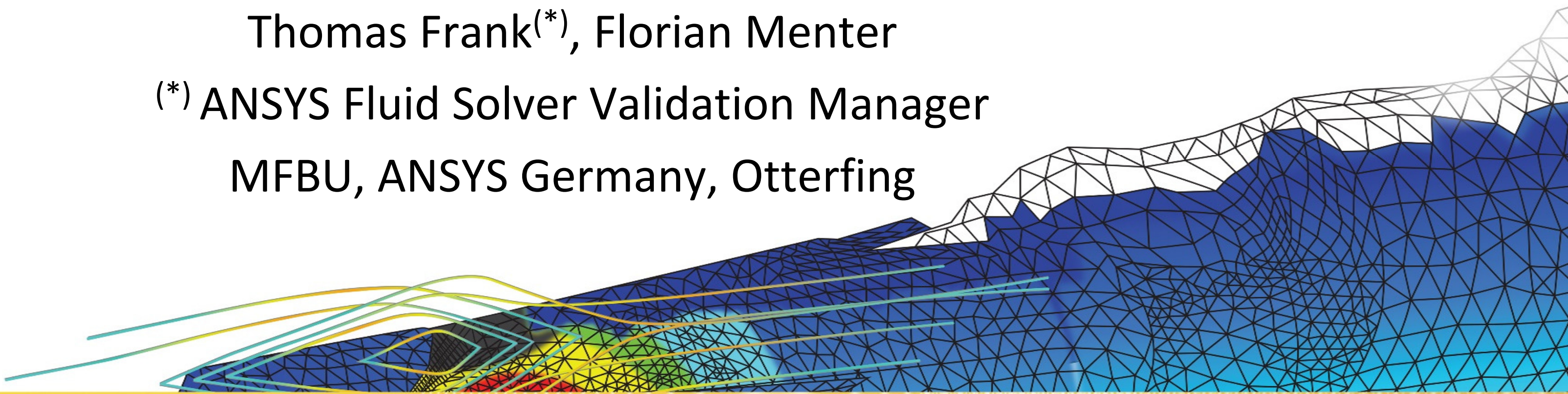
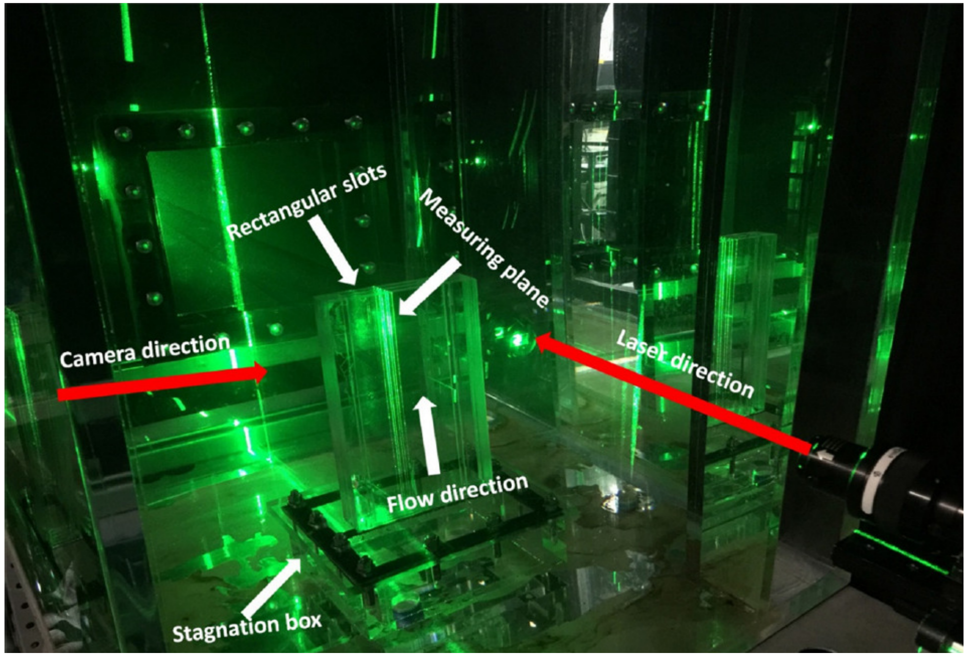
Validation of URANS SST and SBES in ANSYS CFD for the Turbulent Mixing of Two Parallel Planar Water Jets Impinging on a Stationary Pool

