

Sub-System Level Validation Of Detached Eddy Simulation Model In Sierra/FD For Gravity System Spin-to-Arm Predictions

Srinivasan Arunajatesan (1515) and Mary McWherter-Payne (5422)
August, 2013.

Simulations of a supersonic over-expanded jet issuing into a transonic crossflow have been carried out using the SIERRA/GasDyn (Conchas) code and an unsteady turbulence model. The test case chosen represents a sub-system level validation problem for the prediction of spin-to-arm performance for stockpile gravity bombs. The validation is carried out using measured profiles of mean velocities as the validation metric. Detailed mesh, time step and CFL refinement studies have been carried out. The results demonstrate that the predictions obtained using the SST-DES model are very strongly dependent on the model switching function behavior. This results in prediction of spurious flow features, with the results worsening with refinement. Hence it is concluded that the model doesn't validate.

I. Executive Summary

The spin up of stockpile gravity bombs that occurs when the Spin Rocket Motors (SRM) fire is a critical portion of the flight. During SRM firing, a large region of vortical flow is generated leading to interaction between the vortex and the fin. Simulating the vortex-fin interaction is an important element in predicting spin-up and quantifying margins for the B61-LEP. Previous work¹ has shown that the standard two-equation RANS turbulence models do not possess sufficient fidelity to accurately predict the vortex-fin interaction produced by the SRM. An unsteady turbulence model² which contains improved physics has been implemented in the Sierra gas dynamics code. This validation effort attempts to quantify the capability of new unsteady turbulence models to predict vortex-fin interaction produced by SRMs using sub-system (jet-in-crossflow).

The configuration studied here corresponds to that studied experimentally by Beresh, et al.^{3,4,5}. The setup consists of a supersonic jet issuing from a conical nozzle on the floor of a 12" x 12" test section in a blow-down wind tunnel. The tunnel Mach number is 0.8. The nozzle has an exit diameter of 0.375" and a design Mach number of 3.73. In the experiments, the nozzle orientation is varied from vertical to cant angles up to 45°. Here we only use the 0° cant angle data. PIV measurements of the velocity field (both mean and turbulent fields) are available at a streamwise location that is 33.8 jet diameters downstream of the jet and on the spanwise symmetry plane (for the 0° cant angles). The flow conditions studied here correspond to the test conditions in Beresh, et al.³⁻⁵ with primary focus restricted to the Mach 0.8 and jet-momentum-ratio of 10.2.

In carrying out the validation, care was taken to ensure that the appropriate boundary conditions were prescribed. This was done by carrying out a pre-cursor calculation of the TWT geometry and matching the measured boundary layer profile and pressure gradient profile. In addition, the accuracy of the nozzle boundary conditions were ensured by comparing the pressure expansion in the nozzle to the measured values.

The meshes used in this work spanned a range from the coarsest mesh of 8.2 million nodes to the finest mesh obtained by uniformly refining the coarse mesh by a factor of 2 in each dimension, resulting in a fine mesh of about 65 million nodes. These meshes were generated using industry standard practices – near wall resolution such that $y+=1.0$ and jet shear layer resolution such that ~50 points were present across the jet in the coarse mesh. Mesh refinement studies on these meshes were carried out along with time step refinement and CFL refinement. In addition, the flow field was also computed using a different mesh that had a significantly different topology and provided much better refinement of the jet shear layer. In addition calculations were repeated using a different flow solver using the same model and using a different unsteady turbulence model. The essential conclusions of the effort can be summarized as follows:

1. Mesh Refinement Study showed that
 - a. Errors on Coarse Mesh are smaller than Errors on Fine Mesh
 - b. Coarse Mesh Errors ~ 20-50% in $y/D=[5-12]$
 - c. Fine Mesh Errors ~ 20-100% in $y/D=[5-12]$

SAND 2013-7567 C

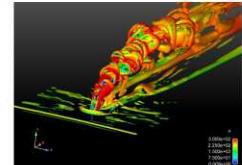
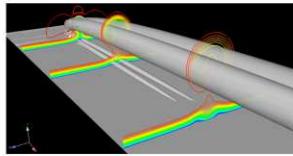
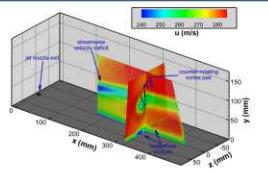
2. Overall predicted Flowfield characteristics showed strong anomalies
 - a. On the Coarse mesh the model behaves like an unsteady RANS Model
 - b. Fine Mesh predicts unphysical flow field with jet splitting into a high momentum jet and a low momentum jet
 - c. Wrong Vortex structure – likely to result in wrong forces on a fin surface
3. The Time Step Refinement Study showed that
 - a. Errors from $\Delta t=1.0e-6$ and $\Delta t=0.2e-6$ were comparable and Nearly identical errors were obtained from both simulations
 - b. Same anomalies seen in both simulations
 - c. Use of Wall functions to decrease effective CFL did not help
 - i. Flow structure is still wrong
 - ii. Did not significantly decrease the error levels
4. Detailed Analysis of the Model showed that
 - a. The model shows strong dependence on function that determines RANS vs LES regions of the flow
 - b. Other codes with same model show nearly identical behavior
 - c. Other Models with different forms show similar sensitivities
 - d. Explicit handle in $k-\epsilon$ Hybrid RANS-LES model to control wall distance dependence was used to explicitly demonstrate the dependence on this parameter.

Based on these results, it was concluded that the SST-DES model in its existing form cannot reliably predict jet-in-crossflow flow field. A strong dependence on the wall distance function needs to be better understood and calibrated or alternate methods to eliminate its unpredictable effects need to be devised.

II. References

1. Arunajatesan, S., "Evaluation of Two-Equation RANS Models for Simulation of Jet-in-Crossflow Problems", AIAA-2012-1199.
2. Strelets, M., "Detached Eddy Simulation of Massively Separated Flows", AIAA-2001-0879.
3. Beresh, S. J., Henfling, J. F., Erven, R. J., and Spillers, R. W., "Crossplane Velocimetry of a Transverse Supersonic Jet in a Transonic Crossflow," AIAA Journal, Vol. 44, No. 12, 2006, pp. 3051–3061.
4. Beresh, S. J., Henfling, J. F., Erven, R. J., and Spillers, R. W., "Penetration of a Transverse Supersonic Jet into a Subsonic Compressible Crossflow," AIAA Journal, Vol. 43, No. 2, 2005, pp. 379–389.
5. Beresh, S. J., Henfling, J. F., Erven, R. J., and Spillers, R. W., "Vortex Structure Produced by a Laterally Inclined Supersonic Jet in Transonic Crossflow", Journal of Propulsion and Power, Vol. 23, No. 2 2007, pp 353-363.

Exceptional service in the national interest



**Sub-System Level Validation Of Detached Eddy Simulation Model In
Sierra/FD For Gravity System Spin-to-Arm Predictions:
V&V Level 2 Milestone**

Srinivasan Arunajatesan, Mary McWherter-Payne



Sandia National Laboratories is a multi-program laboratory managed and operated by Sandia Corporation, a wholly owned subsidiary of Lockheed Martin Corporation, for the U.S. Department of Energy's National Nuclear Security Administration under contract DE-AC04-94AL85000. SAND NO. 2013-XXXXP

Unclassified-Unlimited Release

SAND-2013-7567 C

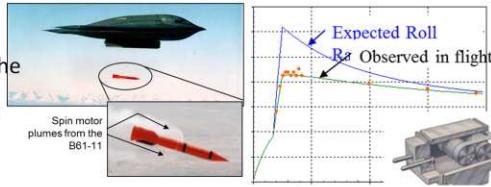
Outline

- Background
 - Motivation
 - Where this work fits into the scheme of things for B61-LEP
- Methods and Models
 - Code and Numerics
 - Turbulence Model
- Milestone Statement and Completion Criteria
- Validation Work
 - Metric
 - Approach
 - Results
 - Analysis
- Conclusions
 - Address Completion Criteria
- Path Forward

Spin Performance

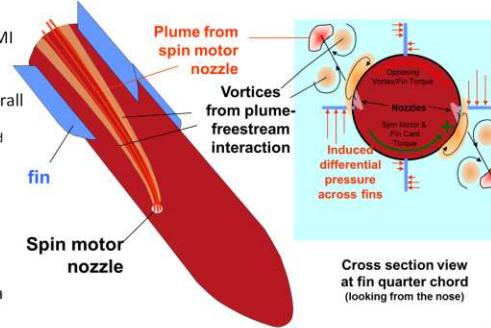
- Free-fall nuclear bombs use spin motors to drive the spin rate above the resonant frequency

- High spin rate immediately after release is used as a unique environment signal for arming



- Observed roll rate performance on B61 & B83

- Less than expected based on bomb aero, motor performance, and roll MI
 - Vortices Induced by Plume generate differential pressure across fins
 - Results in "Counter Torque", reducing overall spin Performance
 - Induced Torque Related to Vortex Strength and Distance from Fins



- Sandia must ensure that spin rate margins are adequate

- Relative "clocking" of motors and fins has been altered to increase available margin.
 - New Design might require use of this extra margin.

