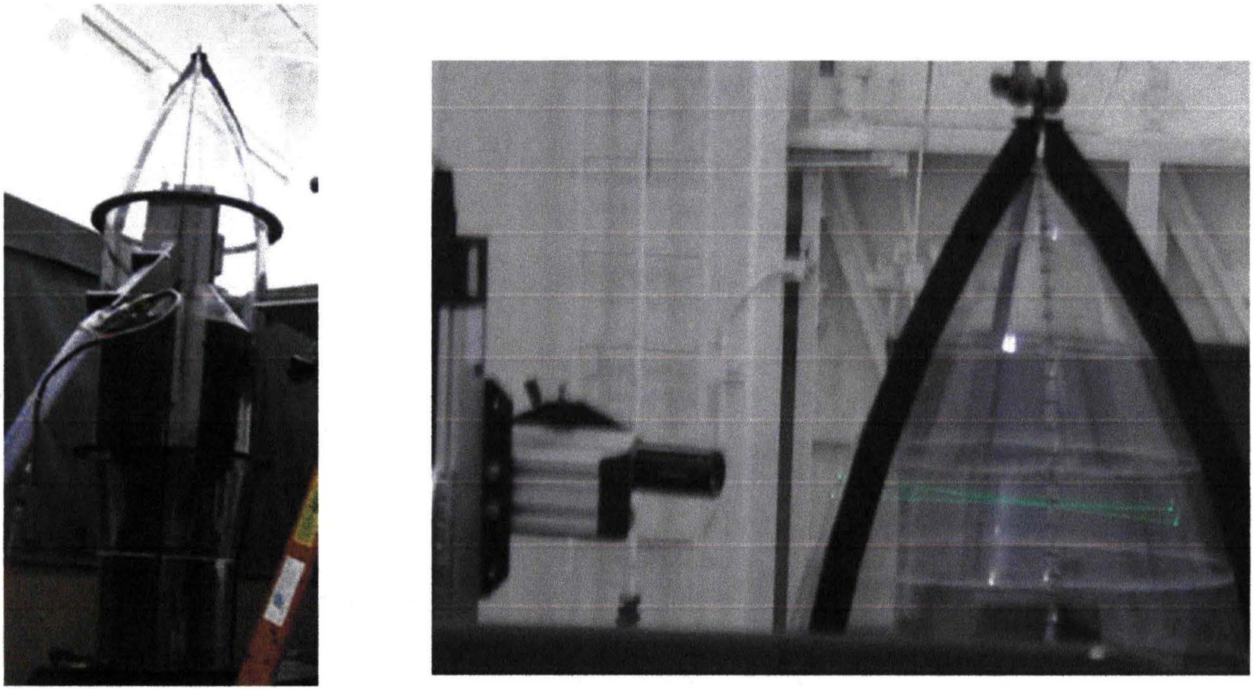
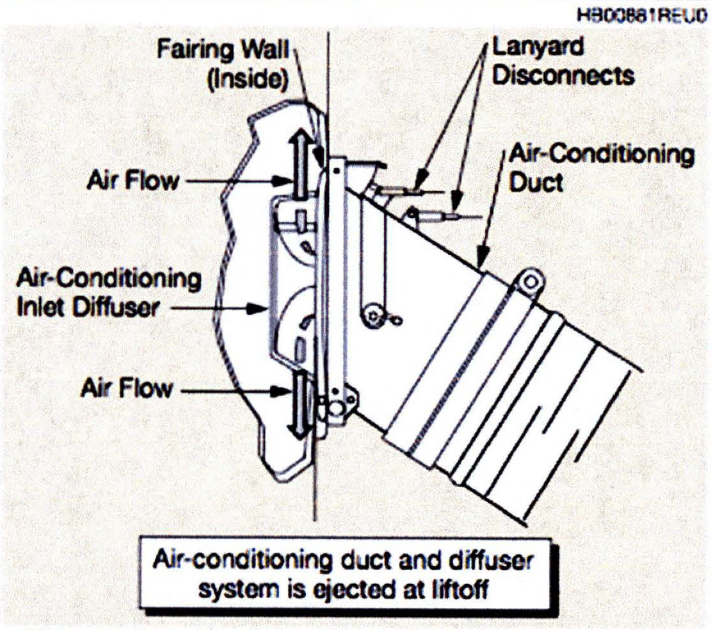


Figure 17 – ECS System Airflow Testing [18]

The example shown above is the only published result of the ECS airflow problem. It is difficult to publish this material due to the proprietary information needed. There are seven different rockets currently being used in the United States for Evolved Expendable Launch Vehicles (EELV). These rockets include the Delta II, Delta IV, Atlas V, Pegasus, Taurus, and Falcon 9 [19]. A summary of each of these rockets's ECS systems that are available in the public information is included below. The proposed problem is to create (3) generic representations that could encompass the flow regimes seen in the EELV fleet. Each of these rockets have a public available source called a payload planners guide or users guide. Each of these guides have been studied extensively and the appropriate information related to the ECS systems are presented next.

Delta II

Air-conditioning is supplied to the spacecraft via an umbilical after the payload fairing is mated to the launch vehicle [20]. The payload air-distribution system provides air at the required temperature, relative humidity, and flow rate as measured [20]. The air-distribution system uses a diffuser on the inlet air-conditioning duct at the fairing interface [20]. If required, a deflector can be installed on the inlet to direct the airflow away from sensitive spacecraft components [20]. The air can be supplied to the payload between a rate of 1300 to 1700 scfm [20]. The diameter of Fairing is 3 meters [20]. Figure 18 shows the Delta II Payload Air Distribution System.



