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

Candidacy Exam / Proposed Dissertation

Curtis Groves
Ph. D. Student Mechanical Engineering
University of Central Florida

Agenda

- ECS System Overview / Problem
- Literature Review
- Summary of ASME Standard
- Example of Backward Facing Step
- Proposed Dissertation Topic
- Summary

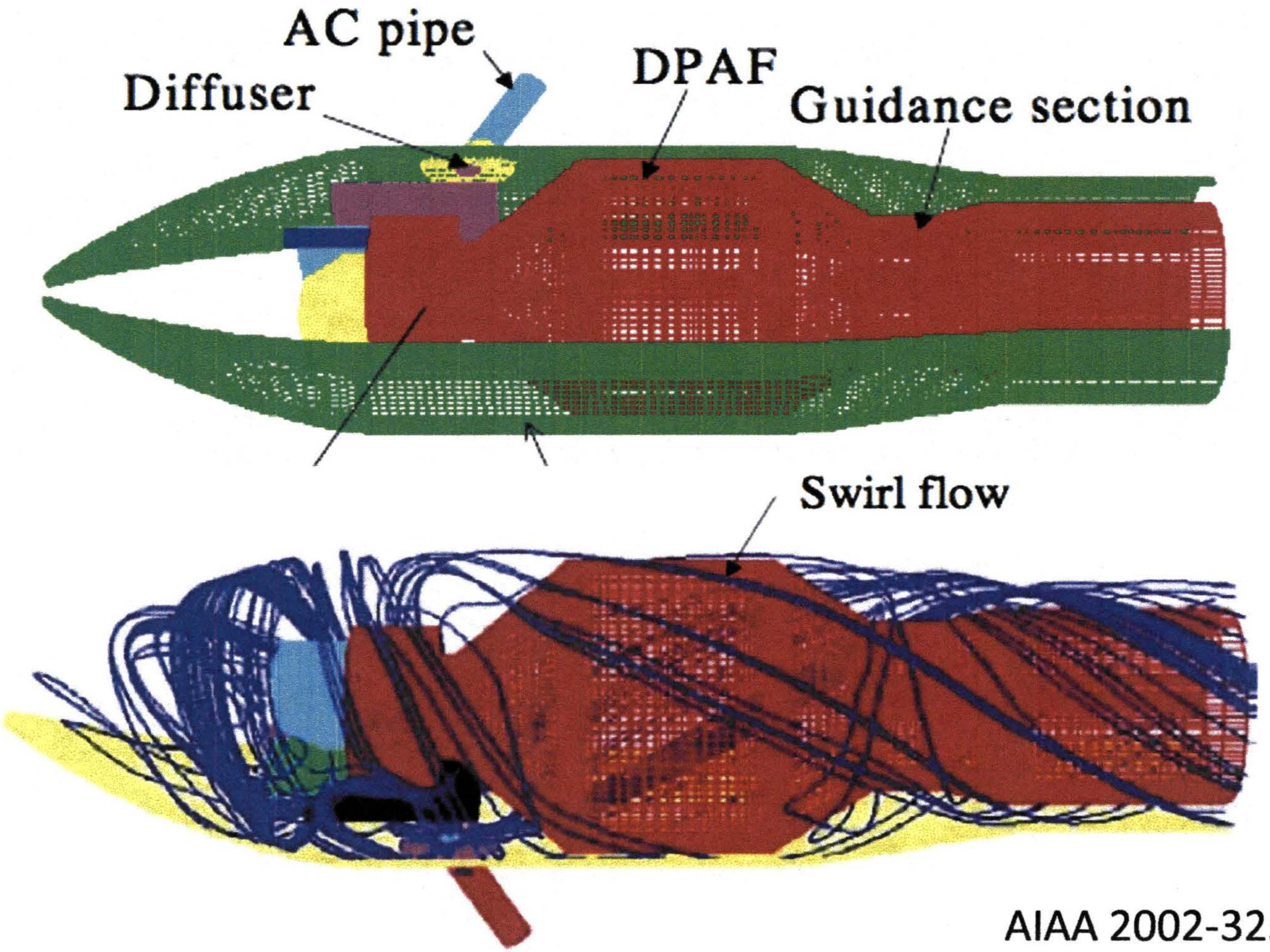
Environmental Control System (ECS)
System Overview

ECS System Overview

- Environmental Control System AIAA 2002-3253
  - Prior to launch, cold air (air conditioning) flows downward around the spacecraft after it has been encapsulated in the Payload Fairing.
  - 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
  - After passing over the spacecraft, it is finally discharged through vents
  - The Payload Fairing air conditioning is cut off at lift off.

ECS System Overview

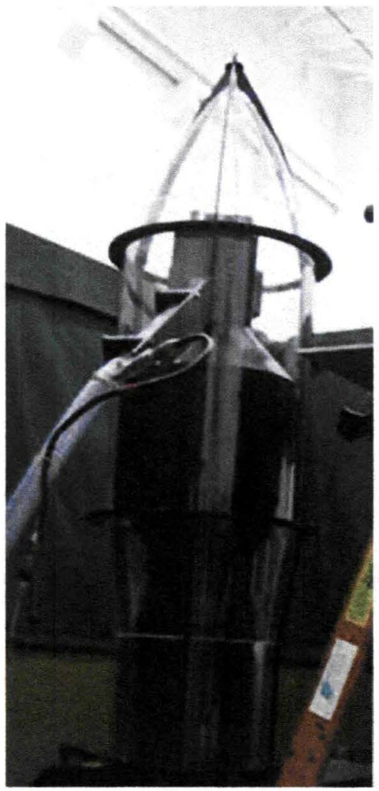
- Environmental Control System (ECS)