Unclassified-Unlimited Release

3

Gravity bombs are spun-up –they use spin-motors to get their spin rate above the resonant frequency

In addition, they use the max spin rate as an arming signal

However, the observed spin rate is less than the designed spin rate, based on motor characteristics, bomb moment-of-inertia etc.

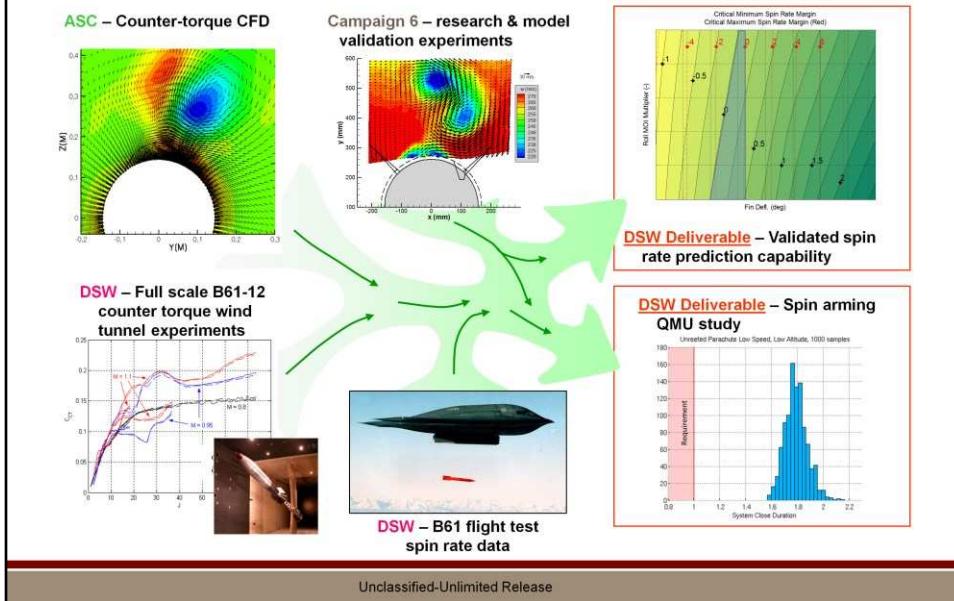
The reason is the interaction between the vortex field they induce and the fins

- The jet issued by the spin motors travel downstream and generate this complex vortex system
- These vortices generate a differential pressure field across the fins
- The torque due to this pressure field acts counter to the torque induced by the spin motors
- Hence the drop in roll rates.

Sandia's responsibility is to ensure that the spin rate margins are adequate

- the benefit to be had from relative clocking of the spin motors and the fins has already been leveraged in the new design
- so the new design, depending on the vortex-fin interactions might require the use of this extra margin
- To ascertain this and to ensure that the overall margins are adequate, accurate prediction capability is needed.

Spin-To-Arm : Overall Concept



Overall deliverable is : Spin arming QMU study

Validated CFD capability forms a part of that

- This validation is done using experimental data from TWT, full scale tests at AEDC and Ames and flight test data where available.
- The CFD code and the models in it are developed under the auspices of ASC
- Using in-house experiments at TWT, full scale experiments at AEDC and NASA Ames in conjunction with Flight tests, these models are validated.
- They form a key element of the tools generating predictions for the spin arming QMU study

Spin Arming QMU Study



- Use of a Multi-Fidelity Approach
 - Gross trends captured by the lowest fidelity levels
 - Increasing fidelity levels capture more details
- Fidelity Levels
 - Reduced Order Model using Correlations for Vortex Strength and Position
 - Quick running, can generate data for all conditions
 - Gross Trends, hand tuned to match certain points
 - RANS Solutions utilizing Two-equation RANS models
 - Can generate results for most conditions
 - Capture General trends and more flow field details, but known shortcomings
 - DES/Hybrid RANS-LES solutions
 - Can generate results for smaller subset of conditions
 - Can capture most details, requires validation
 - Full scale experimental data
 - Provides data for subset of conditions
 - And validation data for RANS and DES simulations.

5

Unclassified-Unlimited Release

A multi-fidelity approach to QMU is being put together

- the lowest fidelity levels are methods that capture the gross trends only
- as we go up in fidelity, more of the details are expected to be captured
- but, fewer data points are available because they tend to become more expensive.

The DES/HRL capability is a key ingredient of this approach.

- it can generate potentially accurate results albeit for a small set of conditions
- but, these methods require to be validated for these problems.

Spin-To-Arm: Counter Torque CFD



- Integrated Codes : Sierra/GasDyn (Conchas)
 - Unstructured Node Centered Finite Volume Code
 - Developed for High Performance and Low Memory Footprint
 - Second order accurate
 - Inviscid Flux options between Roe's Riemann Solver and Modified Steger-Warming Scheme
 - Gradient limiting for monotonicity
 - Edge based Viscous Fluxes for accuracy and performance
 - Implicit scheme for Robust Time integration
 - Only nearest Neighbor sensitivities for low memory footprint
 - BDF2 Scheme for Second order time accuracy
 - Two Equation Shear Stress Transport (SST) RANS model
 - SST-DES model for unsteady flows
- Biggest Challenge for Spin-To-Arm Predictions
 - Turbulence Modeling

6

Unclassified-Unlimited Release

The code being used is conchas – features are shown

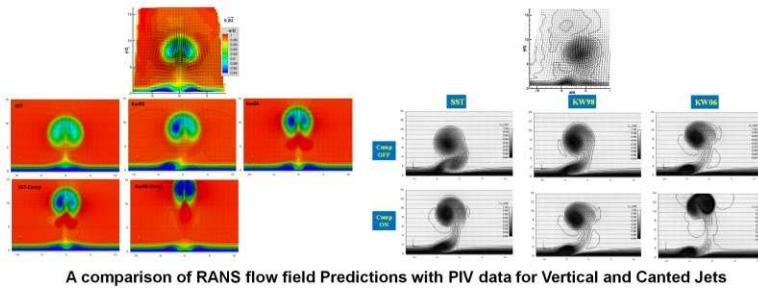
- key aspect is that it is an accurate code written for high performance

Biggest challenge, however, for the flow field of interest is Turbulence modeling

Spin-To-Arm: Counter Torque CFD



- Turbulence Modeling For Counter Torque Predictions
 - Detailed Evaluation of State-of-The-Art Two Equation RANS Models
Carried out shows inadequacy of this class of Models.
 - Comparisons against Sub-System Level Jet-in-Crossflow Problem
 - All Models Generally Capture Shape and Gross Behavior
 - None of the models capture the velocity deficit or vortex location correctly



7

Unclassified-Unlimited Release

Prior work has shown that RANS models are deficient

Main reason for not being able to capture the quantitative details

- the analysis showed that the different stresses evolved at different rates
- Different length scales were needed to match these different evolution rates.

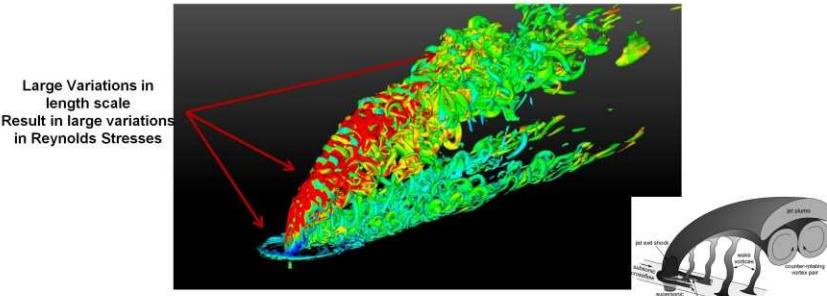
RANS models which force a single length scale on the problem are not capable of capturing this variation.

Hence, they can capture gross features, but not predict vortex strength and location accurately.

Spin-To-Arm: Counter Torque CFD



- Detached Eddy Simulation Modeling
 - Shown in literature to Capture large variations in the evolutions of the Reynolds stress components
 - More accurate flow field description at greater computational cost
 - Validation For the Jet-in-Crossflow problem required



Details of the Turbulent Flow Field

Captured Using DES

Unclassified-Unlimited Release

8

DES, as seen in literature, can capture these widely varying length scales

But, Cost is much greater –

- These are time accurate calculations
- finer meshes are needed to resolve relevant length scales,
- These result in smaller time steps and hence Long run times to get averages.

Also, this method has not been validated for these problems.