Air-conditioning duct and diffuser system is ejected at liftoff

Figure 4-1. Payload Air Distribution System

Figure 18 – Delta II Payload Air Distribution System [20]

Delta IV

The air is supplied to the payload at a maximum flow rate of 36.3 kg/min to 72.6 kg/min (80 to 160 lb/min) for 4-m fairing launch vehicles and 90.7 kg/min to 136.0 kg/min (200 to 300 lb/min) for 5-m fairing launch vehicles [21]. Air flows around the payload and is discharged through vents in the aft end of the fairing [21]. Fairing sizes are 4 meters and 5 meters in diameter [21]. Figure 19 and 20 depict the Delta IV airflow system.

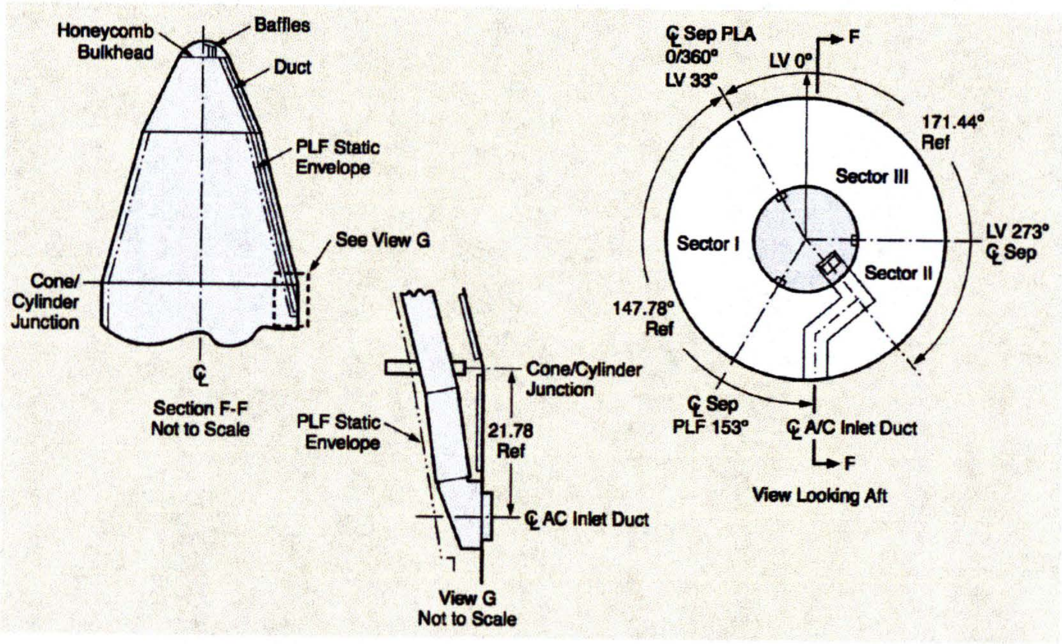


Figure 4-2. 5-m Metallic Fairing Payload Air-Distribution System

Figure 19 – 5m Metallic Payload Air-Distribution System [21]

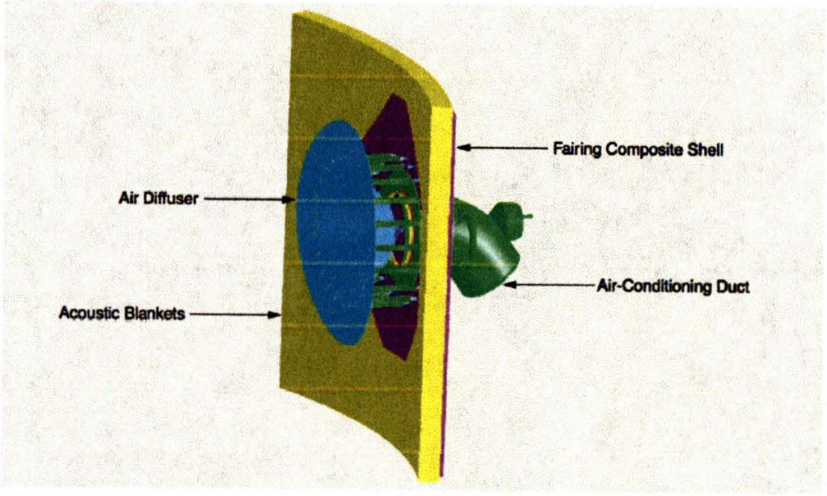


Figure 4-1. Standard 4-m Composite Fairing and 5-m Composite Fairing Air-Conditioning Duct Inlet Configuration

Figure 20 – Standard Air-Conditioning Duct Inlet [21]

Atlas V

Internal ducting defectors in the PLF direct the gas upward to prevent direct impingement on the spacecraft [22]. The conditioning gas is vented to the atmosphere through one-way flapper doors below the spacecraft [22]. The PLF air distribution system will provide a maximum air flow velocity in all directions of no more than 9.75 mps (32 fps) for the Atlas V 400 and 10.67 mps (35 fps) for the Atlas V 500 [22]. There will be localized areas of higher flow velocity at, near, or associated with the air conditioning outlet [22]. Maximum air flow velocities correspond to maximum inlet mass flow rates [22]. Reduced flow velocities are achievable using lower inlet mass flow rates [22].

- Flow Rates

  A) Atlas V 400: 0.38–1.21 kg/s ±0.038 kg/s (50–160 lb/min ±5 lb/min), [22]

  B) Atlas V 500: 0.38–2.27 kg/s ±0.095 kg/s (50–300 lb/min ±12.5 lb/min) [22]

The fairing sizes are 4meters and 5 meters in diameter [22].