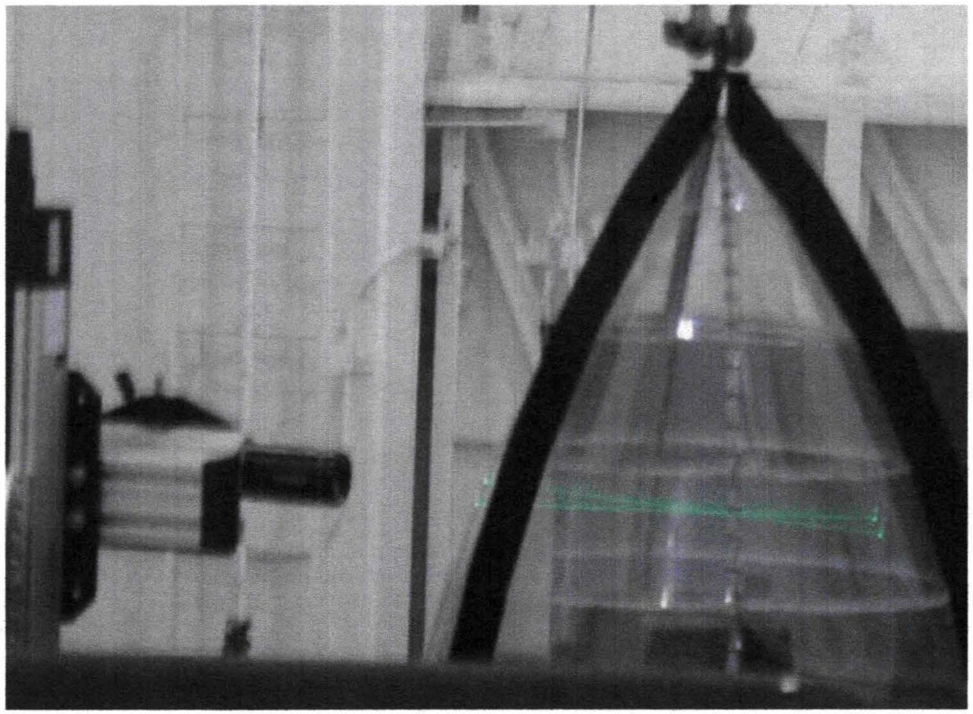


AIAA 2002-3253

ECS System Overview

- Example of an ECS system airflow test

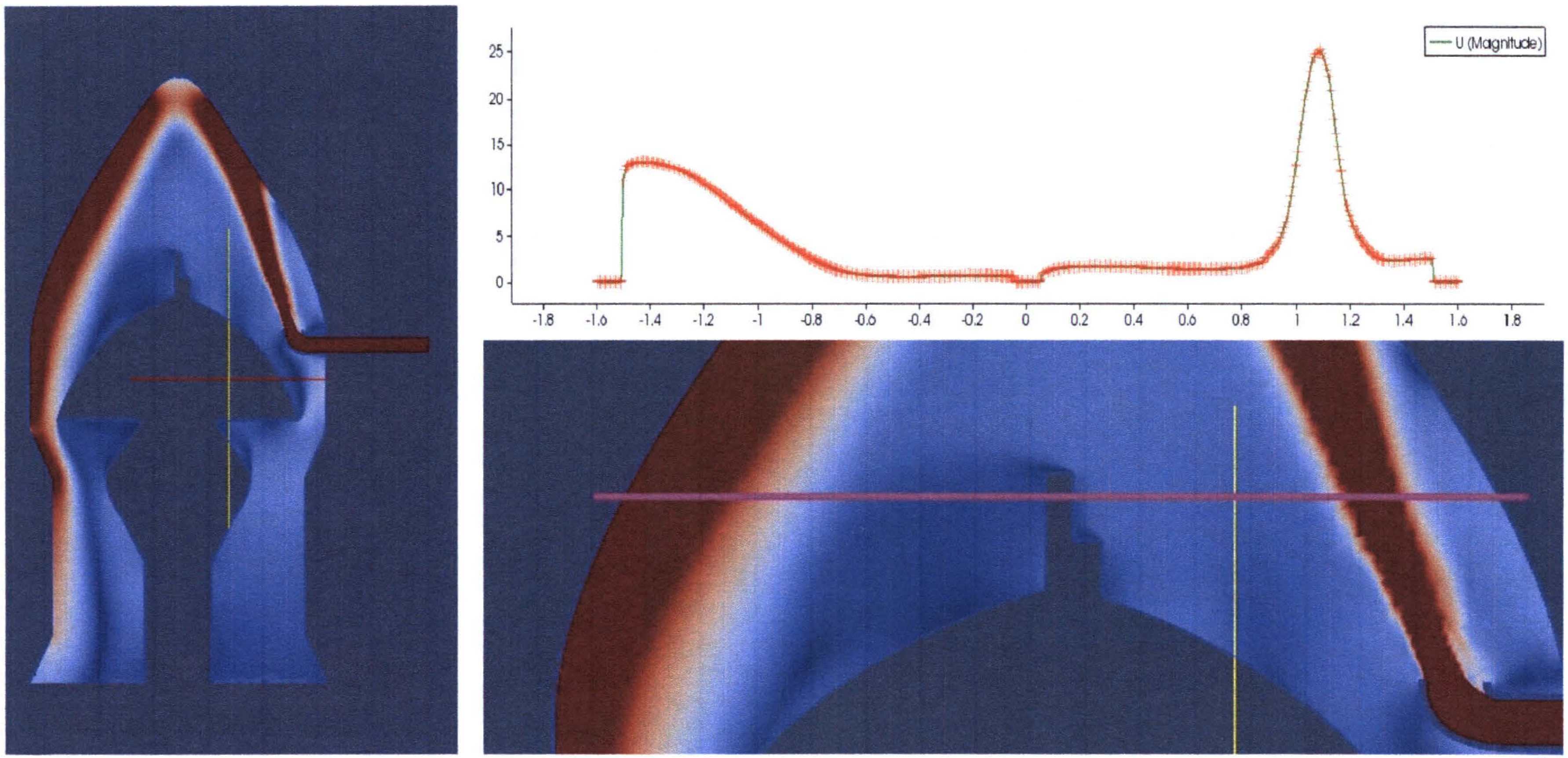




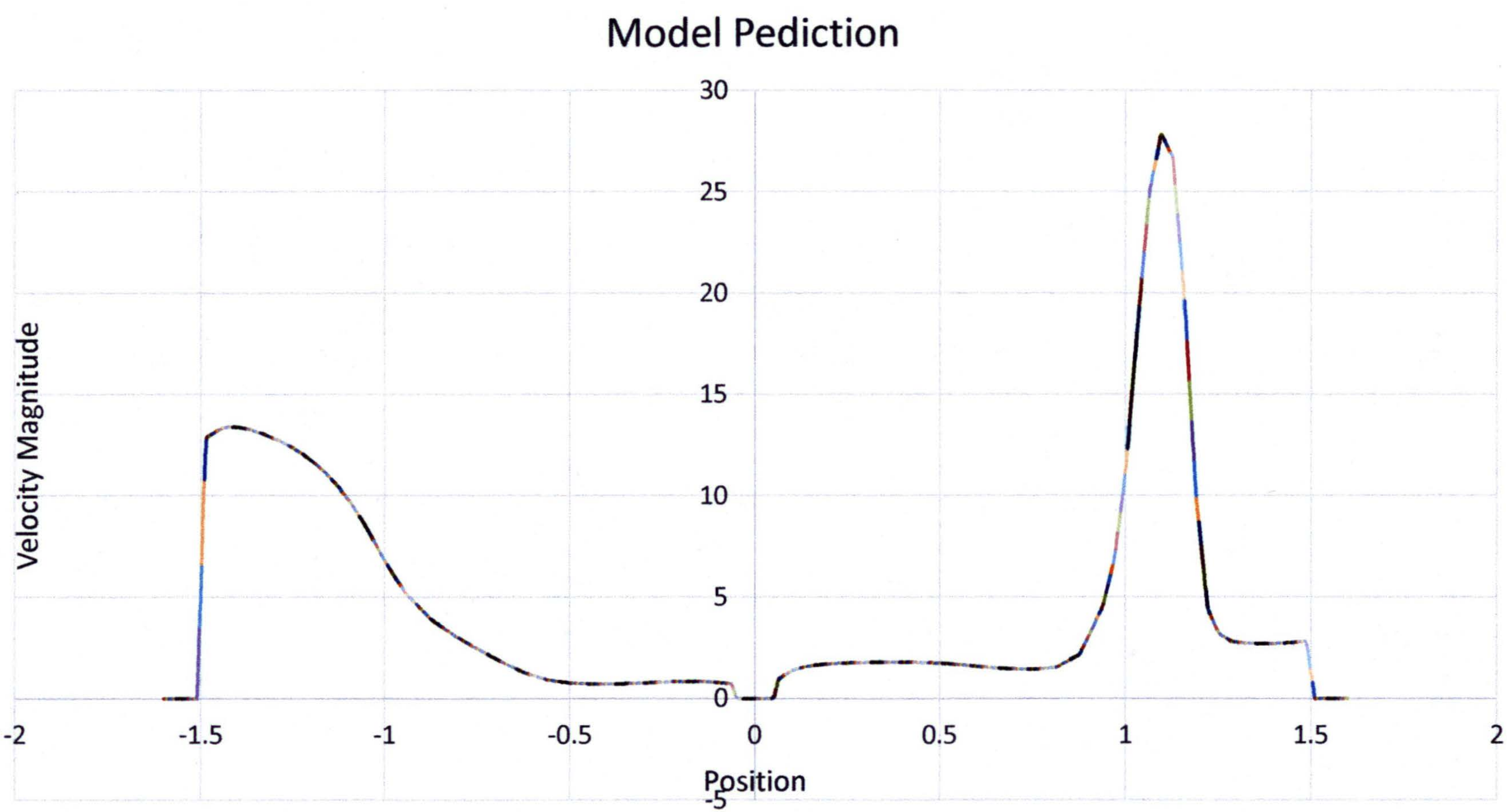
Kandula, M., Hammad, K., and Schallhorn, P., “CFD Validation with LDV Test Data for Payload/Fairing Internal Flow,” AIAA-2005-4910, 2005.

Problem, Example

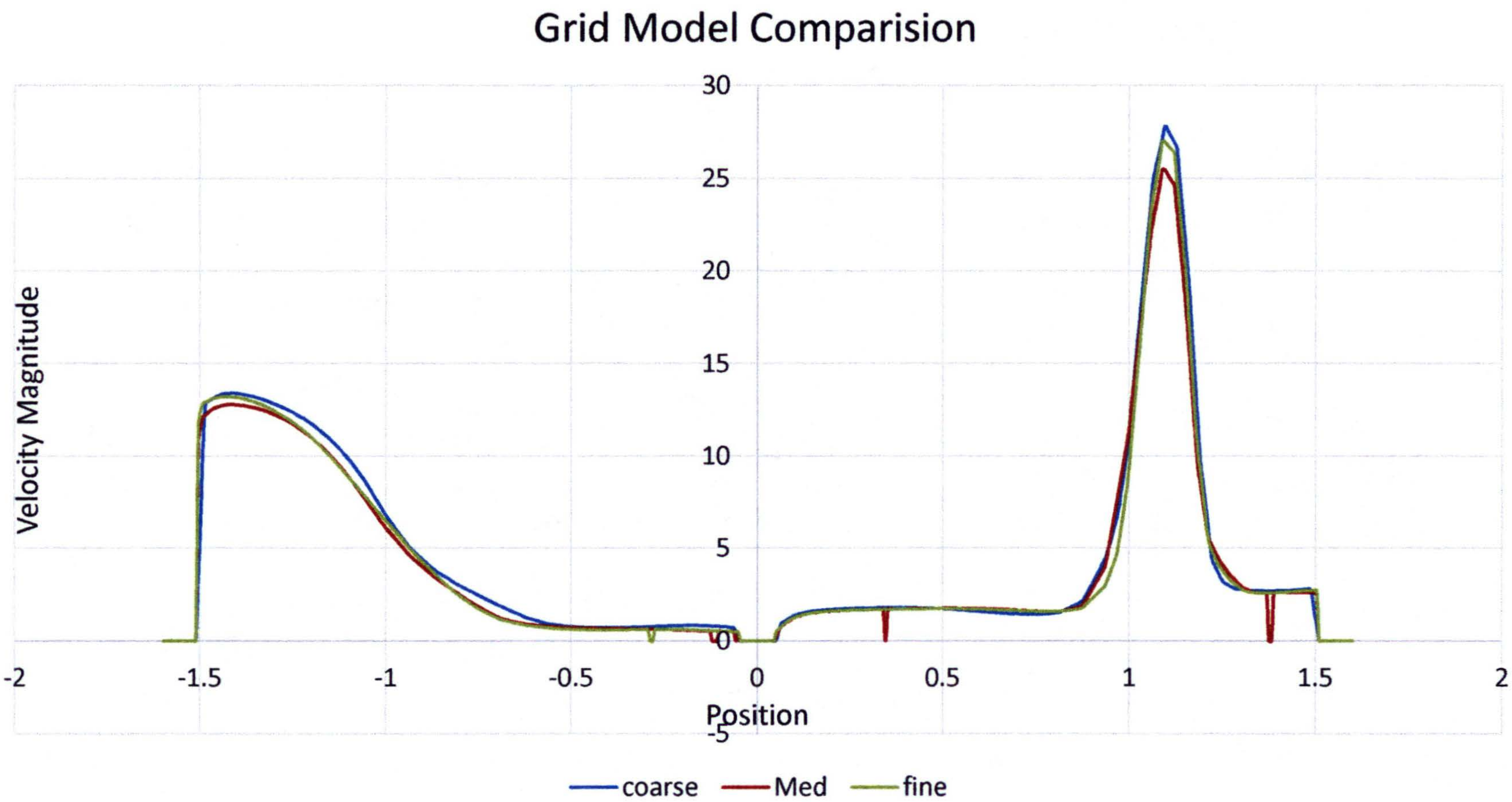
- CFD has errors and uncertainties



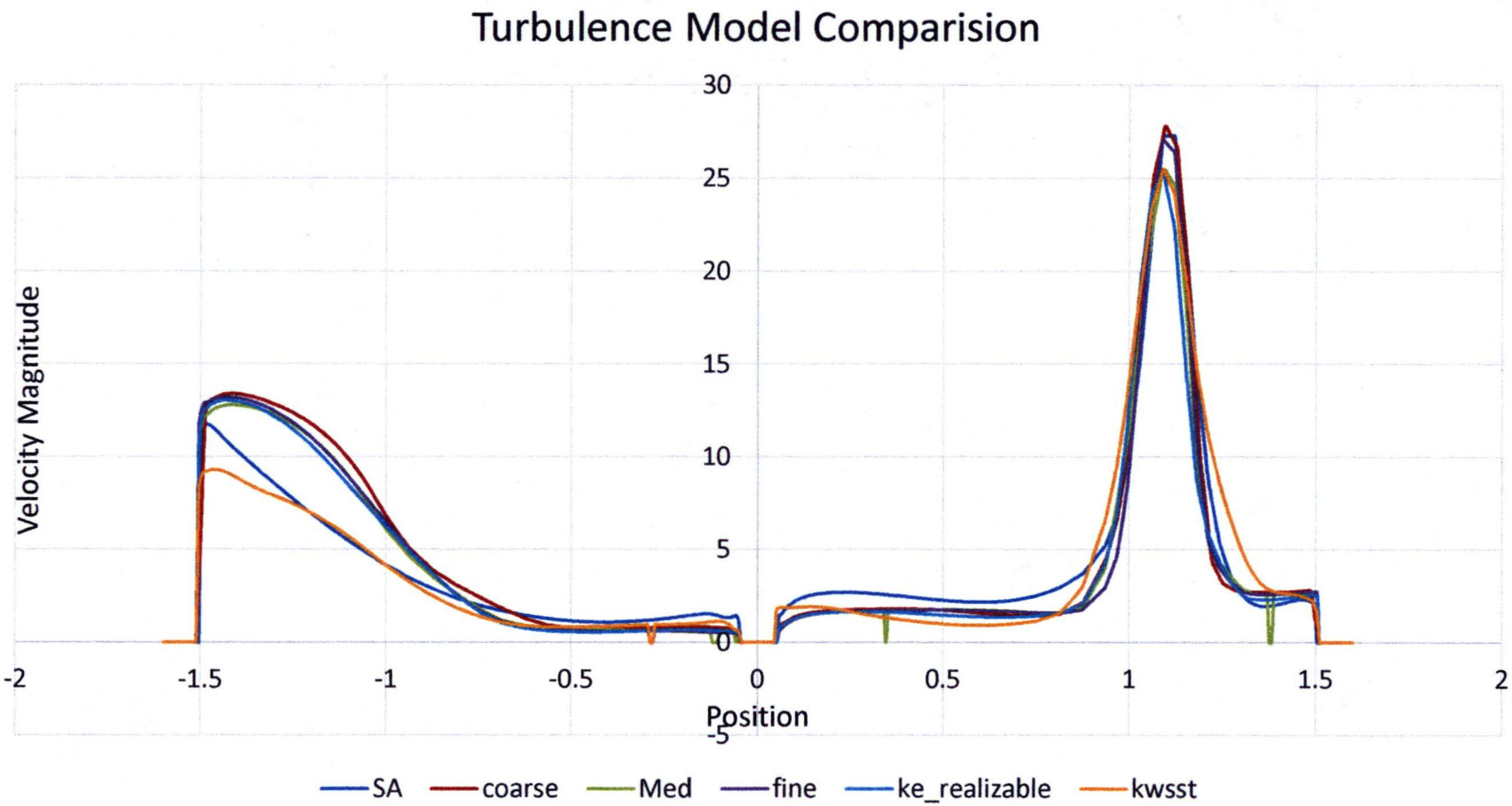
Problem, Example – How Good is this Prediction?



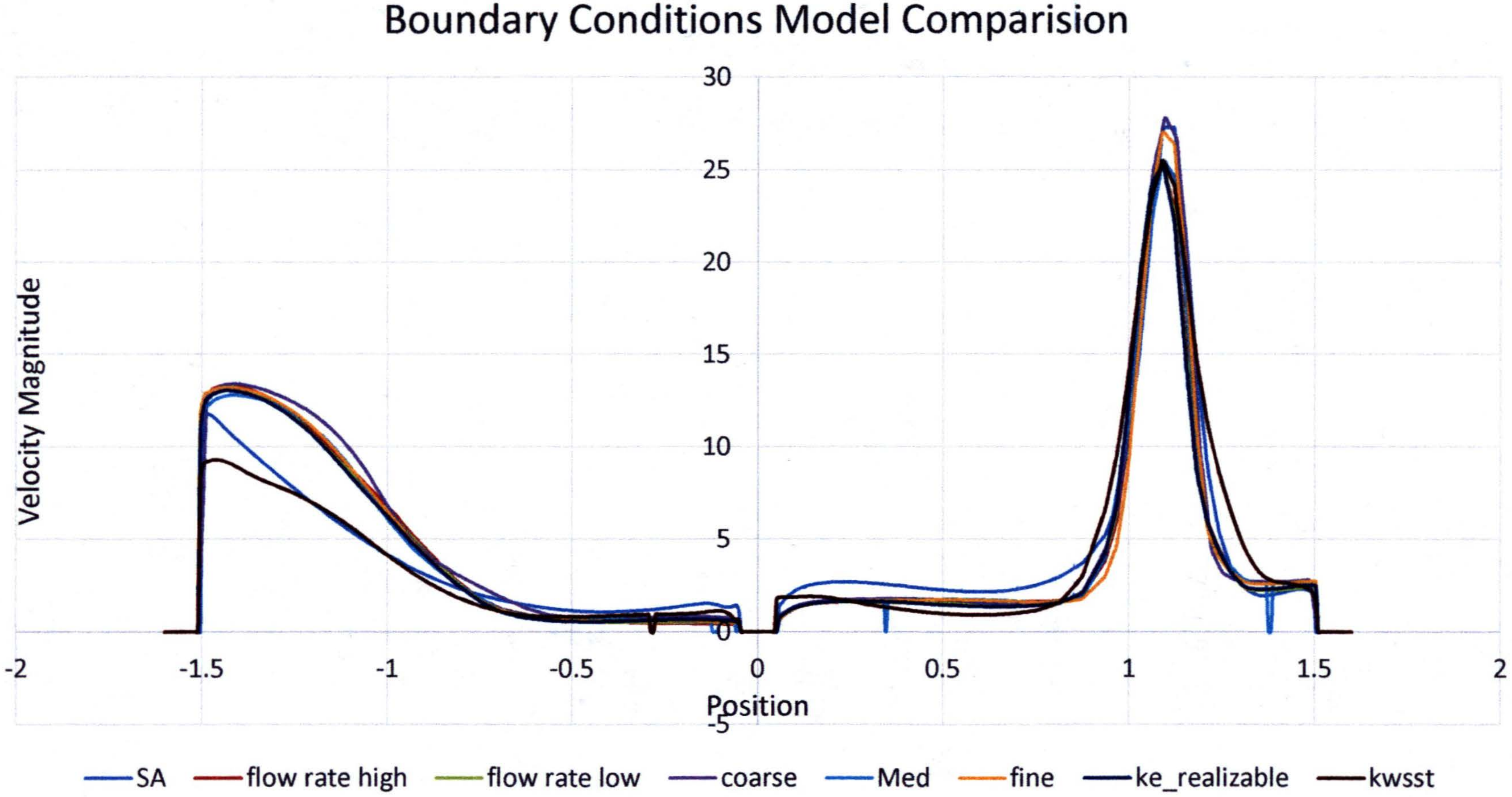
Problem, Example – Grid Independence Study