Spin-To-Arm: Counter Torque CFD



- Detached-Eddy Simulation Modeling Approach
 - Motivated by desire to overcome cost of resolving near-wall turbulence
 - Large Classes of flowfields consist of vast regions of unsteady turbulence well removed from wall regions
 - Dynamics of these flows are often strongly dictated by these "Detached" regions
 - It is feasible to resolve these regions well
 - LES like models perform reasonably well in these regions.
- Approach:
 - Maintain Near-Wall Steady RANS modeling
 - Resort to Unsteady "resolved" turbulence capture (like LES) away from walls, in "detached" regions
- Modeling:
 - Modify the length scale definition to switch from RANS scales near walls to Mesh scales away from the walls
 - Near Wall (Attached) regions : Time accurate RANS solution
 - Detached regions : Resolved (upto Mesh Scales) Unsteady Turbulence

9

Unclassified-Unlimited Release

DES was motivated by desire to overcome near wall modeling costs in LES type methods

So uses RANS in near wall regions and LES like models in detached regions

The models do this by switching the length scales from turbulent length scales to mesh scales away from walls

SST Turbulence Model



- DES Model in Sierra/GasDyn (Conchas):
 - Based on Shear Stress Transport (SST) RANS model
- SST RANS Model
 - Solve Transport Equations for Turbulent Kinetic Energy (k) and Specific Dissipation Rate(ω)
 - Combines the free shear layer behavior of k - ε model and near wall behavior of k - ω model
 - Blended using a selector function
 - = 1 at the wall
 - = 0 away from walls

$$\frac{\partial \bar{\rho} \tilde{k}}{\partial t} + \frac{\partial (\bar{\rho} \bar{u}_j \tilde{k})}{\partial x_j} - \frac{\partial}{\partial x_j} \left[\bar{\mu} + \sigma_k \frac{\bar{\rho} \tilde{k}}{\bar{\omega}} \frac{\partial \tilde{k}}{\partial x_j} \right] = P_k - D_k + S_k$$
$$\frac{\partial \bar{\rho} \tilde{\omega}}{\partial t} + \frac{\partial (\bar{\rho} \bar{u}_j \tilde{\omega})}{\partial x_j} - \frac{\partial}{\partial x_j} \left[\bar{\mu} + \sigma_\omega \frac{\bar{\rho} \tilde{\omega}}{\bar{\omega}} \frac{\partial \tilde{\omega}}{\partial x_j} \right] = P_\omega - D_\omega + S_\omega$$

$$SST = F_1(k - \omega) + (1 - F_1)(k - \varepsilon)$$
$$F_1 = \tanh(\arg_1^4)$$
$$\arg_1 = \min \left[\max \left(\frac{\sqrt{k}}{0.09\omega y}, \frac{500\nu}{y^2\omega} \right), \frac{4\bar{\rho}\sigma_\omega k}{CD_{k\omega} y^2} \right]$$

Unclassified-Unlimited Release

DES models are typically built on RANS models

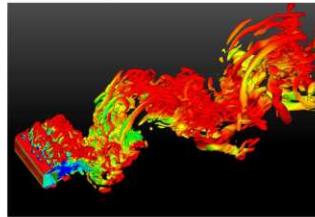
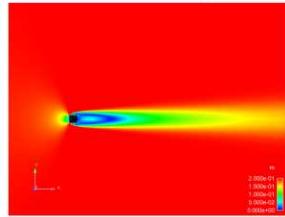
We will use the SST DES model, which is built on SST RANS model which blends k-eps away from walls with k-w near walls using a selector function which is shown.

SST-DES Turbulence Model



- Detached Eddy Version of SST Model
 - DES Switching using a modified dissipation length scale
 - When Turbulent Length Scale is > Mesh Resolution Dissipation is increased to reduce modeled TKE and Eddy Viscosity
 - Reduced Eddy Viscosity permits flow to become unsteady

$$D_k = \beta^* \rho k \omega F_{DES}$$
$$F_{DES} = \max \left[\frac{L_t}{C_{DES} \Delta}, 1 \right]$$

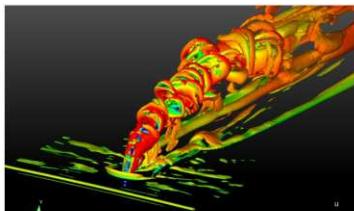


11

Unclassified-Unlimited Release

In DES mode, the SST DES model uses the length scale ratio function to switch. So with RANS solutions you will get what you see in fig. on left. With DES mode on, the wake becomes unsteady and flow structures develop and evolve.

**FY13 V&V Program: Predict the aerodynamic performance of a full scale stockpile gravity bomb with Sierra Gas Dynamics code,
Srinivasan Arunajatesan, 65755/004.03.01**



Iso-surfaces of Q-criterion colored by streamwise velocity from the jet in cross flow case.

FY13 Project Goal: Validate the Sierra Gas Dynamics Code Conchas for the simulations of Jets-in-crossflows using an unsteady turbulence model.

Project Description and Challenge: Simulating the vortex-fin interaction is an important element in predicting spin-up and quantifying margins for the B61-LEP. Previous work has shown that the standard two-equation RANS turbulence models do not possess sufficient fidelity to accurately predict the this interaction. Simulations of a C6 funded jet-in-crossflow experiment will be completed using an unsteady turbulence model. The resulting predictions will be used to quantify spatial and temporal convergence.

Relevancy:

- This project will demonstrate the ability to use high-fidelity computational fluid dynamics simulations to impact stockpile issues that require aerodynamic analyses.

- **PI, Team, and Org#(s):** Srinivasan Arunajatesan (ORG 1515), Christopher W.S. Bruner (ORG 1515), Mary McWherter-Payne (ORG 5422), Mike Nucci (ORG 1523)

FY13 Spend Plan & Deliverables by Quarter:

Q1-Q2 (125K) :

Comparisons with experiments for simulations of jets at different momentum ratios into a Mach 0.8 free stream flow.

Q3-Q4(125K) :

Comparisons with experiments for simulations of Jets at $J=10.2$ momentum ratio into free streams at different Mach numbers.

Unclassified-Unlimited Release

Quad chart – please note the 0.5 FTE level funding.

Milestone



- **Title:** Sub-System Level Validation of Detached Eddy Simulation Model in Sierra/FD for gravity system Spin-to-Arm Predictions
- **Description:** The spin up of stockpile gravity bombs that occurs when the Spin Rocket Motors (SRM) fire is a critical portion of the flight. During SRM firing, a large region of vortical flow is generated leading to interaction between the vortex and the fin. Simulating the vortex-fin interaction is an important element in predicting spin-up and quantifying margins for the B61-LEP. Previous work has shown that the standard two-equation RANS turbulence models do not possess sufficient fidelity to accurately predict the vortex-fin interaction produced by the SRM. An unsteady turbulence model which contains improved physics has been implemented in the Sierra gas dynamics code. This validation effort will begin to quantify the capability of new unsteady turbulence models to predict vortex-fin interaction produced by SRMs using sub-system (jet-in-crossflow) validation data.
- **Completion Criteria:** Simulations of a C6 funded jet-in-crossflow experiment will be completed using the unsteady turbulence model. The resulting predictions will be used to quantify spatial and temporal convergence. Comparisons will be made between the simulations and experiment for the quantities of interest (time-averaged vortex strength and location).

13

Unclassified-Unlimited Release

Milestone statement : key completion criteria:

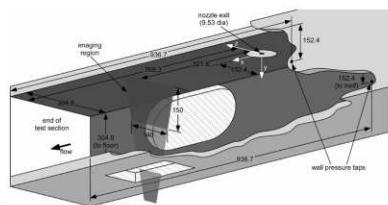
- carry our simulations
- mesh and time steps refinement will be carried out
- quantify errors and convergence

Will use vortex strength and location as key metrics

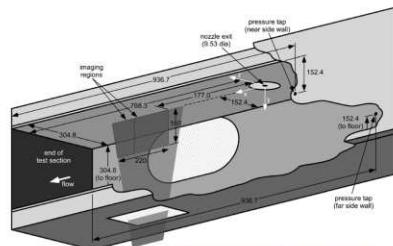
Flow Conditions

- Data Available at Wide Range of Conditions
 - Current Study Focuses on Mach 0.8 and $J=10.2$
- Data Also Available for Canted Nozzles
 - Cant Angles $0^\circ, 15^\circ, 30^\circ, 45^\circ$
 - Current work uses only 15° data

	0.5	0.6	0.7	0.8
16.7				<input checked="" type="checkbox"/>
10.2	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
5.6				<input checked="" type="checkbox"/>
2.8				<input checked="" type="checkbox"/>



Cross Plane PIV



Symmetry Plane PIV

14

Unclassified-Unlimited Release

Flow data we will use is the PIV data available from The TWT experiments by Beresh et al.