Pegasus

The fairing is continuously purged with filtered air [23]. The flowrate of air through the fairing is maintained between 50 and 200 cfm [23]. The air flow enters the fairing forward of the payload and exits aft of the payload [23]. There are baffles on the inlet that minimize the impingement velocity of the air on the payload [23]. The fairing diameter is 0.97 meters [23].

Taurus

Upon encapsulation within the fairing and for the remainder of ground operations, the payload environment will be maintained by the Taurus Environmental Control System (ECS) [24]. The fairing inlet conditions are selected by the Customer [24]. The fairing diameters are 63 inches and 92 inches [24].

Falcon 9

Once fully encapsulated and horizontal, the Environmental Control System (ECS) is connected [25]. Payload environments during various processing phases are [25]:

- In hanger, encapsulated – Flow Rate: 1,000 cfm [25]
- During rollout: 1,000 cfm [25]
- On pad: Variable from 1000 to 4500 cfm [25]

The fairing diameter is 5.2 meters [25].

5.1 Proposed Objectives and Methods:

The proposed objectives and methods are presented in Table 2.

Table 2: Objectives and Methods

Objectives	Methods
1. Develop, model, and perform a CFD analysis of (3) generic non-proprietary environmental control system / spacecraft configurations	FLUENT/OPENFOAM are commercially/opensource CFD software capable of modeling the turbulent, highly 3-D, relatively incompressible flow found in spacecraft environmental control systems.
2. Perform an uncertainty analysis of the CFD model	The state of the art method from ASME Journal of Fluids Engineering "Comprehensive Approach to Verification and Validation of CFD Simulations" will be used. This method requires three separate grids and solutions, which quantify the error bars around CFD predictions. Fluent/OPENFOAM will be used.
3. Compile a table of uncertainty parameters	A table of uncertainty parameters will be constructed that could be used to estimate the uncertainty in a CFD model of an ECS/spacecraft.

5.2 Proposed Method for Objective 1:

Objective: Develop, model, and perform a CFD analysis of (3) generic non-proprietary environmental control system / spacecraft configurations

The following information can be concluded about the publically releasable ECS system data presented in the previous section. The fairing sizes are approximately 1m, 1.6m, 2.3m, 3m, 4m, 5m in diameter. It is proposed that the (3) generic fairing diameters are selected to envelop the EELV fairing configurations as follows.

- 0.75m
- 3.5 m
- 5.5 m

The inlet conditions range from 1000 cfm to 4500 cfm.

The three proposed generic models have been created in an Computer Aided Drafting (CAD) model software Pro/ENGINEER. The proposed configurations are shown in figure 21, 22, and 23, respectively.