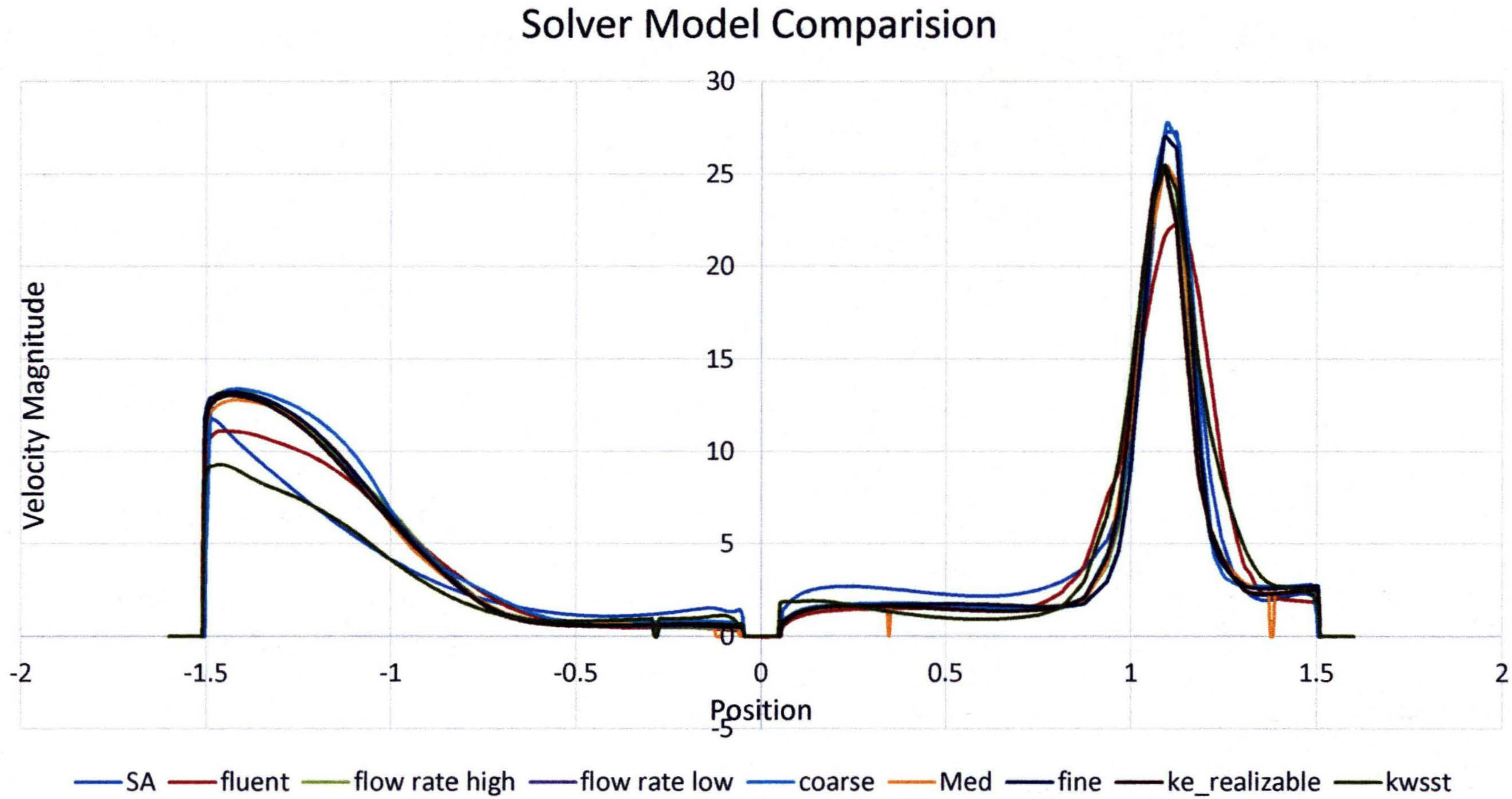
Problem, Example – Turbulence Models



Problem, Example – Boundary Conditions



Problem, Example – Solver



Literature Review

Papers Reviewed

- [1] Iudicello, F., "Guide for the verification and validation of computational fluid dynamics simulations," AIAA G-077-1998, 1998.
- [2] 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.
- [3] Roache, P.J., "Verification and Validation in Computational Science and Engineering," Hermosa Publishers, 1998.
- [4] 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.
- [5] 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.
- [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] 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.
- [8] "Uncertainity Analysis in CFD Verification and Validation Methodology and Procedures," International Towing Tank Conference – Recommended Procedures and Guidelines, September 2008.
- [9] 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.
- [10] Celik, I., "Critical Issues With Quantification of Discretization Uncertainty in CFD," National Energy Technology Laboratory Workshop on Multiphase Flow Science, August 2011.
- [11] "Standard for Verification and Validation in Computational Fluid Dynamics and Heat Transfer," American Society of Mechanical Engineers, ASME V&V 20-2009.

Literature Review

- CFD is extensively used in industry, government, and academia to design, investigate, operate, and improve understanding of fluid physics [1].
- 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 [1].
- One needs to evaluate the uncertainty in the results of a CFD simulation to postulate a level of credibility.

Literature Review

- 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 [2].
- Other journals have issued similar statements [3].
- 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 [4].
  - In 1997, Roache published "Quantification of Uncertainty in Computational Fluid Dynamics" [3]. Roaches research also used the Richardson Extrapolation method to quantify CFD uncertainties.

Literature Review

- In 1998, the AIAA has published a "Guide for the Verification and Validation of Computational Fluid Dynamics Simulations" [1].
- This document provides guidelines for assessing credibility via verification and validation [1].
- The document does not recommend standards due to issues not yet resolved, but defines several terms [1].
  - "Uncertainty is defined as a potential deficiency in any phase or activity of the modeling process that is due to lack of knowledge [1]."
  - "Error is defined as a recognizable deficiency in any phase or activity of modeling and simulation that is not due to lack of knowledge [1]."
  - "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 [1]."
  - Uncertainty and error are normally linked to accuracy in modeling and simulation [1].
- The guide defines four predominate error sources:
  - insufficient spatial discretization convergence
  - insufficient temporal discretization convergence
  - lack of iterative convergence, and computer programming,
- The guide emphasizes that systematically refining the grid size and time step is the most important activity in verification [1].
- The guide has outlined the terms and an overall structure to performing validation, but does not offer a quantitative method.

Literature Review

- 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" [5].
- 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 [7] and used the methodology documented in IIHR Report 407.
- Numerical errors and uncertainties in CFD can be estimated using iterative and parameter convergence studies [5].
- 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 [5].
- The literature provides an approach for estimating errors and uncertainties in CFD simulations for each of the three cases [6, 7, 5].

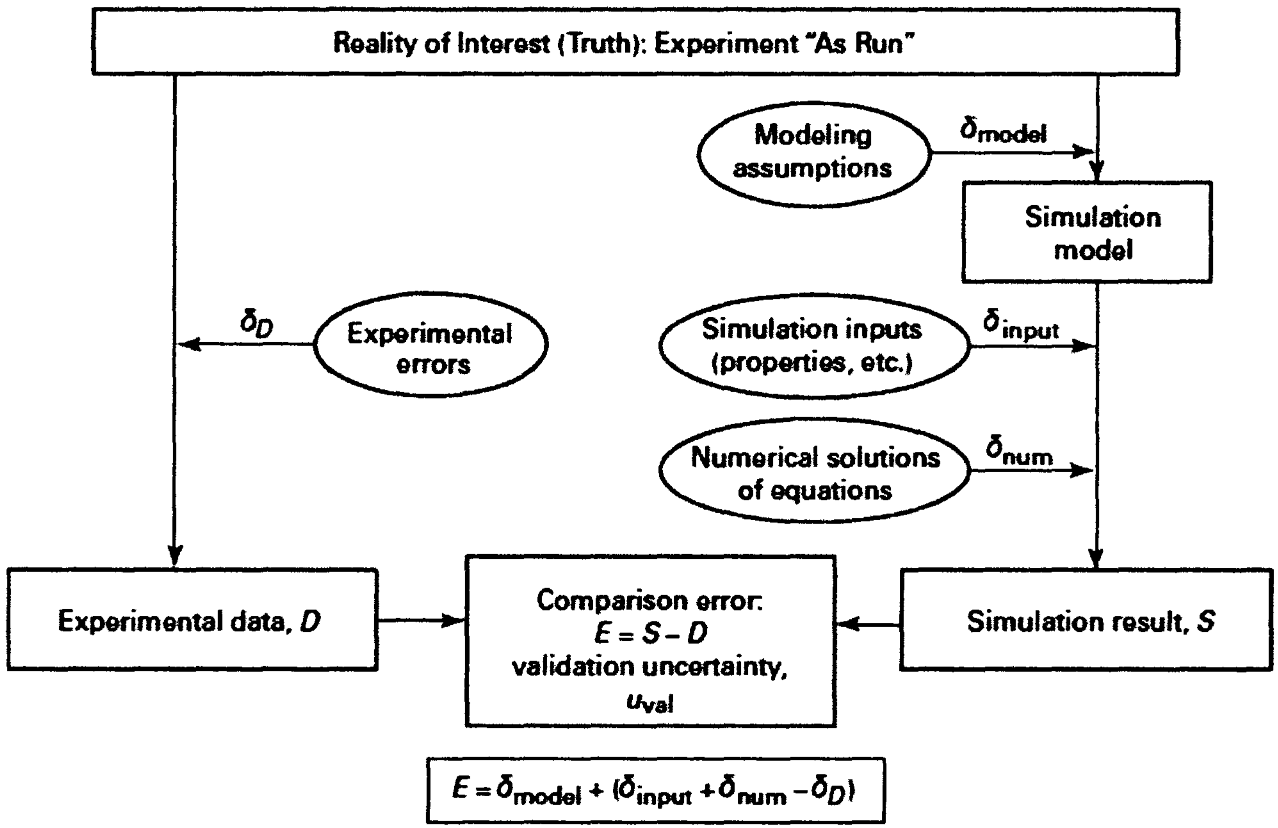
Literature Review

- In 2008, the International Towing Tank Conference (ITTC) has published "Recommended Procedures and Guidelines – Uncertainty Analysis in CFD Verification and Validation Methodology and Procedures" [8].
  - 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" [8].
- Also in 2008, the ASME Journal of Fluids Engineering published a "Procedure for Estimating and Reporting of Uncertainty Due to Discretization in CFD Applications" [9].
- In 2009, the American Society of Mechanical Engineers published "Standard for Verification and Validation in Computational Fluid Dynamics and Heat Transfer"[11].
  - This standard follows the same approach outlined in the previous Literature and defines the following procedure for estimating uncertainty.
  - This is the current "State of the Art"

Summary of ASME Standard

"Standard for Verification and Validation in Computational Fluid Dynamics and Heat Transfer"[11]