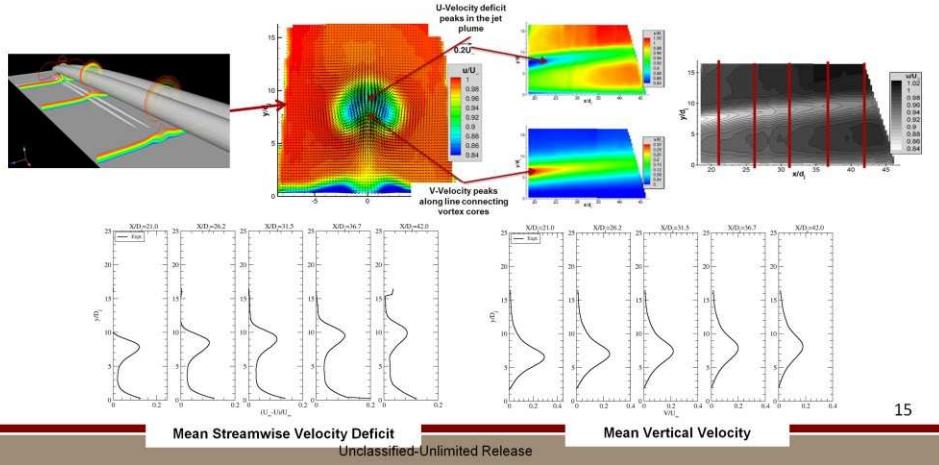
Two planes of PIV data are available

- spanwise mid plane of the tunnel and,
- a cross plane at $33.6D_j$ downstream of jet.

We will focus on $M=0.8, J=10.2$ vertical jet condition

Validation Metric

- Key Parameters : Vortex Strength and Location
 - Magnitude of Streamwise Velocity Deficit and Vertical Velocity => Vortex Strength
 - Location of Velocity Deficit Peak and Vertical Velocity Peak => Vortex Location



15

Jet issues and evolves downstream forming a counter-rotating vortex pair just below it. The flow structures are shown

- On the ground plane you see the horse-shoe vortex pair
- PIV X-plane data shows signature of the jet plume and the vortices.
- The streamwise velocity deficit magnitude peaks in the plume and vertical velocity peaks just below it.

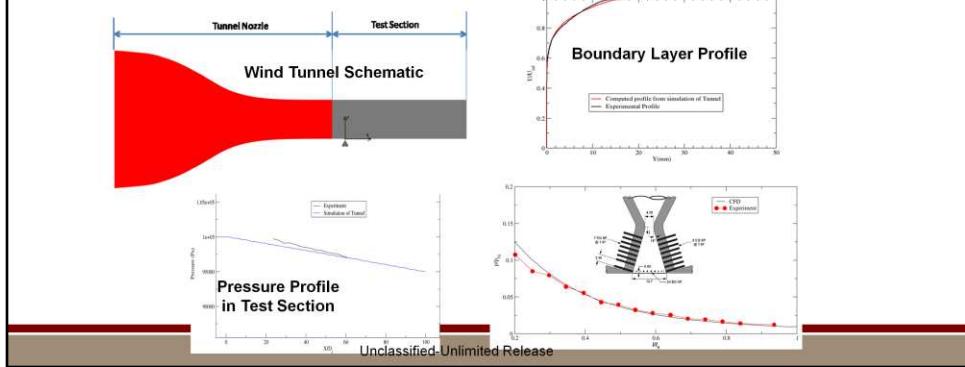
The locations of these peaks and mags. Of the peaks are representative of the vortex strength and location.

We will use these profiles to quantify validation in this work.

Validation Approach

- Boundary Conditions

- "Precursor" Empty Wind Tunnel Calculation is Used to Provide Inflow Profiles for DES calculations
 - Entire TWT tunnel from downstream of settling chamber is modeled in a separate calculation
 - Downstream pressure boundary condition is adjusted to match the boundary layer profile and pressure gradient measured in the experiments
 - Solution from this at the station corresponding to boundary of DES domain is extracted and used to provide boundary conditions to DES simulation
- Nozzle: Wall Pressures in Nozzle compared to Measured Profiles
 - Ensures Correct Jet boundary conditions.



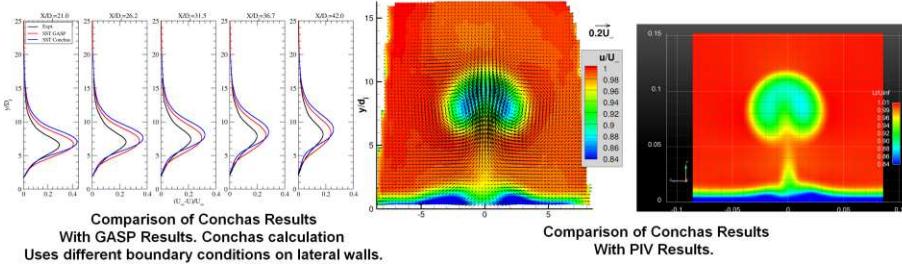
We use a precursor calculation to provide steady boundary conditions for the simulations.

We match boundary layer thickness, profile, pressure gradient in the tunnel and jet nozzle pressure expansion to ensure correct simulations conditions.

Model Implementation Check



- Compared RANS results from Conchas with Results from Commercial Code GASP
 - GASP had been used on this problem before to examine RANS models for this flowfield
 - RANS Results Agreed reasonably with previous RANS calculations
 - Differences attributed to previous calculations using slip side walls



17

Unclassified-Unlimited Release

We compared RANS results from Conchas with results from commercial code called GASP to make sure we get same results.

This is important since DES model is built on RANS model.

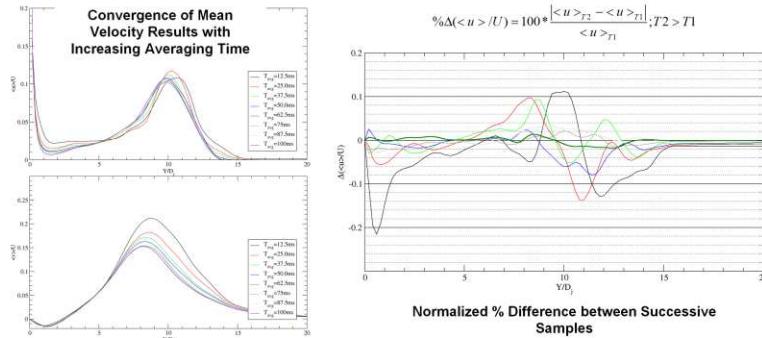
Note : Slightly different meshes were used – GASP mesh doesn't lend itself to refinement easily, so used a different mesh was used for conchas

Also GASP used slip side walls for tunnel, conchas uses no-slip side walls.

Hence the small differences in the results.

Simulation Convergence

- Comparison against Experiment using Time Averaged Velocity Profiles
 - A preliminary study was carried out to determine time to convergence
 - Averages were collected for increasing time intervals to determine how long it takes for the mean velocity profiles to converge
 - Simulation takes ~0.1 secs to converge to within 2% difference between samples.



18

Unclassified-Unlimited Release

To understand how long to run, we need to know how long the results take to converge.

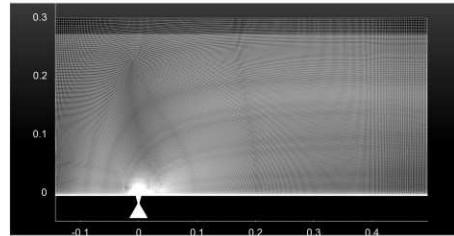
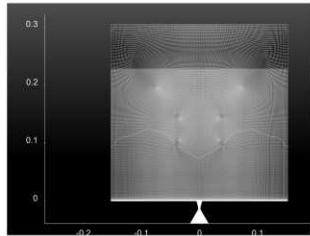
To understand this we ran a preliminary study

- Collected stats for different time intervals
- When diff. between averages at successive intervals < 2% convergence was declared.

Mesh Refinement: Meshes



- Coarse Mesh corresponds to RANS Mesh : ~8.2 million Nodes (S2)
- Uniform Refinement to Obtain Fine Mesh : ~ 65 million Nodes (S1)
- Generated using Standard Best Practices
 - Near Wall Spacing such that $y+ < 1.0$
 - Jet Shear Layer Spacing such that ~50 points across jet in coarse mesh



19

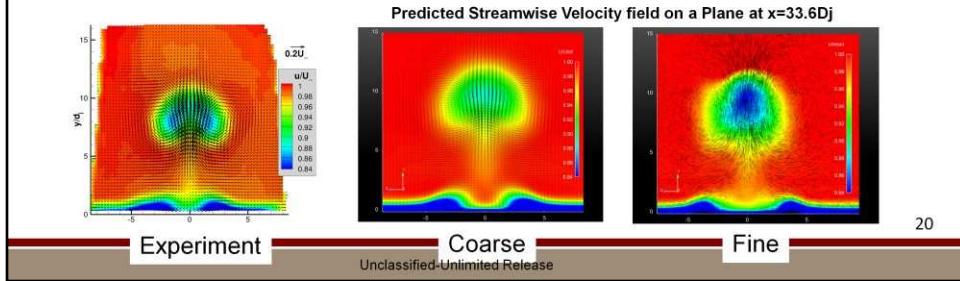
Unclassified-Unlimited Release

Meshes used. The 65m points meshes are roughly where we can be at most for turn around and practical use.