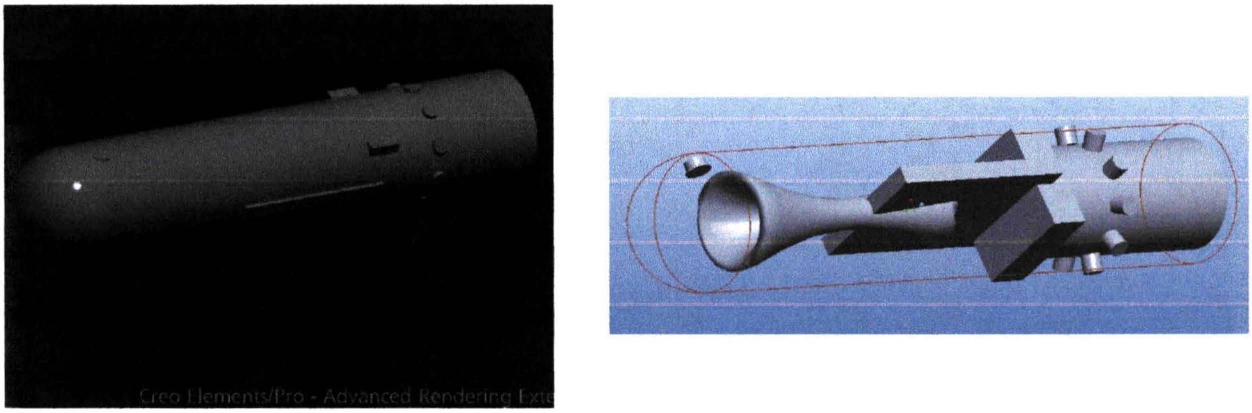


Figure 21 – Proposed CAD ECS Model with 0.75m Diameter Fairing

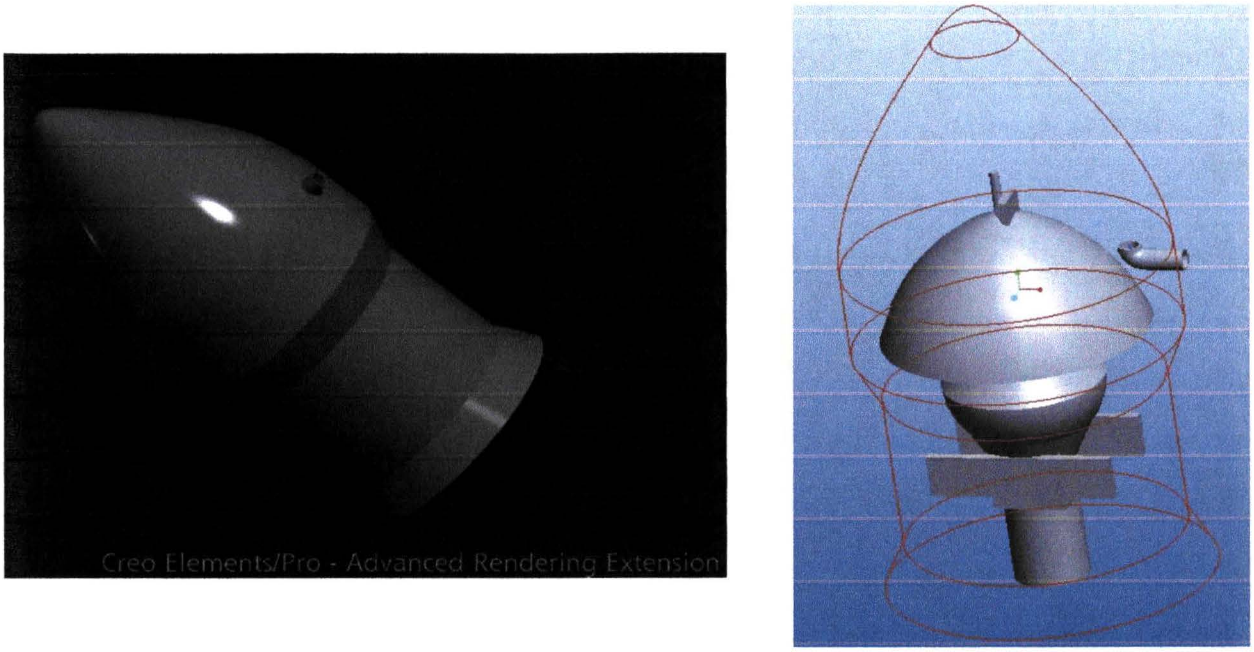


Figure 22 – Proposed CAD ECS Model with 3.5m Diameter Fairing

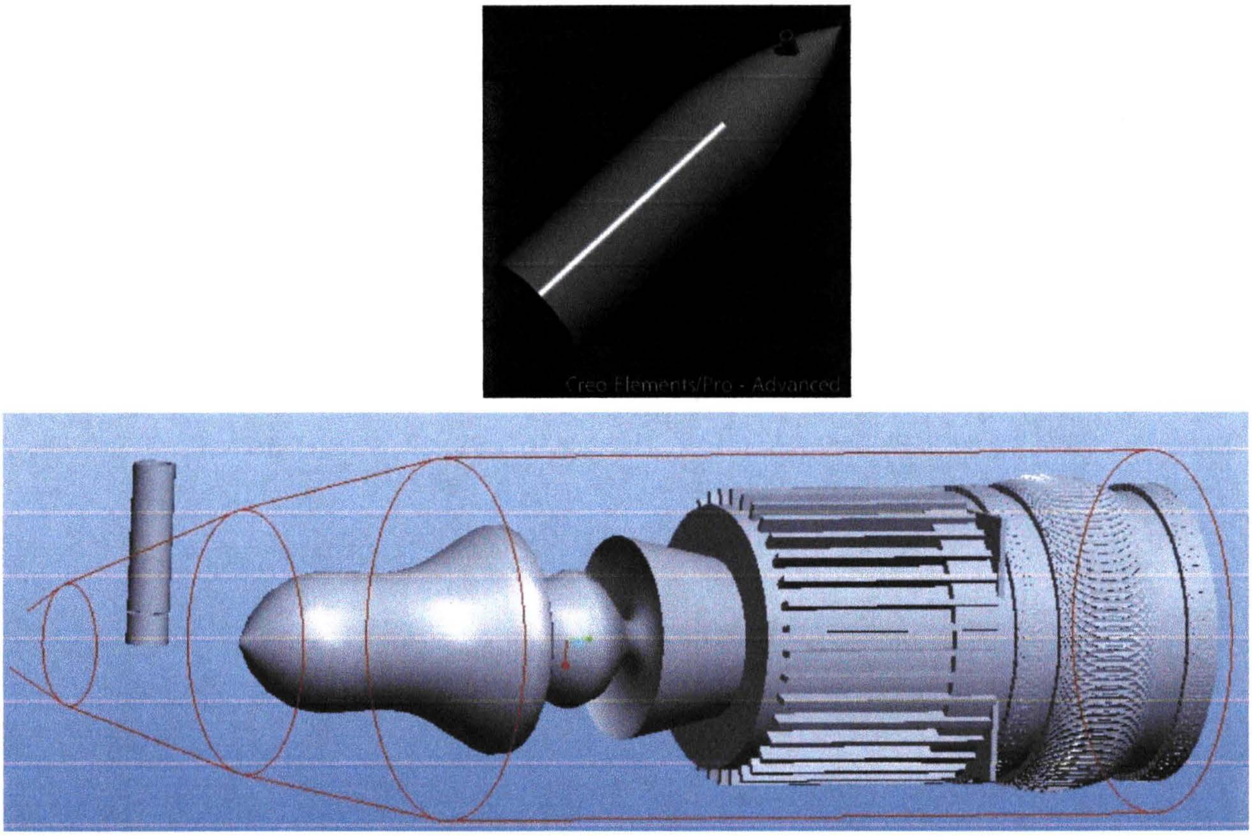


Figure 23 – Proposed CAD ECS Model with 5.5m Diameter Fairing

The CAD models can be translated into an iges file for ANSYS Workbench and ANSYS Fluent as shown in Figure 24. Figure 24 shows example CFD case for the proposed geometry. The contours are of velocity magnitude.