Thomas Frank(*), Florian Menter
(*) ANSYS Fluid Solver Validation Manager
MFBU, ANSYS Germany, Otterfing



Content

- Introduction – the ASME V&V 30 Benchmark Problem 1
- Best practice related analysis of the benchmark problem:
  - Geometry and experimental uncertainties
  - Boundary conditions
  - Steady vs. transient flow analysis
  - Selection of turbulence model: SST vs. SBES
  - Mesh independency of results
    (2 hex meshes : 6.5M & 54.4M hex elements)
- Cross-comparison of ANSYS CFX, ANSYS Fluent and AIM Fluids solvers with data
- Summary & conclusion



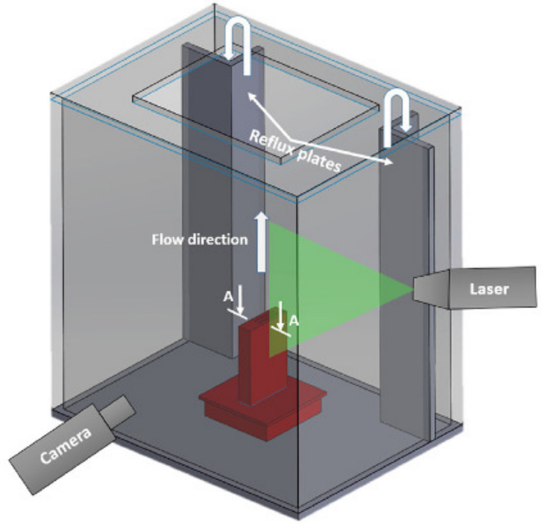
H. Wang, S. Lee, Y.A. Hassan: "*Particle Image Velocimetry measurements of the flow in the converging region of two parallel jets*", Journal Nuclear Engineering & Design (2015), pp. 1-9

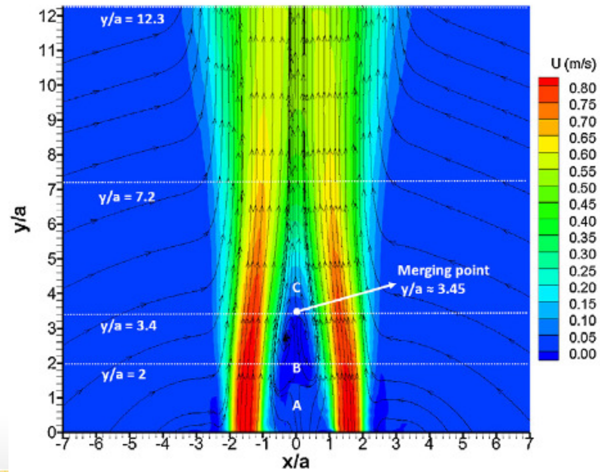
Introduction

- ASME Verification & Validation Symposium – since 2009
  → V&V 30 Standardizing Committee:
  Verification and Validation in Computational Simulation of Nuclear System Thermal Fluids Behavior
- ASME V&V 2016: publication of a call for participation for a first flow benchmark (Jan. 2016 → May 2016)
- Benchmark sessions on ASME V&V Symposiums in 2016 & 2017
- Intention to continue this series of flow benchmark problems with the goal to foster exchange on applied CFD best practices amongst experts in this field of CFD applications
  → Contribution to formulation of related ASME V&V Standards for Computational Fluid Dynamics (CFD) Investigations

The ASME V&V Benchmark Problem 1
- Turbulent Mixing of Two Parallel Planar Water Jets Impinging on a Stationary Pool -

- Underlying experiment donated by Yassin A. Hassan, Texas A&M University and A.E. Ruggles, Univ. of Tennessee, Knoxville
- Two parallel planar jets impinging on a stationary pool of water and being subject to turbulent mixing
  → relevance for conditions in the upper plenum of advanced liquid metal-cooled reactors or for VHTR lower plenum
- Experimental conditions:
  - Working fluid: Water at 23°C and ambient pressure
  - Planar nozzle cross sections: $a \times l$ = 5.8mm × 87.6mm
  - Nozzle height: h = 279.4mm ~ 48·a
  - Reynolds number: Re = 9100
  - Average Inlet velocity: V_{in} = 0.75 m/s
  - Inlet mass flow rate per jet: $\dot{m}_F$ = 0.385 kg/s
  - Average inlet turb. intensity: Tu = 5.3 %