Mesh Refinement: Results



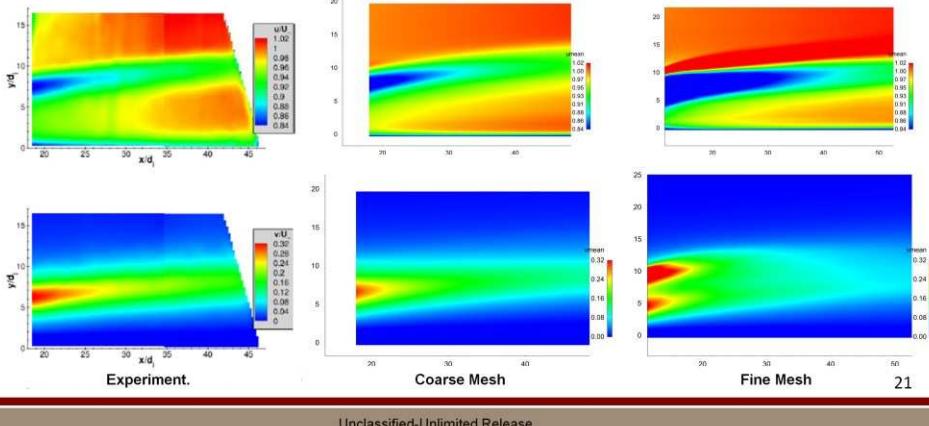
- Qualitatively
 - Neither mesh produces the qualitative shape of the jet plume at the downstream location
 - Jet plume clearly sits too high
 - Mushroom shape of the flow field not re-produced in either case
 - Shows strong differences in evolution of the plume and the CVP.
 - Coarse Mesh solution is “better” than fine mesh solution
 - Fine mesh shows “blob” structure not representative of flow field seen in Experiment.



Main take away – neither mesh works. Wrong flow structure, more wrong for finer mesh!

Mesh Refinement: Results

- Symmetry Plane at
 - Coarse Mesh Flow Structure is more representative of the experimentally observed flow structure
 - Jet plume sits too high, Velocity Deficit is over predicted
 - Fine Mesh flow structure farther off from experiments
 - Double Hump Seen in v -velocity and high speed flow still visible on the streamwise velocity plots

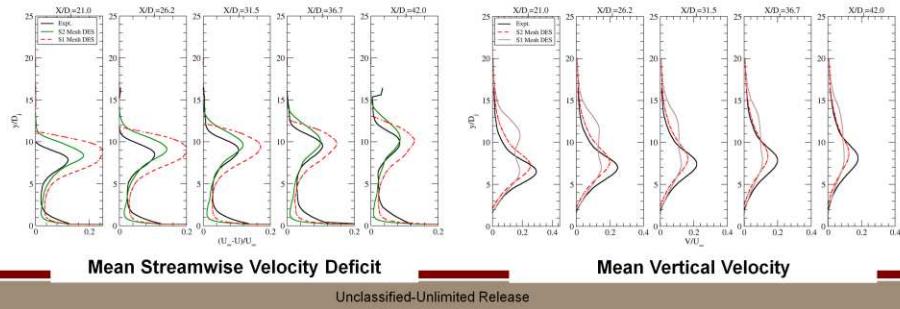


Strong differences seen between computed and measured values
Weird double streak seen in vertical vel. Contours.

Mesh Refinement: Results



- Streamwise Velocity Deficit Profiles
 - Results show an over prediction of velocity deficit on both meshes
 - Coarse mesh errors are smaller than fine mesh errors
 - Based on the location of the peak in profiles, jet trajectory is predicted to be too high
- Vertical Velocity Profiles
 - Peak location is too high
 - Predicted CVP trajectory is too high
 - Peak magnitude is lower than the measured peaks => weaker vortex.

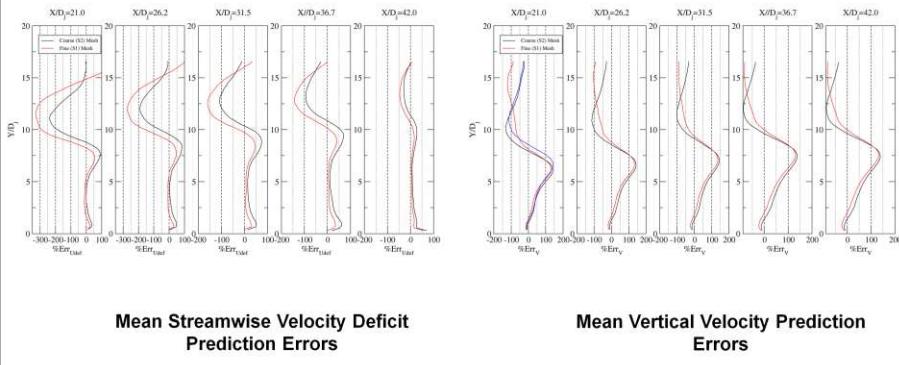


Profiles corresponding to previous slides.

Mesh Refinement: Results

■ Errors In Velocity Profiles

- %s measured relative to average value at each station
- Large errors across the entire profile on both meshes
- Errors are worse on fine mesh than on coarse mesh



23

Unclassified-Unlimited Release

Errors at each station computed relative to a mean vel at that station.

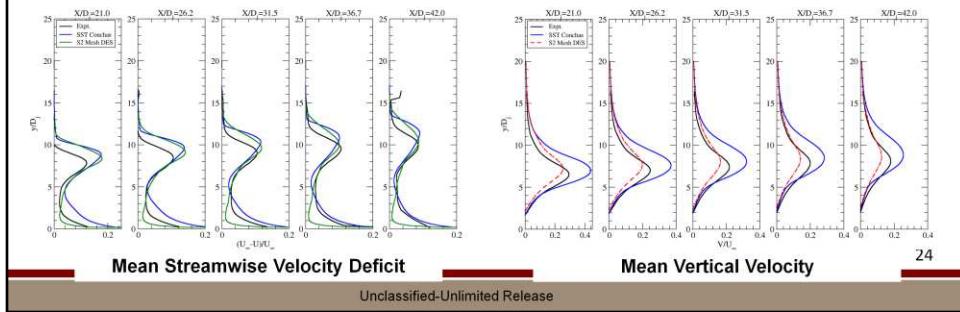
Coarse mesh errors – 20-50%

Fine mesh errors – 20-100%!! Worse!

Analysis: Coarse Mesh Results



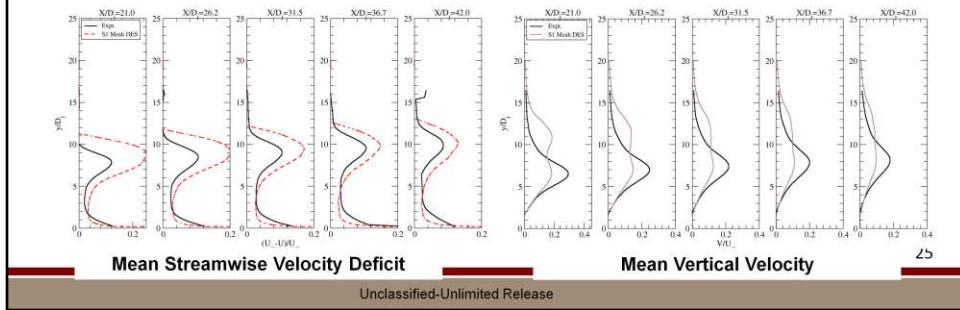
- Velocity Deficit Solution Closely follows RANS solution
- Vertical velocity prediction improves slightly with DES relative to RANS
 - Vortex location is still $\sim 10\%$ too high
- => This model on this mesh behaves like an unsteady RANS model



Coarse mesh behaves like unsteady RANS...

Analysis: Fine Mesh Solution

- Velocity Deficit
 - Much higher than the Measured Values
 - Much wider plume predicted than the measurements
 - Plume Sits Higher than the measurements
- Vertical Velocity
 - Strange Double Hump seen in the profiles



Fine mesh is clearly picking up unphysical behavior with double hump in vertical vel. Profiles.