Overview of the Validation Process



Approach

- Estimate Interval within which δ_{model} falls with a given degree of confidence
  - Assume Gaussian normal distribution, 90 % confidence
    - $U_{90\%} = {}_{+/-}1.65*(U_{val})$
- Error Sources (U_{num}, U_{input}, U_D)

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

- Uncertainty Equation

$$\boxed{U_{90\%} = +/-1.65 * \sqrt{{U_{num}}^2 + {U_{input}}^2 - {U_D}^2}}$$

Numerical Uncertainty, U_{num}

- 5 Step Procedure for Uncertainty Estimation
  - Step 1: Representative Grid Size

For nonstructured grids one can define

$$h = \left[\left(\sum_{i=1}^{N} \Delta V_i \right) / N \right]^{1/3} \qquad (2\text{-}4\text{-}4)$$

where

N = total number of cells used for the computations

ΔV_i = volume of the i^{th} cell [4]

$$h_1 < h_2 < h_3$$

Numerical Uncertainty, U_{num} (continued)

- Step 2: Select 3 significantly (r>1.3) different grid sizes

$$r_{21} = h_2/h_1$$

$$r_{32} = h_3/h_2$$

- Use CFD Simulation to analyze key variables, φ

$$\varepsilon_{32} = \varphi_3 - \varphi_2$$

$$\varepsilon_{21} = \varphi_2 - \varphi_1$$

Numerical Uncertainty, U_{num} (continued)

– Step 3: Calculate observed order, p

$$p = [1/\ln(r_{21})][\ln|\varepsilon_{32}/\varepsilon_{21}| + q(p)]$$

$$q(p) = \ln\left(\frac{r_{21}^{p} - s}{r_{32}^{p} - s}\right)$$

$$s = 1 \cdot \mathrm{sign}(\varepsilon_{32}/\varepsilon_{21})$$

Numerical Uncertainty, U_{num} (continued)

– Step 4: Calculate extrapolated values

$$\varphi_{ext}^{21} = \left(r_{21}^{p}\varphi_1 - \varphi_2\right) / \left(r_{21}^{p} - 1\right)$$

$$e_a^{21} = \left|\frac{\varphi_1 - \varphi_2}{\varphi_1}\right|$$

Numerical Uncertainty, U_{num} (continued)

– Step 5: Calculate Fine Grid Convergence Index & Numerical Uncertainty

$$\mathrm{GCI}_{\mathrm{fine}}^{21} = \frac{Fs \cdot e_a^{21}}{r_{21}^p - 1}$$

The Factor of Safety, $Fs = 1.25$

– Assumption that the distribution is Gaussian about the fine grid, 90% Confidence

$$U_{num} = GCI / 1.65$$

Input Uncertainty, U_{input}

- Input error is based on a Taylor Series expansion in parameter space

$$u_{\text{input}}^2 = \sum_{i=1}^{n} \left(\frac{\partial S}{\partial X_i} u_{X_i} \right)^2 \qquad (3\text{-}2\text{-}1)$$

where

S = simulation result

u_{X_i} = corresponding standard uncertainty in input parameter X_i

X_i = input parameter

Compute Uncertainty

– Uncertainty

$$U_{90\%} = +/-1.65 * \sqrt{U_{num}^{2} + U_{input}^{2} - U_{D}^{2}}$$

Backward Facing Step Example

AIAA-2013-0258

Comprehensive Approach to Verification and Validation of CFD Simulations Applied to Backward Facing Step – Application of CFD Uncertainty Analysis

Curtis E. Groves and Marcel Ilie, PhD
University of Central Florida, Orlando, FL 32816

Paul A. Schallhorn, PhD
National Aeronautics and Space Administration, Kennedy Space Center, FL, 32899

Background

- **How good is CFD?**

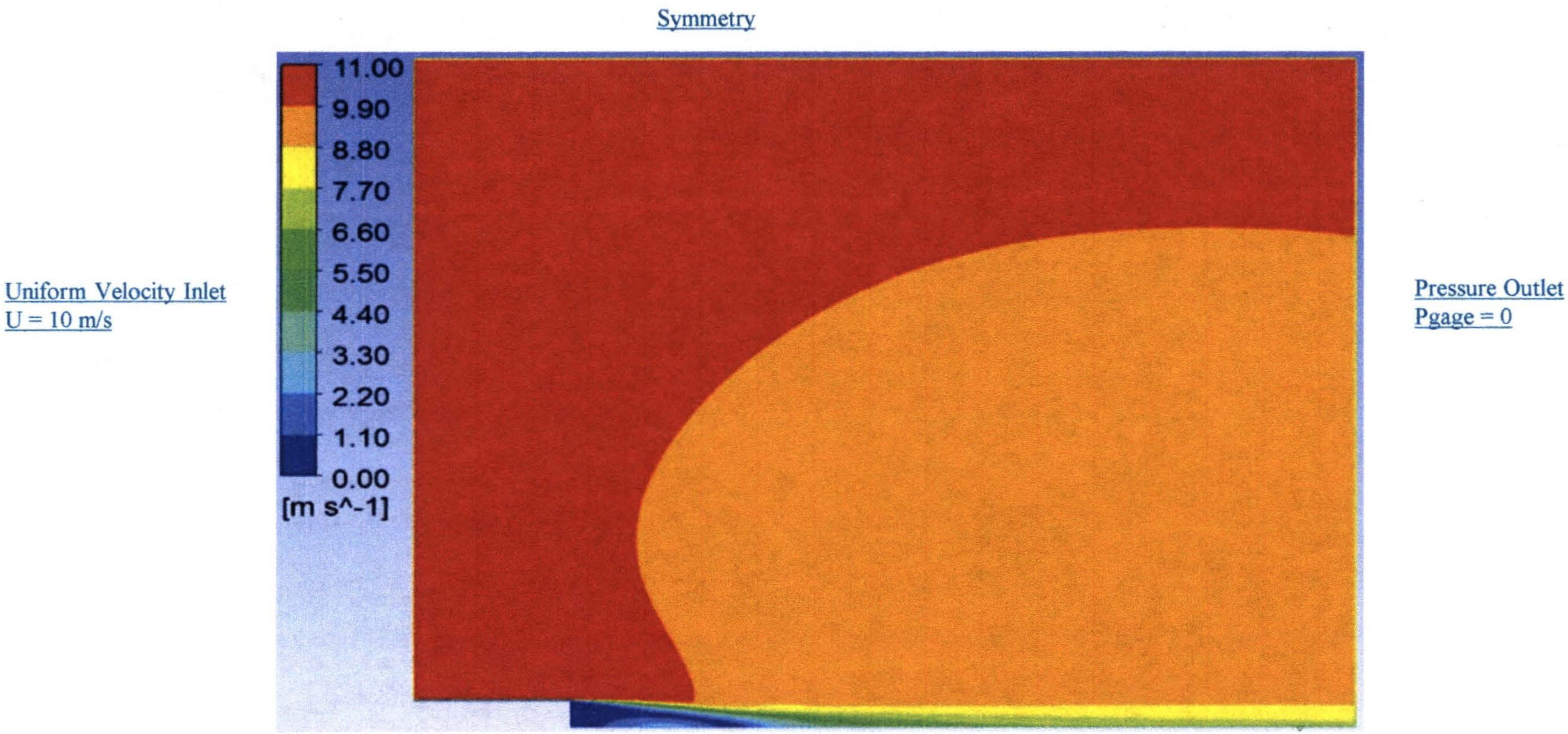
- There are uncertainties and errors in using CFD
  - No standard method for evaluating uncertainty in CFD
  - Potential Errors include:
    - physical approximation error
    - computer round-off error
    - iterative convergence error
    - discretization errors
    - computer programming errors
    - usage errors
    - turbulence induced errors

http://www.cfd4aircraft.com/int_conf/IC3/welcome/welcome.html

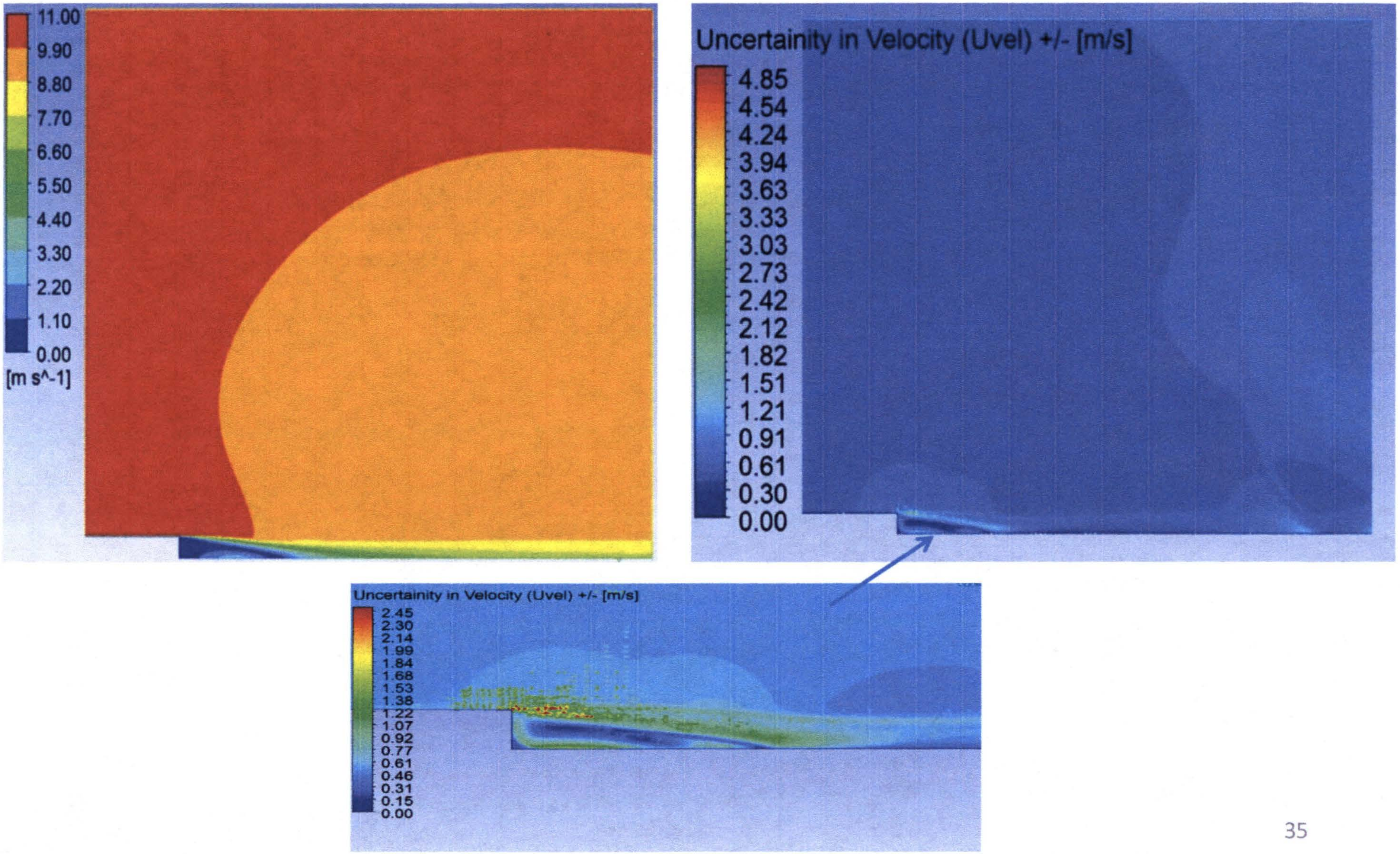
Background

- **This presentation describes an approach to validate the uncertainty in using CFD. The method will use the state of the art uncertainty analysis applied to the ke-realizable turbulence model to predict the velocity uncertainty of a backward facing step.**

Velocity Magnitude Prediction – Backward Facing Step



Velocity Prediction with Uncertainty



Method

- To estimate the uncertainty, the following ASME Standard was used.

ASME V&V 20-2009 “**Standard for Verification and Validation in Computational Fluid Dynamics and Heat Transfer**”

- A thorough literature review was used to determine the current “State of the Art” for estimating uncertainties and is included in the published paper.

Summary of Method

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

$$\varepsilon_{21} = S_{k2} - S_{k1}$$
$$\varepsilon_{32} = S_{k3} - S_{k2}$$
$$R_k = \varepsilon_{21} / \varepsilon_{32}$$

- Three convergence conditions are possible:

Monotonic convergence: $0 < R_k < 1$

Oscillatory convergence: $R_k < 0$[i]

Divergence: $R_k > 1$

Summary of Method –cont.

- 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 $r = r(X_1, X_2, \ldots X_J)$ as shown,

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

- 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

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

Uncertainty Variables ke-realizable (OPENFOAM – SimpleFoam)

- There are 87 Different Input Parameters for the ke-realizable model in SimpleFoam
  - These include:
    - Boundary Conditions
    - Wall Functions
    - Fluid Properties
    - Turbulence Parameters
    - Solution Schemes
    - Solvers
    - Mesh
    - ect.