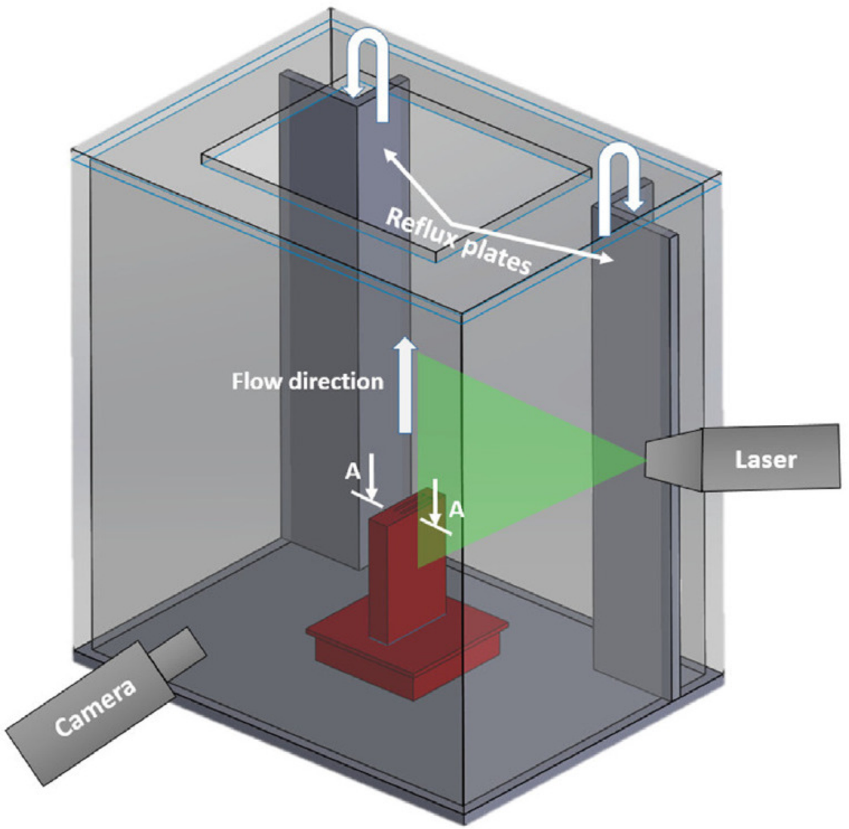


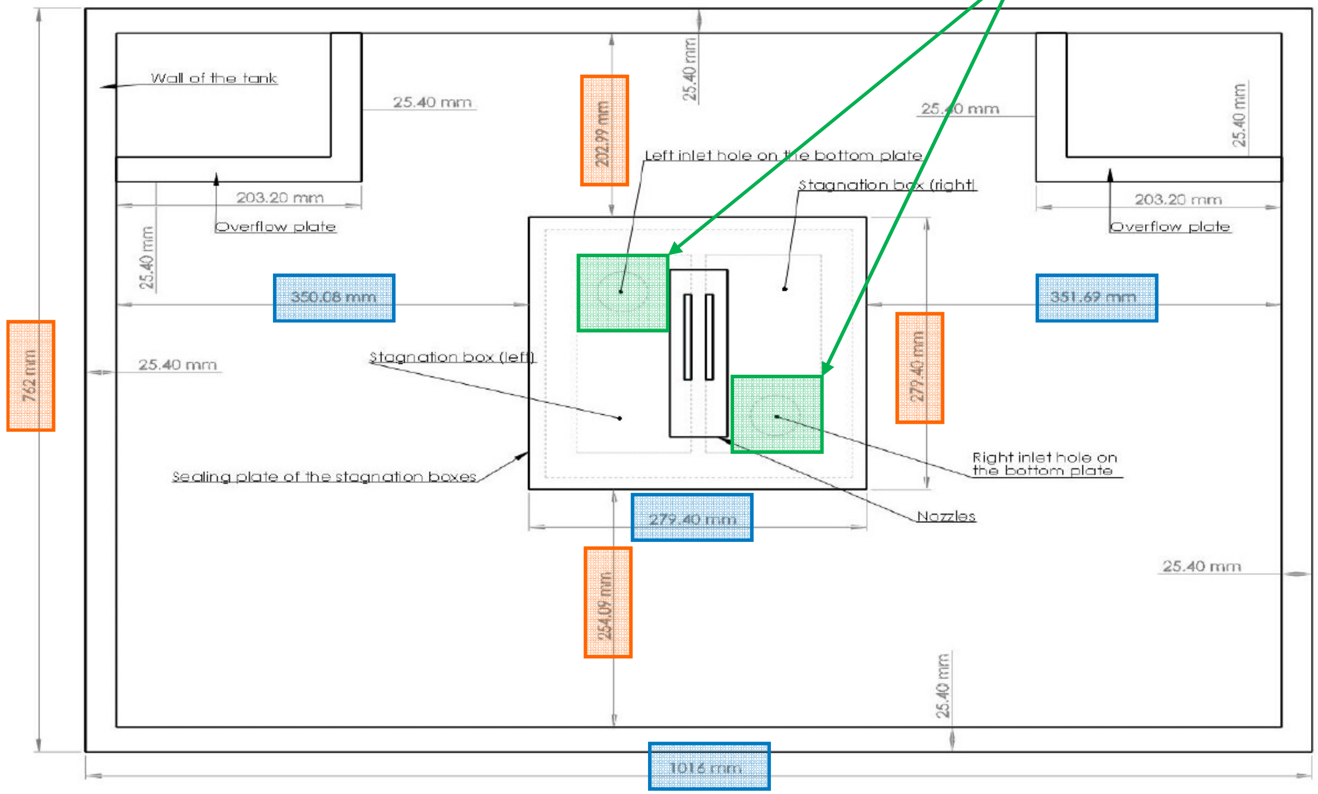


Best Practice Related Analysis of the Benchmark Problem Geometry and Experimental Uncertainties