Analysis: Fine Mesh Solution

- DES Model is a Mesh Based Model
 - Transition between RANS regions and “LES” regions are dictated by the Switching Function F_{DES}

$$D_k = \beta^* \rho k \omega F_{DES}$$

$$F_{DES} = \max \left[\frac{L_t}{C_{DES} \Delta}, 1 \right]; L_t = k^{1/2} / \omega$$

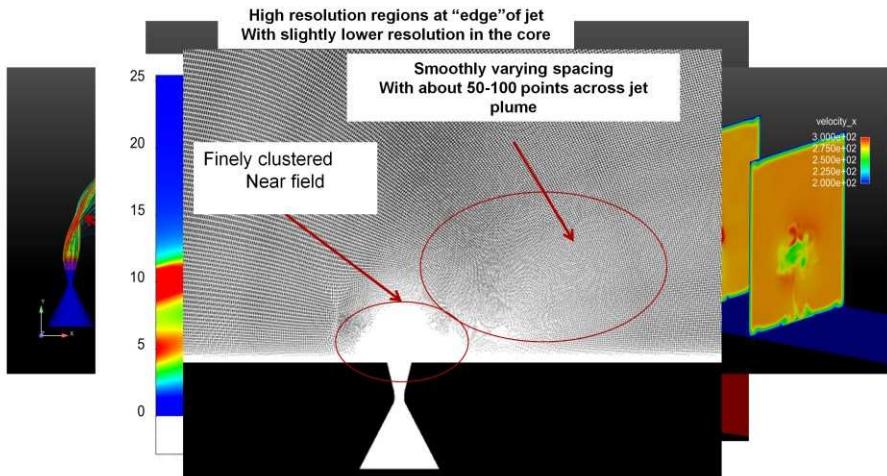
- When $\Delta \sim (k^{1/2}/\omega) - F_{DES} \sim 1$.=> RANS behavior
- When $\Delta \ll (k^{1/2}/\omega) - F_{DES} \gg 1$.=> LES behavior
 - Dissipation of Turbulent Kinetic Energy increases, reducing the modeled turbulence (eddy viscosity) levels in the flowfield
 - This allows the flow field to become unsteady
- Model Behavior can be understood by examining this transition function

26

Unclassified-Unlimited Release

The switching function behavior can illuminate model behavior in general.

Analysis: Fine Mesh Results



Unclassified-Unlimited Release

Main Points

- mesh is made according to standard practices
- length scale ratio shows expected behavior in cross planes
- symm plane shows "les" regions are not penetrating the high speed jet.
- so high mom. Fluid doesn't mix with crossflow at all and forms a separate jet.
- signature of high speed fluid seen way downstream
- hence double hump in vertical vel profiles.

Analysis: Potential Issues

- Aggressive Time Stepping
 - Although Time step size is small to obtain enough samples, largest CFL in the mesh is ~ 1200 .
 - Is this affecting accuracy?
- RANS-LES Switching Functions Shows “LES” regions confined to edges of the jet
 - High speed fluid regions don’t see “LES” type resolution
 - Need to resolve Shear Layer at the exit of the nozzle better
 - Resolution of the jet downstream seems adequate.
- Alternatively, need a different DES model
 - Either a different switching function or different parameters

28

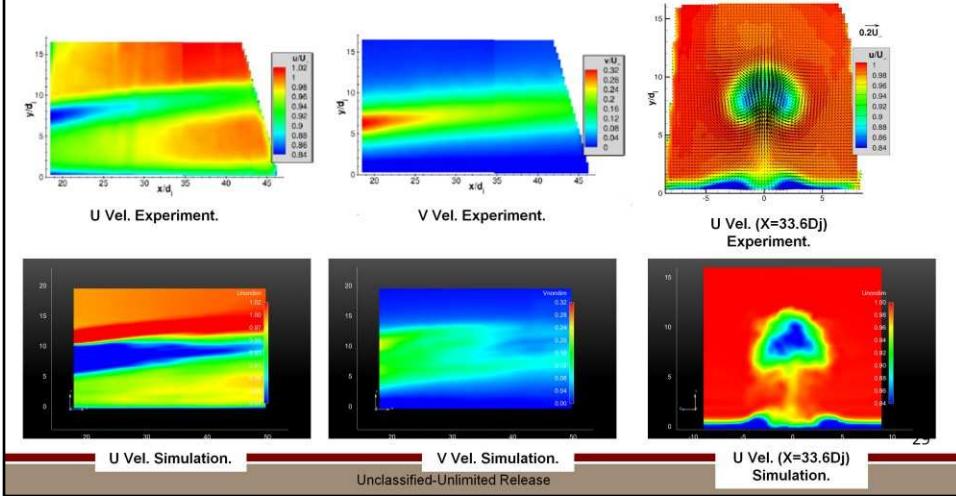
Unclassified-Unlimited Release

Potential causes.

Time Stepping Study



- Initial Case $\Delta t=1.0e-6s$
- Re-Ran Case on Fine Mesh using $\Delta t=0.2e-6s$

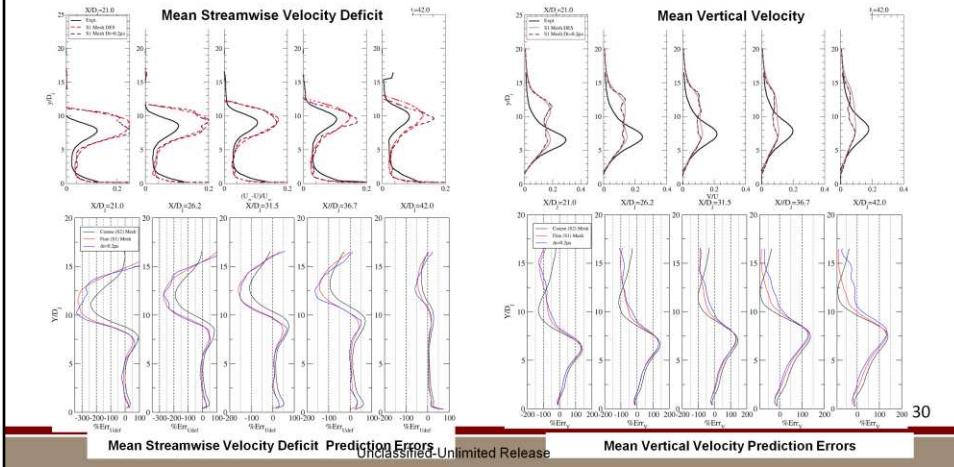


By lowering time step errors don't diminish.

Time Stepping Study

Profiles

- Very little difference compared to larger time step case
- Much larger streamwise velocity deficits than measured values
- Double hump behavior in the vertical velocity profiles.



Quantitatively....

Time Stepping Study



- Large Effective CFLs in Wall resolved calculations
 - Primarily due to strong near wall clustering
 - High speed jet fluid has to pass through tiny cells.
- Reduce Effective CFL using Wall Functions near walls
 - RANS form of wall function used
 - Near Wall regions are in RANS regions => this is acceptable.
- Wall spacing
 - Increased by 100 => near wall $y+ \sim 100$
 - Rest of the mesh clustering remains the same as fine mesh.

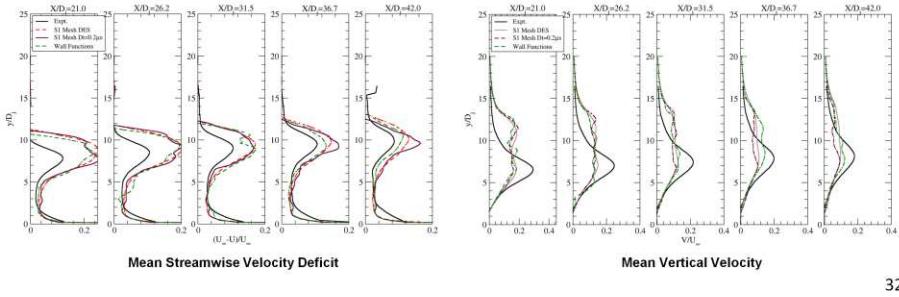
31

Unclassified-Unlimited Release

Use of wall functions at viscous wall enables us to reduce effective CFL and potentially get better accuracy.

Time Stepping Study

- Not much difference compared to Baseline case or smaller timestep case
 - Overprediction of velocity deficit
 - Double hump profile in vertical velocity.

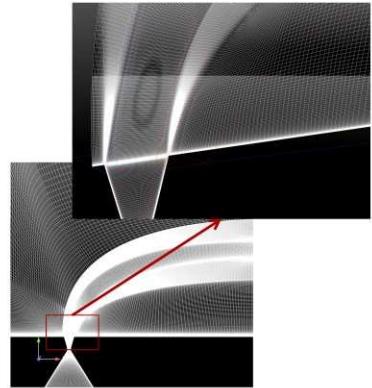
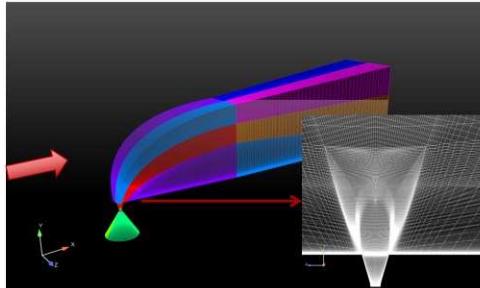


Unclassified-Unlimited Release

But doesn't work. Same behaviour as fine mesh seen.