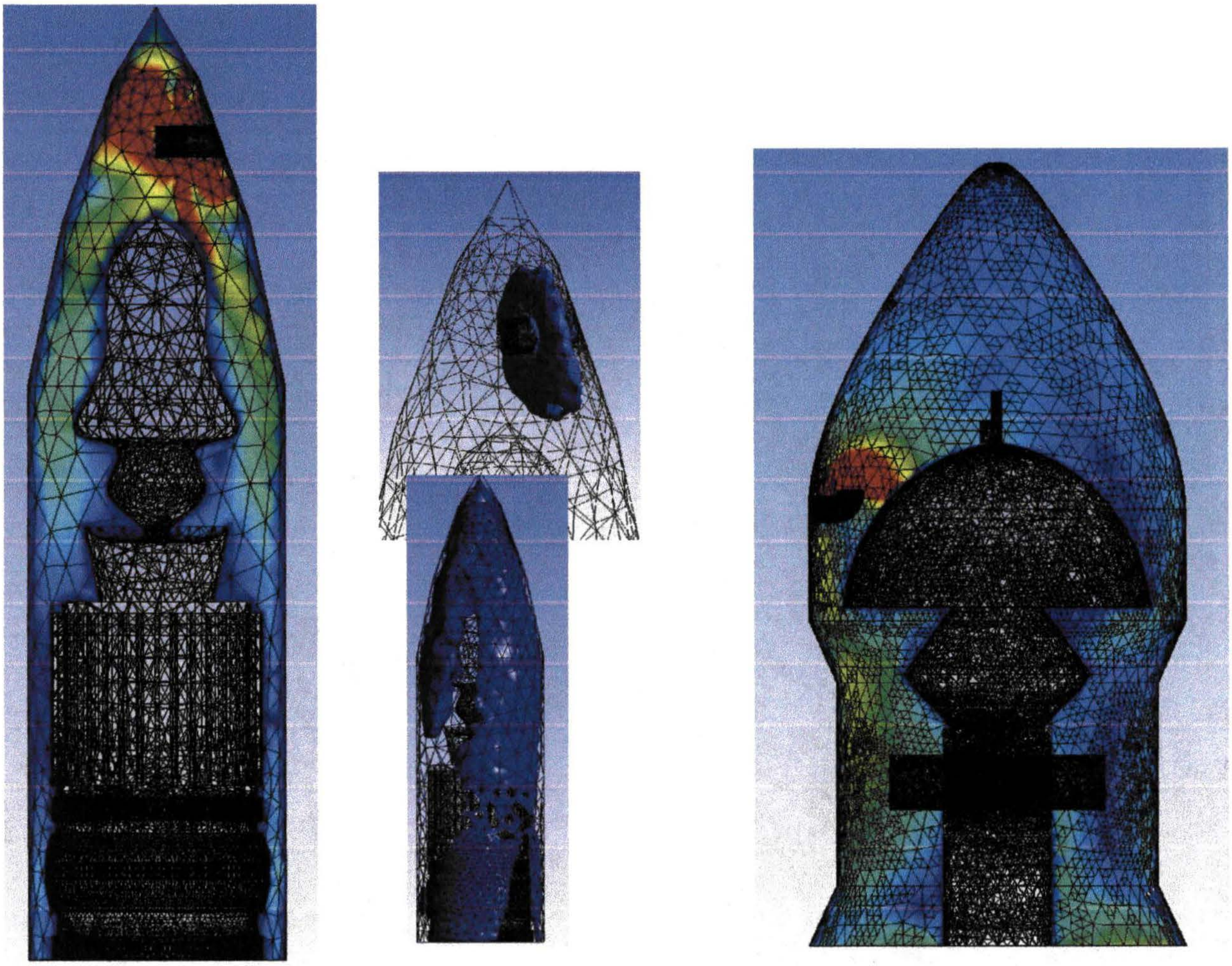


Figure 24 – Proposed FLUENT Modeling

In addition to using FLUENT, the solver OPENFOAM will be used as the primary solver. FLUENT will only be used to access the uncertainty of the solver. OPENFOAM is more versatile for the proposed research problem due to the open source code and no licensing issues. OPENFOAM additionally has the capability through snappy hexmesh to import the CAD as an

STL files and mesh the geometry. An example of the 3.5-meter proposed geometry is shown below. Figure 25 is the mesh and Figure 26 is the solution using SIMPLEFOAM.