Non-symmetric placement in accordance with the measures in the drawings

- Provided geometry and geometrical uncertainties:





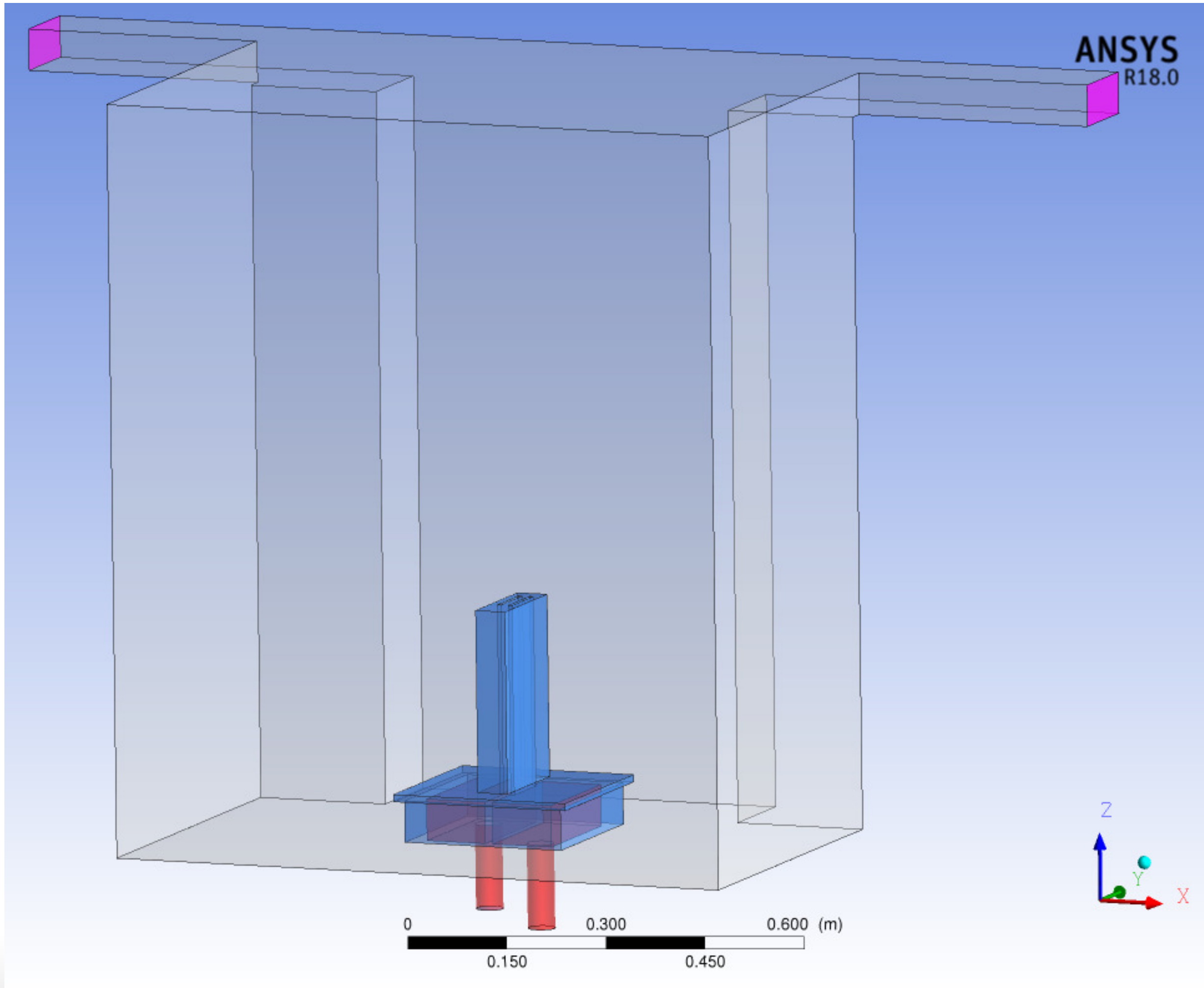
Best Practice Related Analysis of the Benchmark Problem
Boundary Conditions