Uncertainty Variables Considered

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 in order from top to bottom.

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

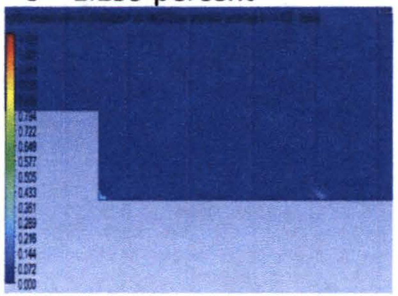
Oscillatory Variables

- The uncertainty for each of the following was calculated for each cell using the following method outlined by Stern, Wilson, Coleman, and Paterson. ***S*** is the simulated result. For this case it is the upper velocity $\boldsymbol{S_U}$ and the lower velocity $\boldsymbol{S_L}$.
  - **epsilion turbulent mixing length dissipation rate inlet (m^2/s^3)**
  - **k turbulent intensity kinetic energy inlet (m^2/s^2)**
  - **Pressure outlet (Pa)**
  - **Velocity Inlet (m/s)**
  - **Kinematic viscosity nu=17.06e-06 [13.6e-06 -> 23.06e-06] (m^2/s) represents air [0-50-100] degrees C**
  - **Grid size**
  - **Turbulence Models**
  - **Solver**

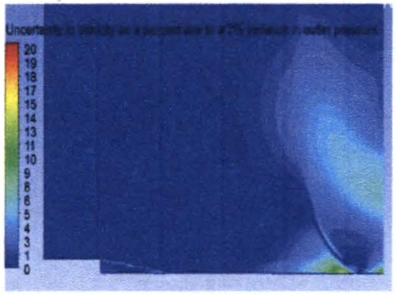
$$U_{Oscillatory} = \frac{1}{2}(S_U - S_L)$$

Results (Oscillatory Variables)

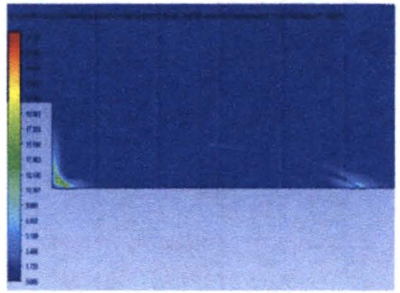
epsilion turbulent mixing length dissipation rate inlet (m^2/s^3)
0 – 1.155 percent



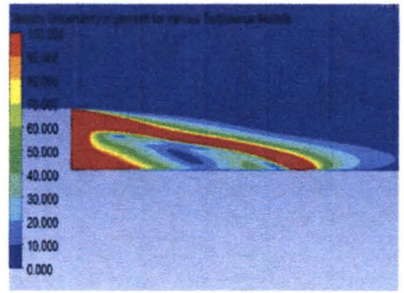
Pressure outlet (Pa)
0 – 20 percent



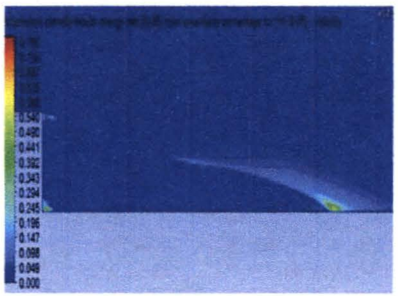
Kinematic viscosity nu=17.06e-06 [13.6e-06 -> 23.06e-06] (m^2/s) represents air [0-50-100] degrees C
0 – 27.727 percent



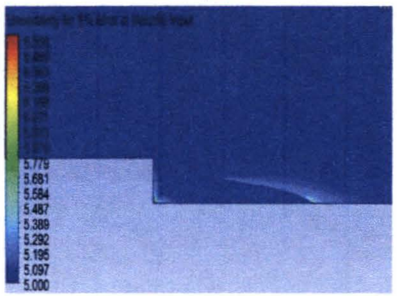
Turbulence Models
> 100 %



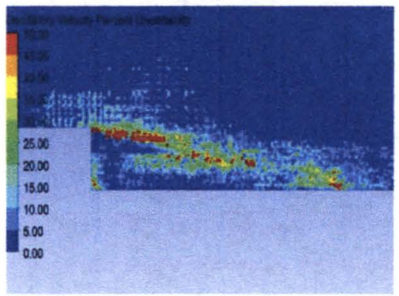
k turbulent intensity kinetic energy inlet (m^2/s^2)
0 – 0.785 percent



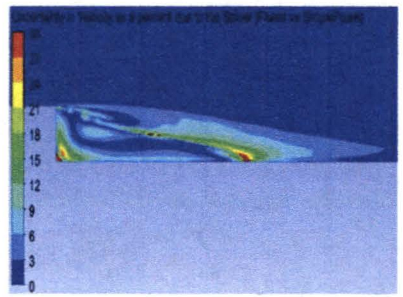
Velocity Inlet (m/s)
0 – 6.558 percent



Grid size
0 – 698 percent



Solver
> 30 %



Percent – is the percentage change in local velocity

Monotonic Convergence Variables (Numerical)

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

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

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

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

Monotonic Convergence Variables (Numerical)

- Step 2, calculate representative grid ratio, ***r*** as shown

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

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

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

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

Monotonic Convergence Variables (Numerical)

- Step 4 is to calculate the extrapolated values as shown

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

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

- 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.

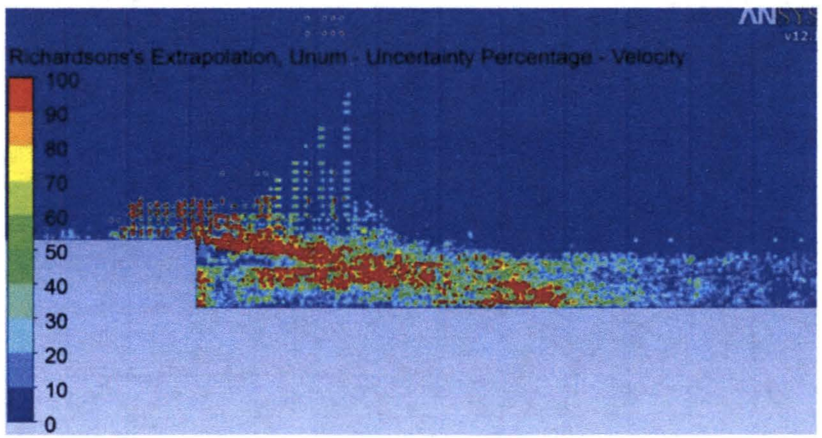
$$GCI_{fine}^{21} = \frac{1.25 * e_a^{21}}{(r_{21}^{p} - 1)}$$

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

Results (Monotonic Convergence)

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 11 as estimated by Richardson's extrapolation method.



Results (Monotonic Convergence)

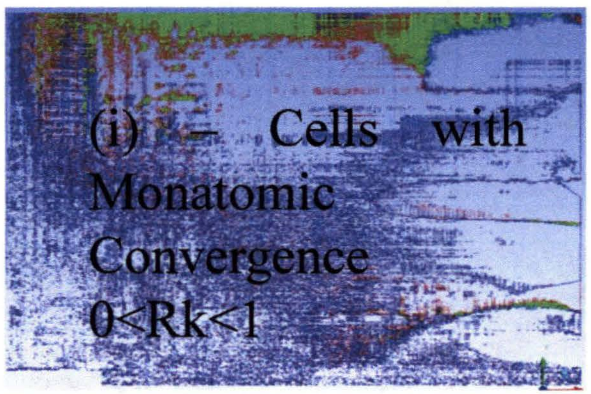
Numerical

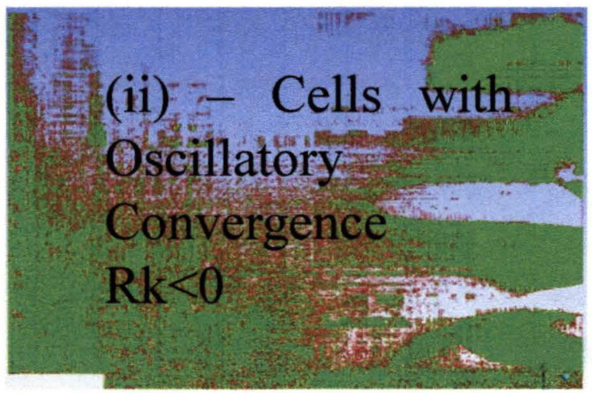
Three convergence conditions are possible:

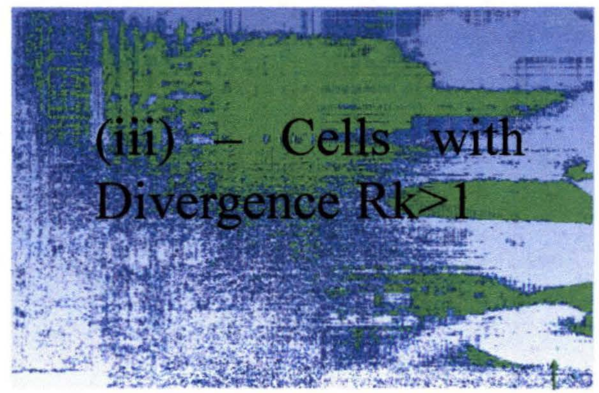
Monotonic convergence: $0 < R_k < 1$

Oscillatory convergence: $R_k < 0$[i]

Divergence: $R_k > 1$



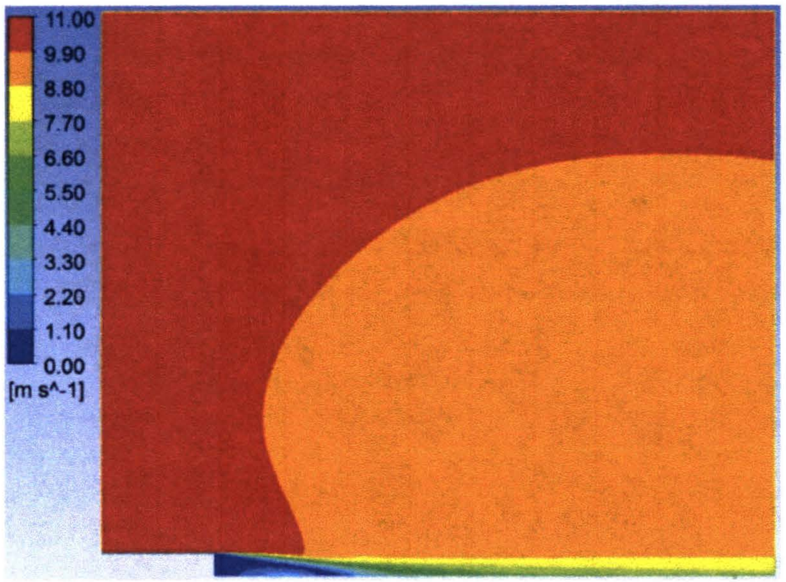


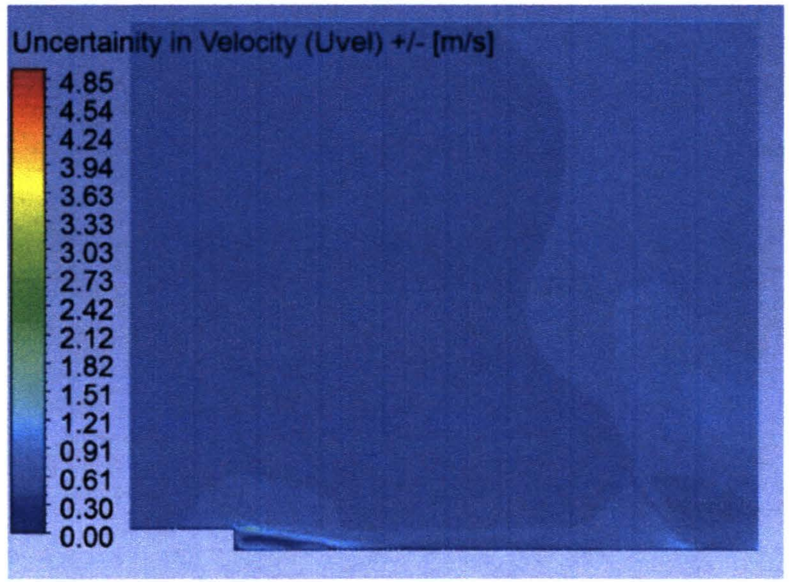


It is believed the errors in this method are due to the turbulence and or the interpolation scheme used between the 3 grids.