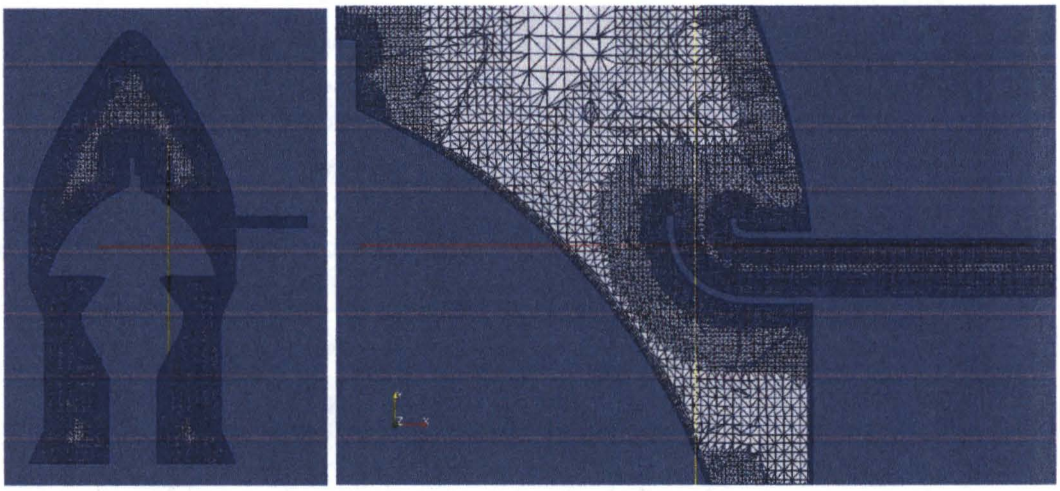


Figure 25 – Proposed OPENFOAM SnappyHEX Mesh Modeling

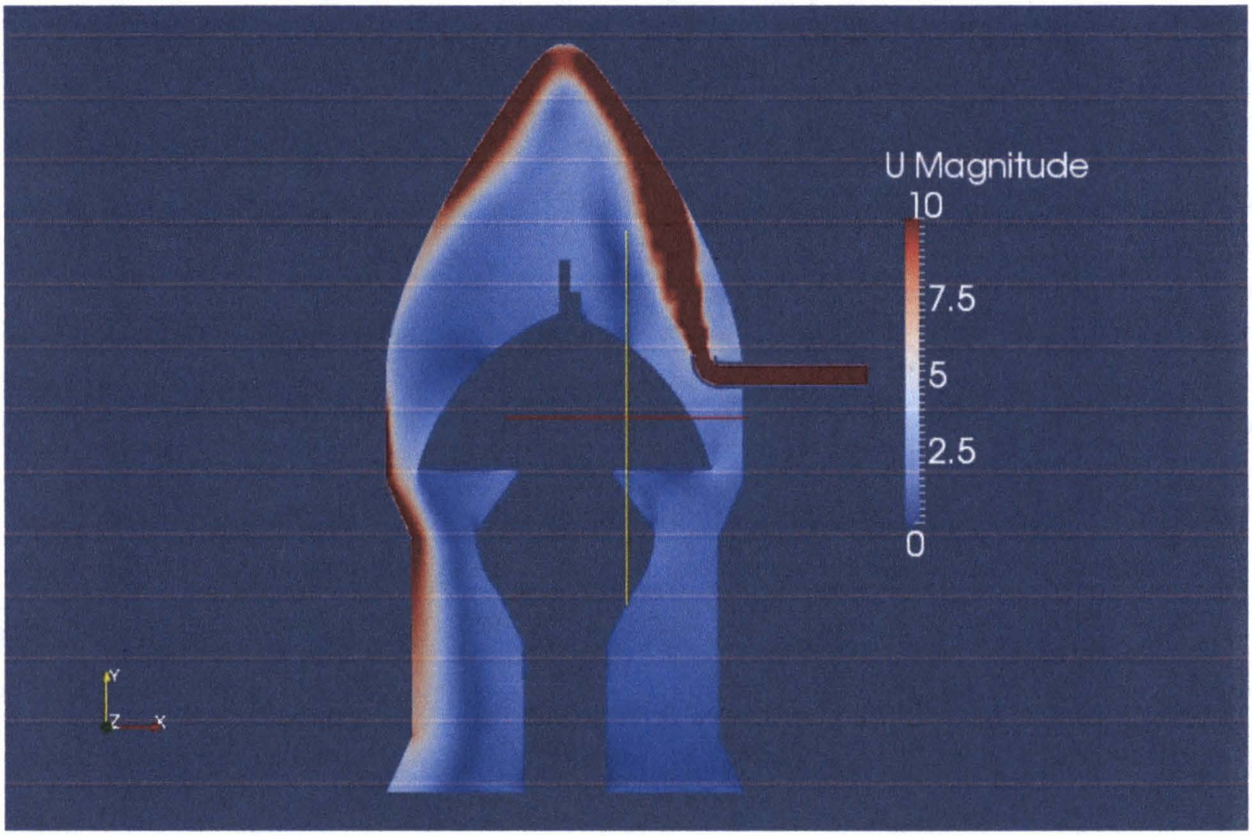


Figure 26 – Proposed OPENFOAM / SIMPLEFOAM Modeling

5.3 Proposed Method for Objective 2:

Objective: Perform an uncertainty analysis of the CFD model

The uncertainty of using CFD to analyze the ECS system for airflow velocities around spacecraft is unknown and not documented. The proposed objective 2 is to apply the methodology laid out in section 4 to the proposed problem. There are several items that will need to be addressed in the section. First, the interpolation scheme that was used for the backward step is unacceptable. The interpolation scheme is introducing errors that make using the monotonic (Richardson's extrapolation method) un-realistic. A better interpolation scheme between the three grids will be developed. Second, using the monotonic numerical results and extrapolating a solution for the entire computational domain will need to be assessed for feasibility. It may prove that using the grid as a separate oscillatory input parameter will suffice. To complete this objective a Comprehensive Approach to Verification and Validation of CFD Simulations – ASME Journal of Fluids Methodology outlined in previous section 4 will be used as a starting point and any inconsistencies or issues will be analyzed and solutions recommended. Again summarizing the method, three separate grids (rough, medium, fine) along with the uncertainty of all input parameters will be used to evaluate the uncertainty in the CFD velocity prediction. The velocity at every point in each of the three solutions will be compared to one another. An example of a single point in the domain is shown in figure 27.