Mesh Resolution Study

- Alternate Topology Mesh
 - Flow Aligned: Mesh Tailored to Capture Jet trajectory, Clustered around the jet shear layer
 - Wall spacing such that $y+ < 1$
 - ~ 250 points in the jet and shear layer
- Coarse Mesh : 17m cells
- Fine Mesh : 65m cells



33

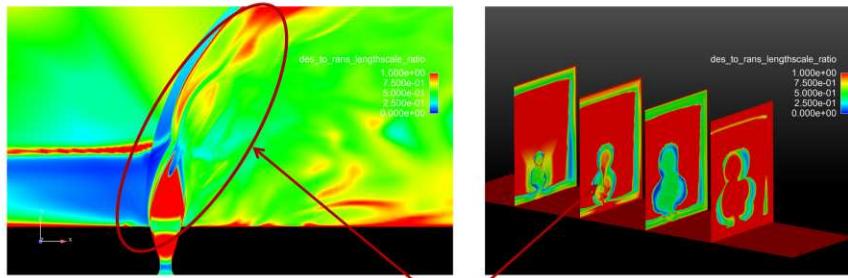
Unclassified-Unlimited Release

Alternate mesh topology – tracks the flow. Much finer shear layer resolution.

Mesh Resolution Study: Alternate Mesh Results



- Length Scale Ratio Behavior
 - Improved behavior in the near field and in the core of the jet.



Significant regions showing improved resolution
In the near field and in the core of the jet

34

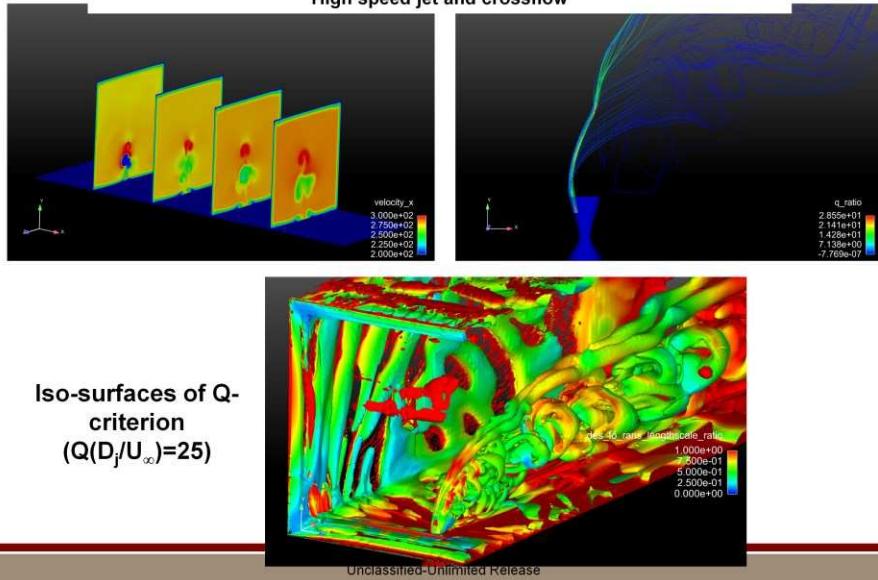
Unclassified-Unlimited Release

Expected and sought differences in the switching function behavior seen. Good Sign?

Mesh Resolution Study: Alternate Mesh Results



High Speed fluid seen Downstream – poor mixing still between
High speed jet and crossflow

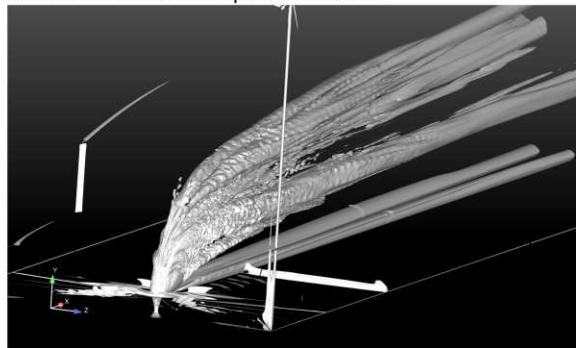


No. Again jet splits – high mom fluid doesn't mix, forming a separate jet
Gives rise to a spurious vortex pair above the jet.

Mesh Resolution Study: Alternate Mesh Results



- Fine Mesh Results Show same behavior
 - Jet Splits into two stream.
 - Lower momentum fluid mixes with crossflow.
 - Higher momentum fluid doesn't.
 - Gives rise to additional spurious vortex.



36

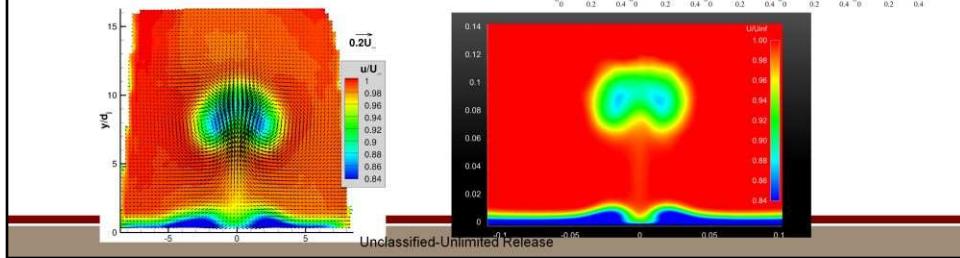
Unclassified-Unlimited Release

Same results seen in fine 65m cell mesh also.

Alternate Model



- Strong sensitivity to Switching Function
 - Is this an artifact of the SST-DES model?
 - K- ϵ Hybrid RANS-LES model
 - Being used for Captive Carry Modeling and FSI work
 - Shows slightly better behavior on Fine Mesh
 - Could not get stable solutions on Coarse Mesh
- Does this Model Show consistent behavior on other meshes?



Is it the model? Can we get better results using the k- ϵ HRL model used for captive carry work.

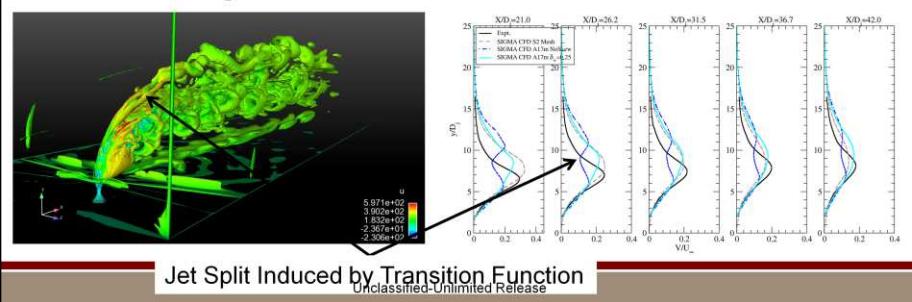
On the S1 mesh it does give better results – we cannot get solutions on the S2 mesh.

Alternate Model



- On the Alternate Mesh, similar behavior to SST-DES
 - Strong Sensitivity seen to Transition function between LES and RANS
 - Transition in this model is a function of wall distance and local turbulence quantities
- Model Sensitivity seen to Wall Distance Function
- Sensitivity Can be altered by altering the distance at which wall distance function goes to zero.

$$V_{T,HYB} = \max(f(d_w / \delta), f(\Delta, K, \varepsilon))V_{T,RANS}$$



If this model is the answer, it should behave consistently on the alternate meshes also.

Unfortunately, the double hump behavior returns on this mesh.

Jet splitting is visible

Using wall distance switch input, we can get this to go away – clear demonstration that it's the wall distance switching function

That's causing this behavior.

Overall Summary



- Mesh Refinement Study
 - Errors on Coarse Mesh are smaller than Errors on Fine Mesh
 - Coarse Mesh Errors \sim 20-50% in $y/D=[5-12]$
 - Fine Mesh Errors \sim 20-100% in $y/D=[5-12]$
 - Overall predicted Flowfield characteristics show strong anomalies
 - Coarse mesh behaves like an unsteady RANS Model
 - Fine Mesh predicts unphysical flow field with jet splitting into a high momentum jet and a low momentum jet
 - Wrong Vortex structure – likely to result in wrong forces on a fin surface
- Time Step Refinement Study
 - Errors from $\Delta t=1.0e-6$ and $\Delta t=0.2e-6$ were compared
 - Nearly identical errors from Both simulations
 - Same anomalies seen in both simulations
 - Use of Wall functions to decrease effective CFL did not help
 - Flow structure is still wrong
 - Did not significantly decrease the error levels
- Analysis of the Model
 - Shows strong dependence on function that determines RANS vs LES regions of the flow
 - Other codes with same model show nearly identical behavior
 - Other Models with different forms show similar sensitivities
 - Explicit handle in $k-\epsilon$ Hybrid RANS-LES model to control wall distance dependence
 - We were able to explicitly demonstrate the dependence on this parameter.