Results

- A root-sum-squared (rss) of the uncertainty variables was calculated (omitting Richardson's Extrapolation)

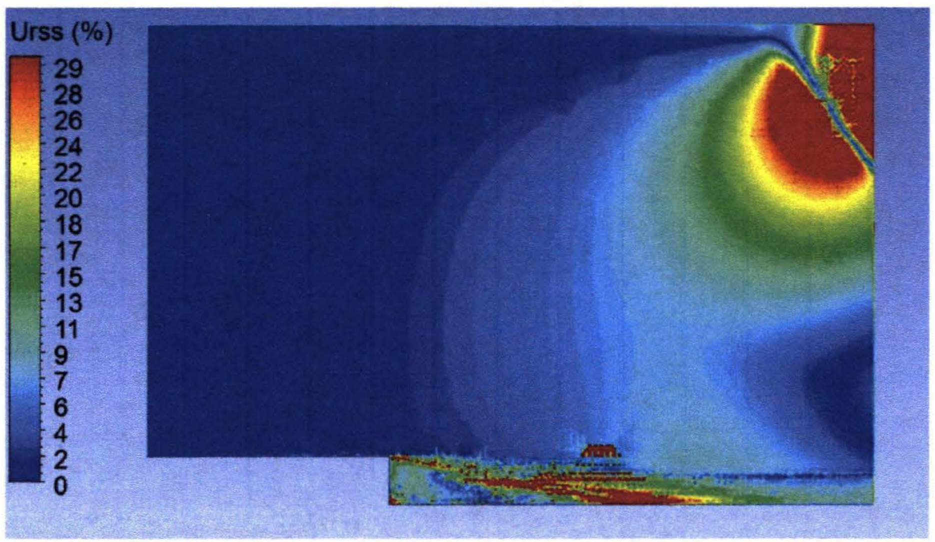




The highest uncertainty is +/- 4.85 m/s.

Note on Domain Sizing

- During several preliminary cases of the grid convergence study, one case provided an excellent example of domain sizing.
  - A CFD analyst is always troubled with trying to keep the domain size large enough to not affect the solution.
    - Using the oscillatory method, one can see the solution differences between the three grids. In the case presented below, the domain size is too small.



Conclusion

- This paper outlines an uncertainty analysis for the ke realizable turbulence model for a backward facing step.
- The velocity magnitude was predicted using CFD.
- The uncertainty parameters listed in Table1 were analyzed using an oscillatory convergence calculation or a monotonic convergence calculation.
- Plots of the velocity magnitude can be combined with a corresponding uncertainty plot for an accurate velocity prediction.
- There are other variables that would influence the uncertainty calculation. Examples of these other parameters include solution schemes, other turbulence models, and time accurate solutions. Future work will include analyzing each of these items.

Conclusion /Recommendation

- The following input uncertainty's are recommended

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		grid specific
		3,311,689		
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

Recommendation / Future Work from AIAA-2013-0258

- It is suggested that the CFD community begin to compile a list of the many input parameters associated with each uncertainty calculation for different problems and output variables.
- Ideally, an analyst could assemble a table of all uncertainty variables and estimate a number based on historical data rather than running separate CFD cases for each variable.

EELV Public Available Information

- The Rockets Behind the Missions:
  - Delta II
  - Delta IV
  - Atlas V
  - Pegasus
  - Taurus
  - Falcon 9

- http://www.nasa.gov/centers/kennedy/launching rockets/

EELV Public Available Information

- Each of these vehicles have a Payload Planners Guide or Users Guide
- http://www.ulalaunch.com/site/docs/product_cards/guides/DeltaIIPayloadPlannersGuide2007.pdf
- http://spacecraft.ssl.umd.edu/design_lib/Delta4.pl.guide.pdf
- http://spacecraft.ssl.umd.edu/design_lib/Atlas5.pl.guide.pdf
- http://www.orbital.com/NewsInfo/Publications/Pegasus_UG.pdf
- http://www.orbital.com/NewsInfo/Publications/taurus-user-guide.pdf
- http://www.spacex.com/Falcon9UsersGuide_2009.pdf

Delta II

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

http://www.ulalaunch.com/site/docs/product_cards/guides/DeltaIIPayloadPlannersGuide2007.pdf

Delta II - Continued

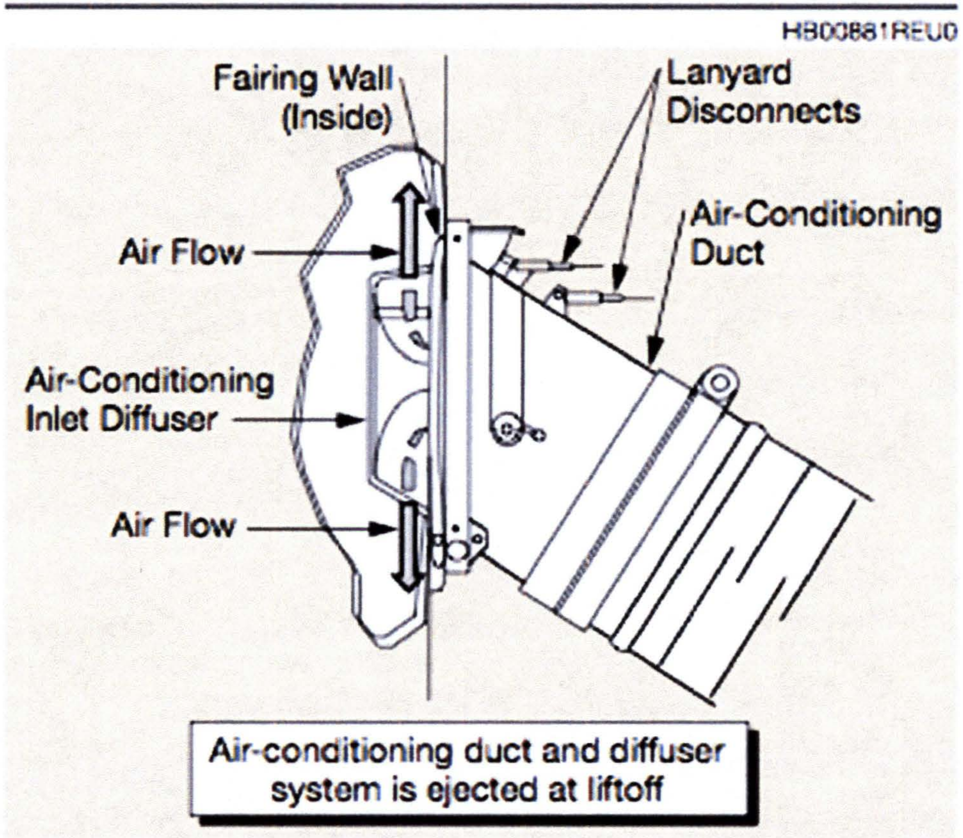


Figure 4-1. Payload Air Distribution System

http://www.ulalaunch.com/site/docs/product_cards/guides/DeltaIIPayloadPlannersGuide2007.pdf http://spacecraft.ssl.umd.edu/design_lib/Delta4.pl.guide.pdf

Delta IV

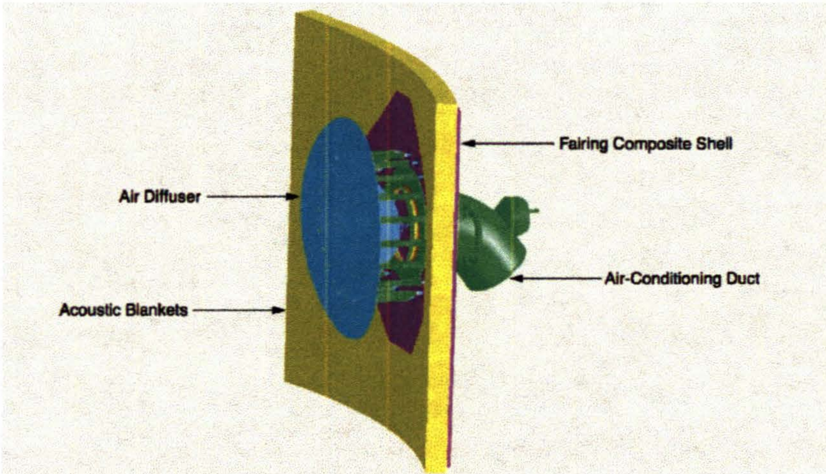


Figure 4-1. Standard 4-m Composite Fairing and 5-m Composite Fairing Air-Conditioning Duct Inlet Configuration

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.

Air flows around the payload and is discharged through vents in the aft end of the fairing.

Fairing sizes 4meter and 5 meters in diameter

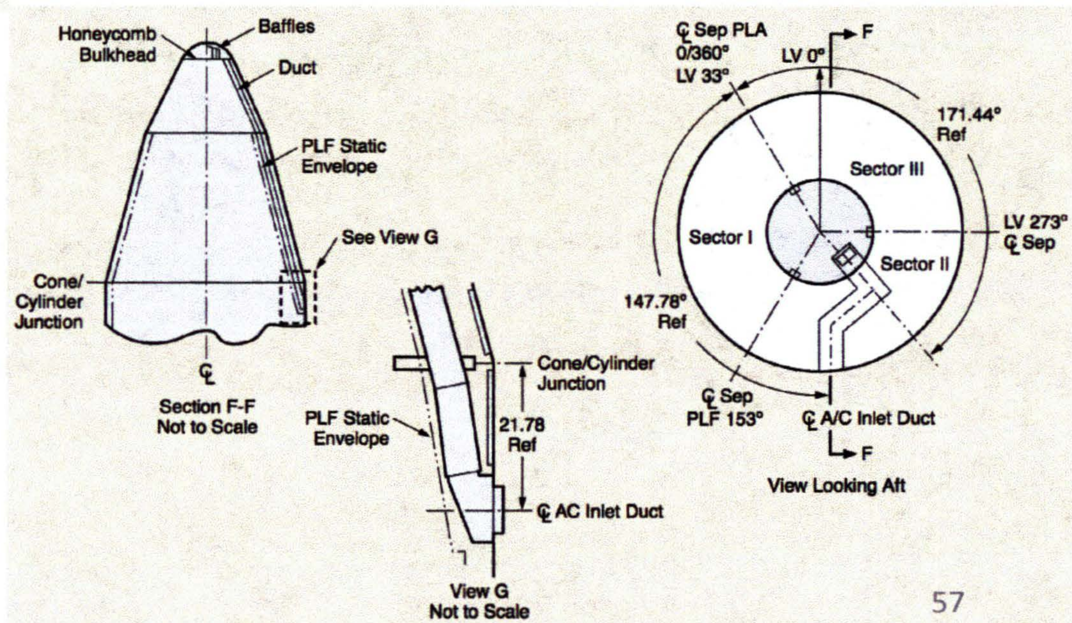


Figure 4-2. 5-m Metallic Fairing Payload Air-Distribution System

Atlas V

- Internal ducting defectors in the PLF direct the gas upward to prevent direct impingement on the spacecraft.
- The conditioning gas is vented to the atmosphere through one-way flapper doors below the spacecraft.
- 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.
- There will be localized areas of higher flow velocity at, near, or associated with the air conditioning outlet.
- Maximum air flow velocities correspond to maximum inlet mass flow rates.
- Reduced flow velocities are achievable using lower inlet mass flow rates.

- Flow Rates

  A) Atlas V 400: 0.38–1.21 kg/s ±0.038 kg/s (50–l60 lb/min ±5 lb/min),

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

- Fairing sizes are 4meters and 5 meters in diameter

http://spacecraft.ssl.umd.edu/design_lib/Atlas5.pl.guide.pdf

Pegasus

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

http://www.orbital.com/NewsInfo/Publications/Pegasus_UG.pdf

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).
- Fairing inlet conditions are selected by the Customer
- Fairing diameters are 63 inches and 92 inches

http://www.orbital.com/NewsInfo/Publications/taurus-user-guide.pdf

Falcon 9

- Once fully encapsulated and horizontal, the Environmental Control System (ECS) is connected
- Payload environments during various processing phases are:
  - In hanger, encapsulated – Flow Rate: 1,000 cfm
  - During rollout: 1,000 cfm
  - On pad: Variable from 1000 to 4500 cfm
- Fairing diameter is 5.2 meters

http://www.spacex.com/Falcon9UsersGuide_2009.pdf

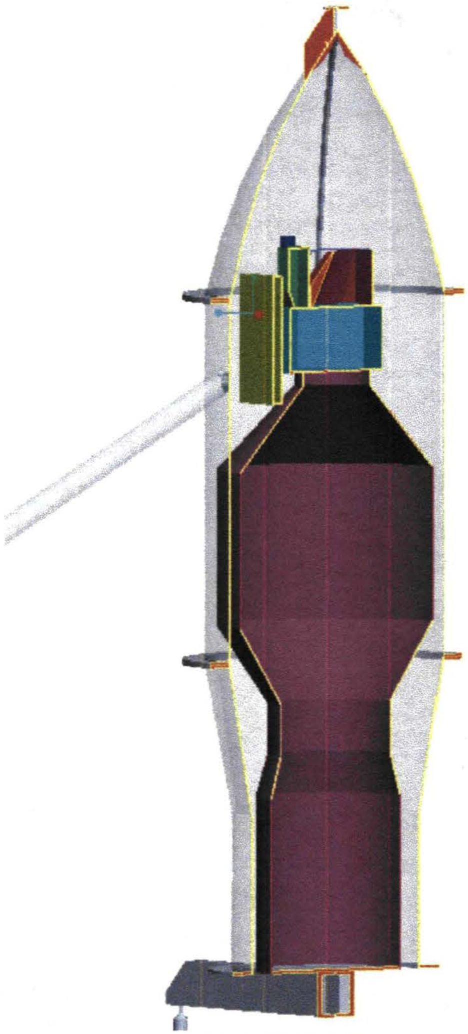
Proposed Dissertation Topic / Research

Objectives / Methods

Objectives	Methods
1. Develop, model, and perform a CFD analysis of (3) generic non-proprietary environmental control system / spacecraft configurations	FLUENT (commercially) and OPENFOAM (open source) available 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.