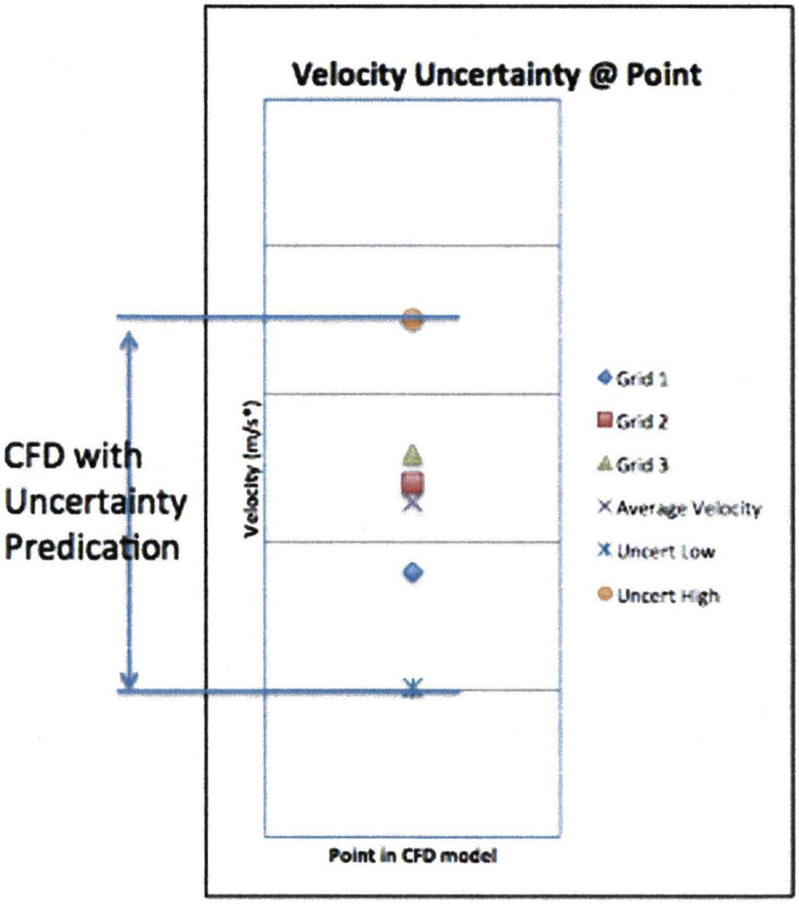


Figure 27 – Example CFD Uncertainty Prediction at a single point in the domain

5.4 Proposed Method for Objective 3:

Objective: Compile a table of uncertainty parameters

A table of uncertainty parameters will be constructed that could be used to estimate the uncertainty in a CFD model of an ECS/spacecraft. Each of the input uncertainties will be normalized in some way that proves the most convenient (by the average inlet velocity or local velocity magnitude). This normalization will be determined. By constructing this table, in the future analyst will be able to estimate the uncertainty by using the table verses running hundreds of CFD models. An example is shown in table 3. Table 3 comes from the backward facing step example provided earlier.

Table 3 – Compilation of Uncertainty Parameters

Type of Variable	Variables Xi	Value	Bias Error	Uncertainity
Boundary Conditions	epsilion turbulent mixing length dissipation rate inlet (m2/s3)	0.5	0.5	1.2% of local velocity
	k turbulent intensity kinetic energy inlet (m2/s2)	0.05	0.05	0.8 % of local velocity
	pressure outlet (Pa)	101325	2%	10x the variation
	velocity inlet (m/s)	10	0.5	1.3x the variation
Fluid Properties	kinematic viscosity nu represents air [0-50-100] deg C	1.79E-06	[13.6e-06 -> 23.06e-06]	28% of the local velocity
Grid Size	Method - Uses Oscillatory Uncertainty	1,192,000 1,862,500 3,311,689		grid specific
Numerical	Method - Uses Richardson's Extrapolation (ASME 5 Step Procedure) – Calculated for Velocity at each Cell			
Solver	OpenFOAM (SimpleFoam) vs. Fluent			30% of the local velocity
Turbulence Models	ke-realiable, kwSST, and SpalartAllmaras			Future work will consider more turbulence models

5.5 Proposed Schedule and Project Management:

The candidate will complete the project management task. The candidate will schedule all tasks required to complete the objectives and is responsible for the successful completion of the project. A preliminary schedule is shown in below. All models will be constructed using the methodology described in the proposed methods section. All models will be run on the NASA KSC servers.

The following is a list of the proposed schedule:

Candidacy Approval ....Spring 2013
Objective 1 Completed..Summer 2013
Objective 2 Completed ....Fall 2013
Objective 3 Completed ....February 2014
Dissertation Completed...March 2014
Defense Completed..April 2014
Graduation. ....May 2014

5.6 Proposed Deliverables:

The following is a list of deliverables and estimated schedule that will be provided to the board for review and comment:

Dissertation Chapters (Problem, Literature Review, Methods, Objective 1)……….August 2013

Dissertation (Updated Previous Chapters + Objective 2 Results) ……………….....January 2014

Dissertation (Updated Previous Chapters + Objective 3 Results) ……………….....February 2014

Dissertation Completed Draft (Update all Chapters) ………………………………March 2014

Final Dissertation Completed …………………………………………………………April 2014

5.7 Proposed Publication Schedule:

1) Literature Review / State of the Art CFD Uncertainty Analysis / Example Method Backward Step. …Completed (1/2013) – AIAA-2013-0258

2) Objective 1 Results and Turbulence Uncertainty Term ……………………..November 2013

The student is targeting the 66th Annual Meeting of the APS Division of Fluid Dynamics in Pittsburgh PA, November 24-26, 2013.

3) Objective 2 Results…………………………………………………………………..January 2014

The student is targeting the 52nd AIAA Aerosciences Meeting in National Harbor, MD, January 6-9, 2014.

4) Each of the Publications 1-3 will be submitted to their corresponding journal for consideration for journal publication.

6.0 – Expected Contribution

The project described has extensive uses in research, government, industry, and education. The main problem with today's research is it is generally numerically based. Testing is extremely expensive and numerical models provide a cheaper, faster way to provide adequate results. Researchers, government, industry, and students use computer models extensively to perform requirement verifications. George Box stated, " All models are wrong but some are useful". The research proposed will quantify the uncertainty in the CFD model for ECS / spacecraft systems. The ASME has published an industry standard to quantify this uncertainty, but there is limited validation in the literature. This method has potential to be extended to any numerically based simulation.

a) Demonstrate a CFD Uncertainty Analysis for 3-D, low speed, incompressible, highly turbulent, internal flow can be calculated for an entire simulation domain

b) Develop a higher order interpolation scheme to be used for grid interpolations and uncertainty quantification

c) Investigate the applicability of using the ASME 5-Step procedure for the entire computational domain to estimate numerical uncertainties

d) Calculate the uncertainty in using different turbulent models

e) Demonstrate this method can contribute to the study of importance of input parameters in CFD

f) Compile a table for uncertainty estimates by input parameter. The table will benefit the community by providing an uncertainty estimate in lieu of running hundreds of CFD simulations

g) Demonstrate the ability to use OPENFOAM to calculate the velocity field of an Environmental Control System

h) Compare the results of OPENFOAM verses an industry standard CFD software program (ie FLUENT).