39

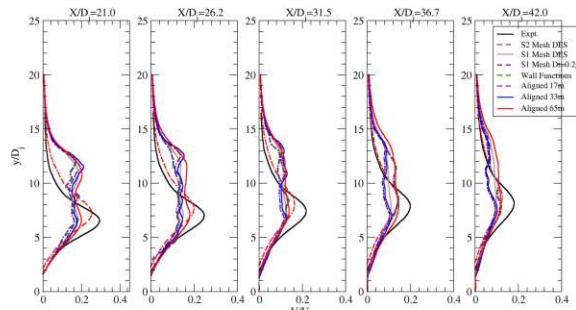
Unclassified-Unlimited Release

Overall – we did mesh refinement, time step refinement, CFL refinement
- in all the flow structure predicted was wrong
- coarse mesh behaved better than fine mesh.
- Dictated by switching function behavior.

Overall Summary

Conclusion

- Model in its existing form cannot reliably predict jet-in-crossflow flow field.
 - Strong dependence on the wall distance function needs to be better understood and calibrated or alternate methods to eliminate its unpredictable effects need to be devised.



40

Unclassified-Unlimited Release

Conclusion – model doesn't validate.

Milestone Completion



- **Completion Criteria:** Simulations of a C6 funded jet-in-crossflow experiment will be completed using the unsteady turbulence model. The resulting predictions will be used to quantify spatial and temporal convergence. Comparisons will be made between the simulations and experiment for the quantities of interest (time-averaged vortex strength and location).
 - ✓ Simulations have been carried out using the unsteady turbulence model.
 - ✓ Comparisons of quantities of interest have been presented for all the cases.
 - ✓ Spatial and temporal errors have been quantified – no convergence is seen
 - ✓ In fact, divergence is observed with mesh refinement
 - ✓ Time step refinement did not reduce errors.
 - ✓ Detailed analysis of results has been carried out
 - ✓ Different Models/ Codes/ Meshes show similar behavior
 - ✓ Node Centered Unstructured Code & Cell Centered Structured Code
 - ✓ SST-DES Model/ k- ϵ Hybrid RANS-LES Model
 - ✓ 7m / 17m / 33m / 60m / 65m cell Meshes with two different topologies

41

Unclassified-Unlimited Release

We have addressed all aspects of completion criteria.

This takes care of the letter of the milestone.

But spirit of milestone is to develop a method for LEP.

Path Forward



- Plans to address DES/Hybrid RANS-LES Models for Jet-in-Crossflow
 - Explore parametric dependence on wall distance function
 - Potential to reduce this dependence using resolved turbulence in the upstream boundary layer
 - Caveat: Feasibility of using DES/Hybrid RANS-LES for design approaching limits of usability.
 - Presents a real problem - It's not possible to cover entire parameter space with test data.
- Still Need Credible Predictive Capability for LEP
 - It is not possible to cover entire envelop using experiments/flight test data
 - Along with reliable predictions, we also need to quantify uncertainties.

42

Unclassified-Unlimited Release

Going forward we plan to address that by examining this switching function behavior.

Potentially use of turbulence in inflow can anchor switching behavior.

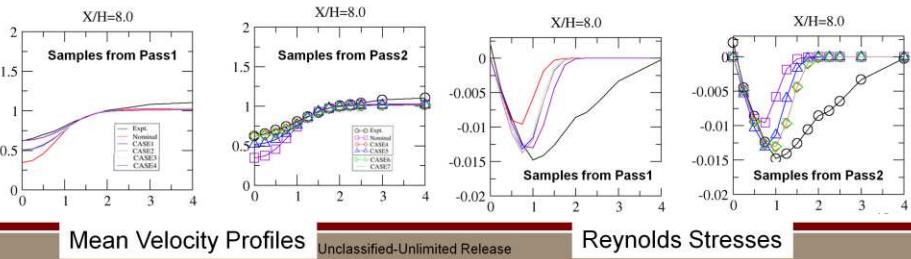
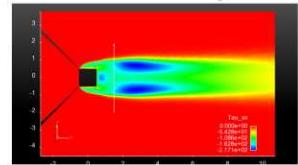
But we must be cost aware.

Also for LEP in the multi-fidelity approach, we need UQ.

Path Forward



- FY 13 V&V Project: UQ of Flow Predictions(65755/004.05.01)
 - In collaboration with Jaideep Ray and Sophia Lefantzi (8954)
 - Goal : **Construct a statistical infrastructure to perform Bayesian calibration of RANS turbulence models using surrogates and MCMC.**
- Deterministic Calibration
 - Carried out for Flow Past a Square Cylinder
 - Experimental Data available for mean velocities and turbulence quantities.
 - Has some similarities to Jet-in-Crossflow flowfield.
 - Pass 1 : Used Only Experimental Reynolds Stress profiles
 - Pass 2 : Used Turbulence Intensities and Reynolds Stresses
 - Yielded a global minimum and a few local minima with comparable cost function values.
- RANS Calculations using these parameters
 - Shows only slightly improved Reynolds Stress Predictions
 - But vastly improved mean flow field predictions!



For this we undertook another project.

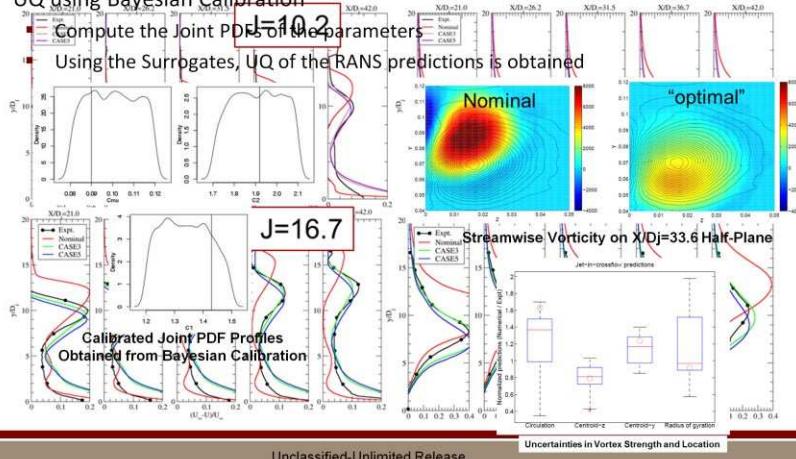
Deterministic calibration using surrogates gives us more optimal parameters
we are able to get better mean flow predictions using this although Re.Stress doesn't
improve much.

Path Forward

- Jet-in-Crossflow Predictions using Calibrated Parameters
 - Tested for Mach 0.8 : $J=10.2$ and $J=16.7$
 - Significantly Improved results are seen for both cases.

- UQ using Bayesian Calibration

- Compute the Joint PDFs of the parameters
 - Using the Surrogates, UQ of the RANS predictions is obtained



These parameters applied to jet in crossflow yield much more promising results than any other method!!

But best thing is we can also get UQ using bayesian calibration for joint PDF and surrogates.

Summary: Path Forward



- DES/Hybrid RANS-LES Models
 - Address Switch function dependency
 - Understand and Calibrate behavior
 - Explore Alternate models
 - Alternate methods (unsteady BCs etc.) to eliminate unpredictable behavior.
- Better Calibrated RANS Turbulence Models
 - Potential to Yield Better results
 - Need to answer several questions before this can be used in production simulations
 - Points to a Potential Approach for predictive capability
 - Advantage: Predictive capability comes with UQ capability.
 - Disadvantage: Need to develop confidence in approach before it can be used for production calculations.

45

Unclassified-Unlimited Release

So in summary, path forward is to

- Address DES issues
- Try and use better calibrated RANS models.

- Thanks
- Questions?

CPU Times



- Conchas
 - Simulation at 1e-6 s needs 100,000 steps
 - S1 (65m Nodes): 1536 Cores (chama): 5000steps : 22 hrs => 20 days
 - WF -S1 (43m Nodes) : 2048k cores (cielo): 8000 Steps : 23 hrs: 12.5 days
 - WF - S1 (43m Nodes): 8192 cores (Cielo): 20000 steps : 23 hrs: 5 days
 - Aligned 21m : 2048 k cores (Cielo): 7000 steps 19 hrs.: 15 days
 - Turnaround time for all the runs including wait time ~40days.