Objective 1:Develop, model, and perform a CFD analysis of (3) **generic** spacecraft and fairing

- Computer Aided Drafting (CAD) model will be created of the 'mockup' spacecraft and fairing using Pro/ENGINEER
- The CAD model is translated into IGES file for ANSYS Workbench and ANSYS FLUENT
- FLUENT/OPENFOAM will be used to iterate a solution on a mesh independent grid



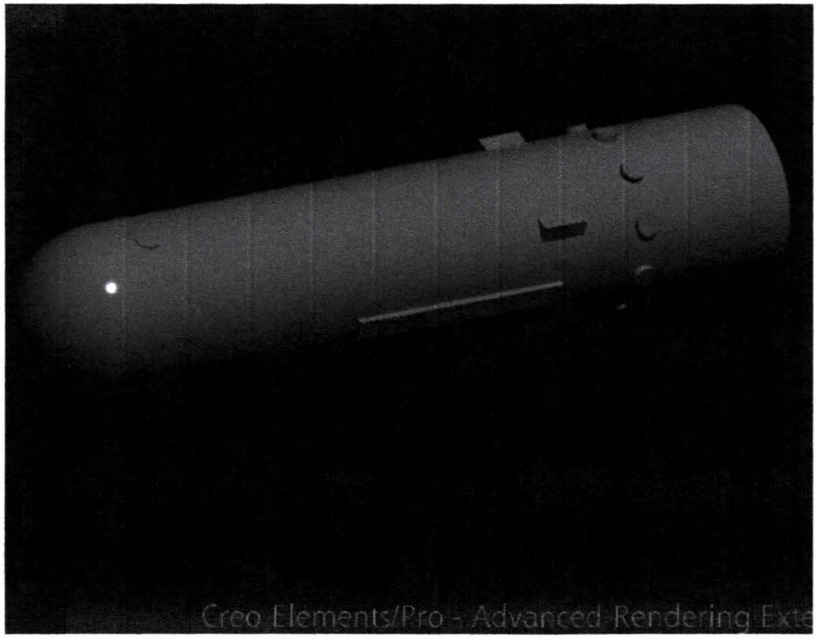
AIAA-2005-4910, 2005

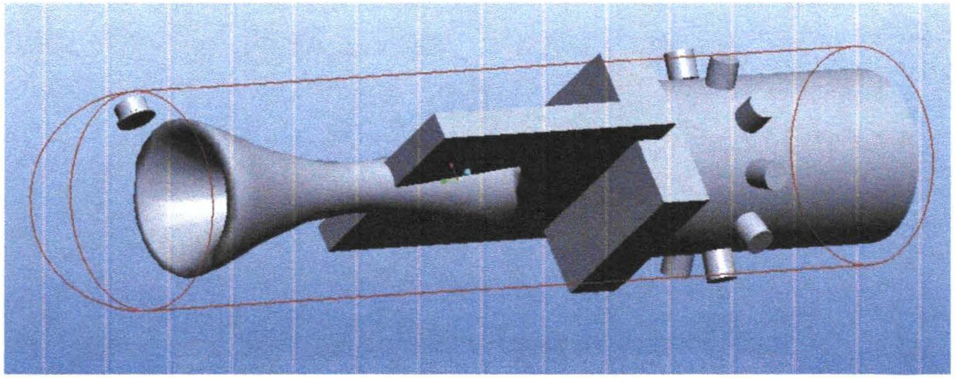
(3) Configurations

- Fairing Sizes are approximately 1m, 1.6m, 2.3m, 3m, 4m, 5m in diameter.
- (3) generic fairing diameters are selected to envelop the EELV fairing configurations
  - 0.75m
  - 3.5 m
  - 5.5 m
- Inlet Conditions range from 1000 cfm to 4500 cfm
- Inlet diameters sizes are unknown, a generic value was chosen
- Spacecraft diameters range with fairing sizes, a generic spacecraft was drawn and scaled accordingly

(3) Configurations

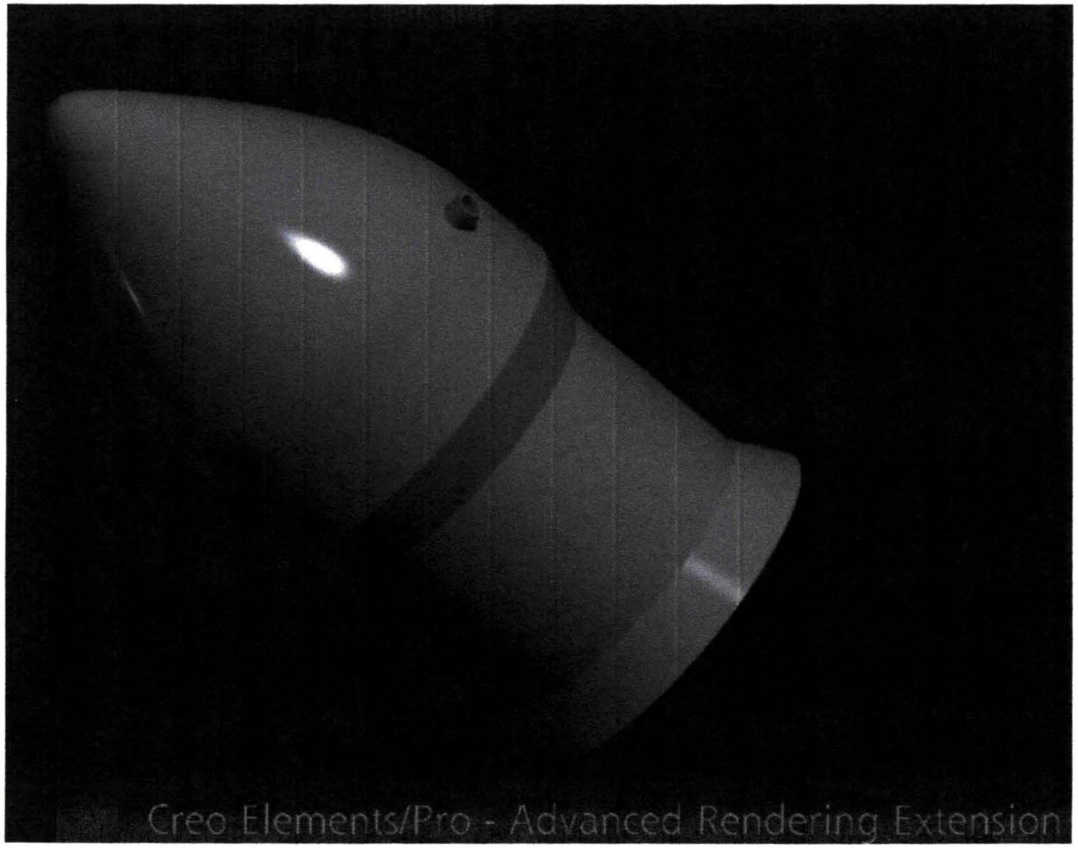
- CAD model of the spacecraft was created in Pro/ENGINEER, 0.75m