References

[1] Roache, P. J., Ghia, K. N., and White, F. M., 1986, “Editorial Policy Statement on the Control of Numerical Accuracy,” ASME JOURNAL OF FLUIDS ENGINEERING, 108, p.2.

[2] Stern, F., Wilson, R. V., Coleman, H. W., and Paterson, E. G., "Verification and Validation of CFD Simulations," Iowa Institute of Hydraulic Research Report No. 407, September 1999.

[3] Iudicello, F., “Guide for the verification and validation of computational fluid dynamics simulations,” AIAA G-077-1998, 1998.

[4] Slater, J., “Uncertainty and Error in CFD Applications”, NPARC Alliance CFD Verification and Validation Web Site,

July 2008. < http://www.grc.nasa.gov/WWW/wind/valid/tutorial/errors.html>

[5] Celik, I., Karatekin, O., “Numerical Experiments on Application of Richardso Extrapolation With Nonuniform Grids,” ASME Journal of Fluids Engineering, 119, Issue 3, p584.

[6] Stern, F., Wilson, R. V., Coleman, H. W., and Paterson, E. G., 2001, "Comprehensive Approach to Verification anD Validation of CFD Simulations - Part 1: Methodology and Procedures," ASME Journal of Fluids Engineering, 123, pp. 793-802.

[7] Roache, P.J., "Verification and Validation in Computational Science and Engineering," Hermosa Publishers, 1998.

[8] Celik, I., and Zhang, W., "Calculation of Numerical Uncertainty Using Richardson Extrapolation: Application to Some Simple Turbulent Flow Calculations," American Society of Mechanical Engineers Journal of Fluid Mechanics, 1995.

[9] Stern, F., Wilson, R. V., Coleman, H. W., and Paterson, E. G., "Verification and Validation of CFD Simulations," Iowa Institute of Hydraulic Research Report No. 407, September 1999.

[10] Stern, F., Wilson, R. V., Coleman, H. W., and Paterson, E. G., 2001, "Comprehensive Approach to Verification and Validation of CFD Simulations - Part 2: Application for Rans Simulation of a Cargo/Container Ship," ASME Journal of Fluids Engineering, 123, pp. 803-810.

[11] "Uncertainity Analysis in CFD Verification and Validation Methodology and Procedures," International Towing Tank Conference – Recommended Procedures and Guidelines, September 2008.

[12] Celik, I., Ghia. U., Roache, P., Freitas, C., Coleman, H., Raad, P., "Procedure for Estimating and Reporting of Uncertainity Due to Discretization in CFD Applications," American Society of Mechanical Enigneers Journal of Fluids Engineering, Vol. 130, July 2008.

[13] Celik, I., "Critical Issues With Quantification of Discretization Uncertainty in CFD," National Energy Technology Laboratory Workshop on Multiphase Flow Science, August 2011.

[14] An American National Standard., "Standard for Verification and Validation in Computational Fluid Dynamics and Heat Transfer", The American Society of Mechanical Engineers. ASME V&V 20-2009.

[15] Kandula, M., Hammad, K., and Schallhorn, P., "CFD Validation with LDV Test Data for Payload/Fairing Internal Flow," AIAA-2005-4910, 2005.

[16] Kandula, M., and Walls, L., "An application of overset grids to payload/fairing internal flow CFD analysis," 20th AIAA Applied Aerodynamics Conference, AIAA-2002-3253, June 2002.

[17] Groves, C., Ilie, M., Schallhorn, Paul. "Comprehensive Approach to Verification and Validation of CFD Simulations Applied to Backward Facing Step," AIAA-2013-0258, 2013.

[18] Kandula, M., Hammad, K., and Schallhorn, P., "CFD Validation with LDV Test Data for Payload/Fairing Internal Flow," AIAA-2005-4910, 2005.

[19] Kennedy Space Center, Launch Services Program

<http://www.nasa.gov/centers/kennedy/launchingrockets/> (1-28-2013).

[20] Delta II Payload Planners Guide.

<http://www.ulalaunch.com/site/docs/product_cards/guides/DeltaIIPayloadPlannersGuide2007.pdf> (1-28-2013).

[21] Delta IV Payload Planners Guide.

<http://spacecraft.ssl.umd.edu/design_lib/Delta4.pl.guide.pdf> (1-28-2013).

[22] Atlas 5 Payload Planners Guide.

< http://spacecraft.ssl.umd.edu/design_lib/Atlas5.pl.guide.pdf> (1-28-2013).

[23] Pegasus Users Guide.

< http://www.orbital.com/NewsInfo/Publications/Pegasus_UG.pdf> (1-28-2013)

[24] Taurus Users Guide.

< http://www.orbital.com/NewsInfo/Publications/taurus-user-guide.pdf> (1-28-2013)

[25] Space X Falcon 9 Users Guide.

< http://www.spacex.com/Falcon9UsersGuide_2009.pdf> (1-28-2013)

----I SINCERELY THANK YOU FOR YOUR TIME AND CONSIDERATION