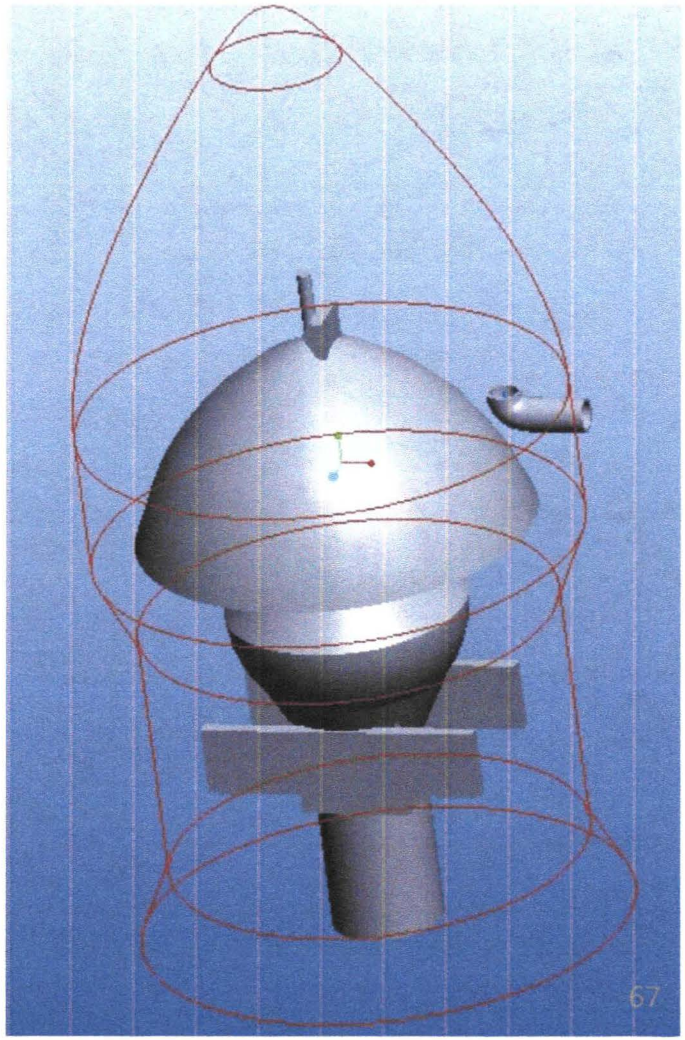




(3) Configurations

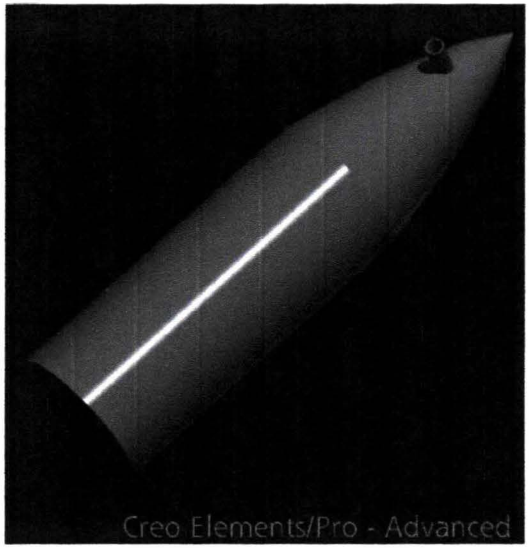
- 3.5m

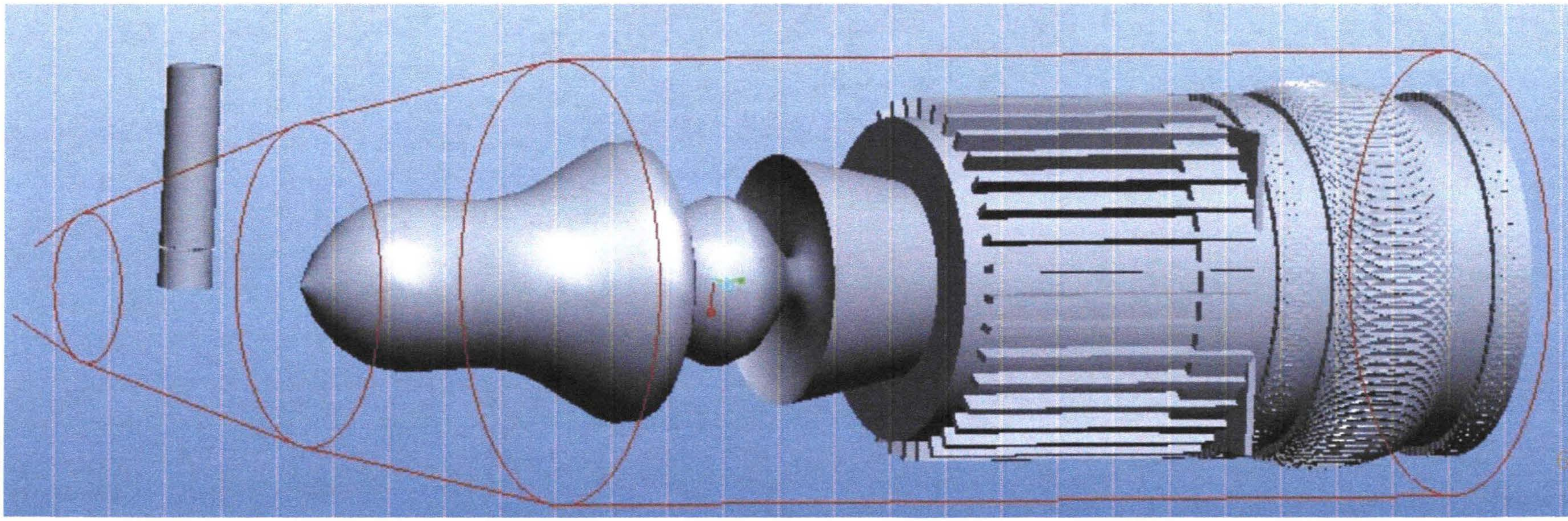




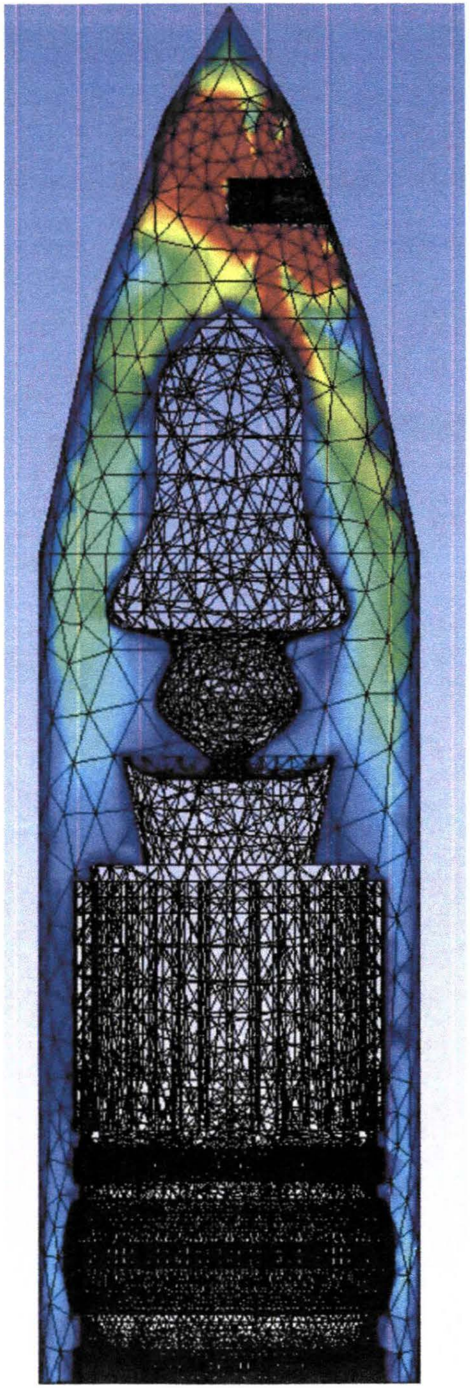
(3) Configurations

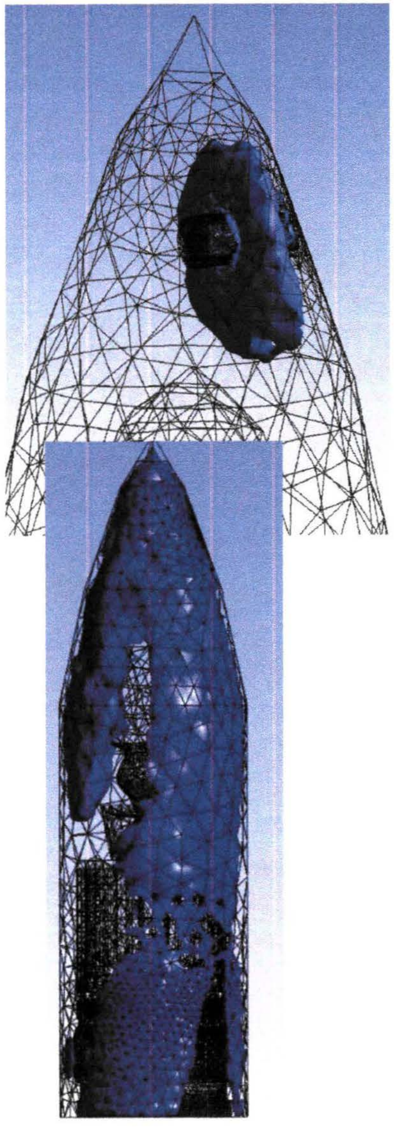
- 5.5m

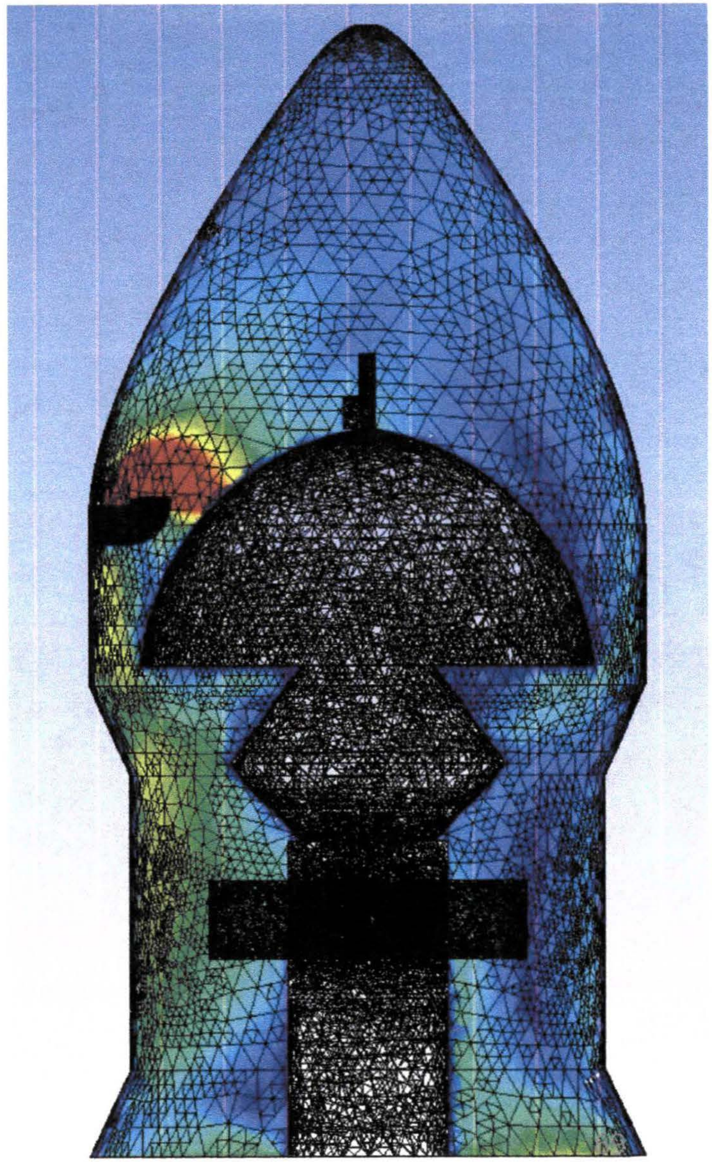




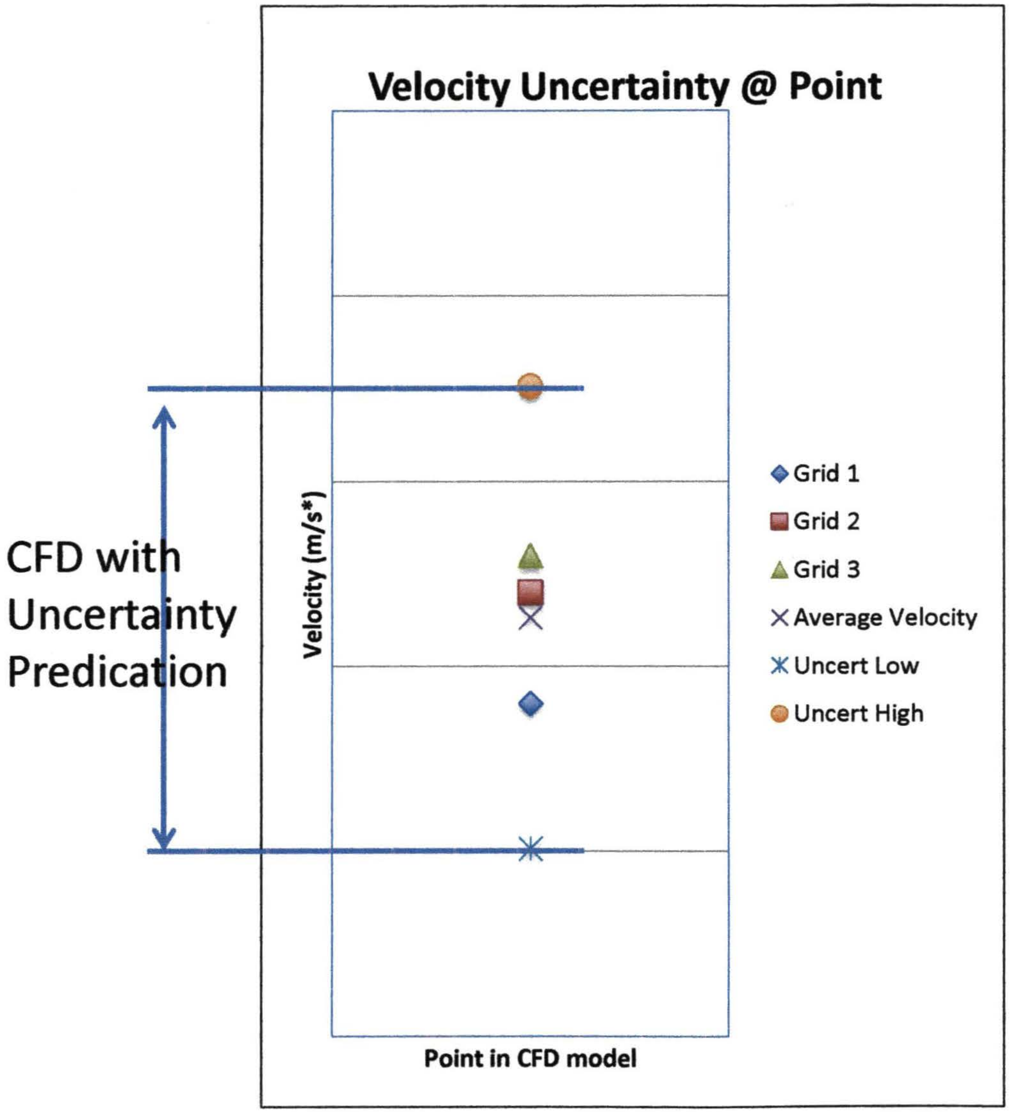
Example CFD







Objective 2:Perform an uncertainty analysis of the CFD model



- Comprehensive Approach to Verification and Validation of CFD Simulations – ASME Journal of Fluids Methodology
  - The method uses three separate grids (rough, medium, fine) to evaluate the uncertainty in the CFD prediction.
  - The velocity at every point in each of the three solutions will be compared to one another.

Objective 3: Compile a Table of Uncertainty Parameters

- (Example from Backward Facing Step): The following uncertainty's are recommended

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		grid specific
		3,311,689		
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

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

Proposed Deliverables

- 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

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- 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 - 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.

Expected Contribution

- Demonstrate a CFD Uncertainty Analysis for 3-D, low speed, incompressible, highly turbulent, internal flow can be calculated for an entire simulation domain
- Develop a higher order interpolation scheme to be used for grid interpolations and uncertainty quantification
- Investigate the applicability of using the ASME 5-Step procedure for the entire computational domain to estimate numerical uncertainties
- Calculate the uncertainty in using different turbulent models
- Demonstrate this method can contribute to the study of importance of input parameters in CFD
- 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
- Demonstrate the ability to use OPENFOAM to calculate the velocity field of an Environmental Control System
- Compare the results of OPENFOAM verses an industry standard CFD software program (ie FLUENT).

Summary

Summary

- ECS Systems velocities are analyzed using CFD
- The uncertainty of this method is unknown and not documented
- The proposed research will culminate into a table of uncertainty parameters for ECS/spacecraft systems and this table will be added to the literature thus to the body of knowledge
- The results will be incorporated into a software program and used by NASA LSP to estimate uncertainties associated with CFD modeling of spacecraft/ECS systems

Thank You

Questions?