- **Experiment:**
  Free surface flow with overflow baffles

  CFD:
  → Closed water filled box,
  Top wall = no-slip wall BC (could be free slip),
  Attached outlet channels to avoid backflow

- **Benchmark:**
  Steady-state inlet BC at nozzle inlets
  Provided measurements for axial velocity and turbulence kinetic energy profiles

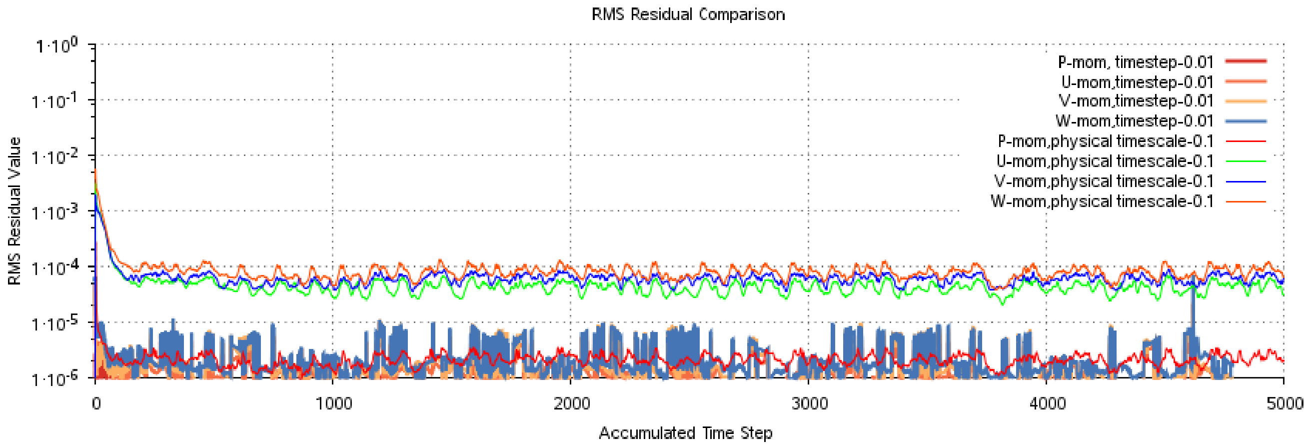
  CFD:
  → Pipe inlets to lower stagnation boxes,
  D=41.4mm resulting in L/D~5,
  Inlet massflow rate BC,
  Inlet turbulence intensity adjusted, so that profile BC conditions at nozzle exits are fairly well matched



Best Practice Related Analysis of the Benchmark Problem Steady-state vs. Transient

- Convergence for a steady-state simulation – strictly speaking – can only be obtained on rather coarse mesh level 1
- Convergence level for URANS SST is very satisfactory
- PIV velocity vector images show strongly transient jet behavior in the close vicinity of the jet nozzles.

ANSYS CFX, Mesh 1, Convergence of Steady-state SST Simulation vs. Transient SST Simulation



- Inherently transient flow in stagnation chambers upstream the nozzle exits.

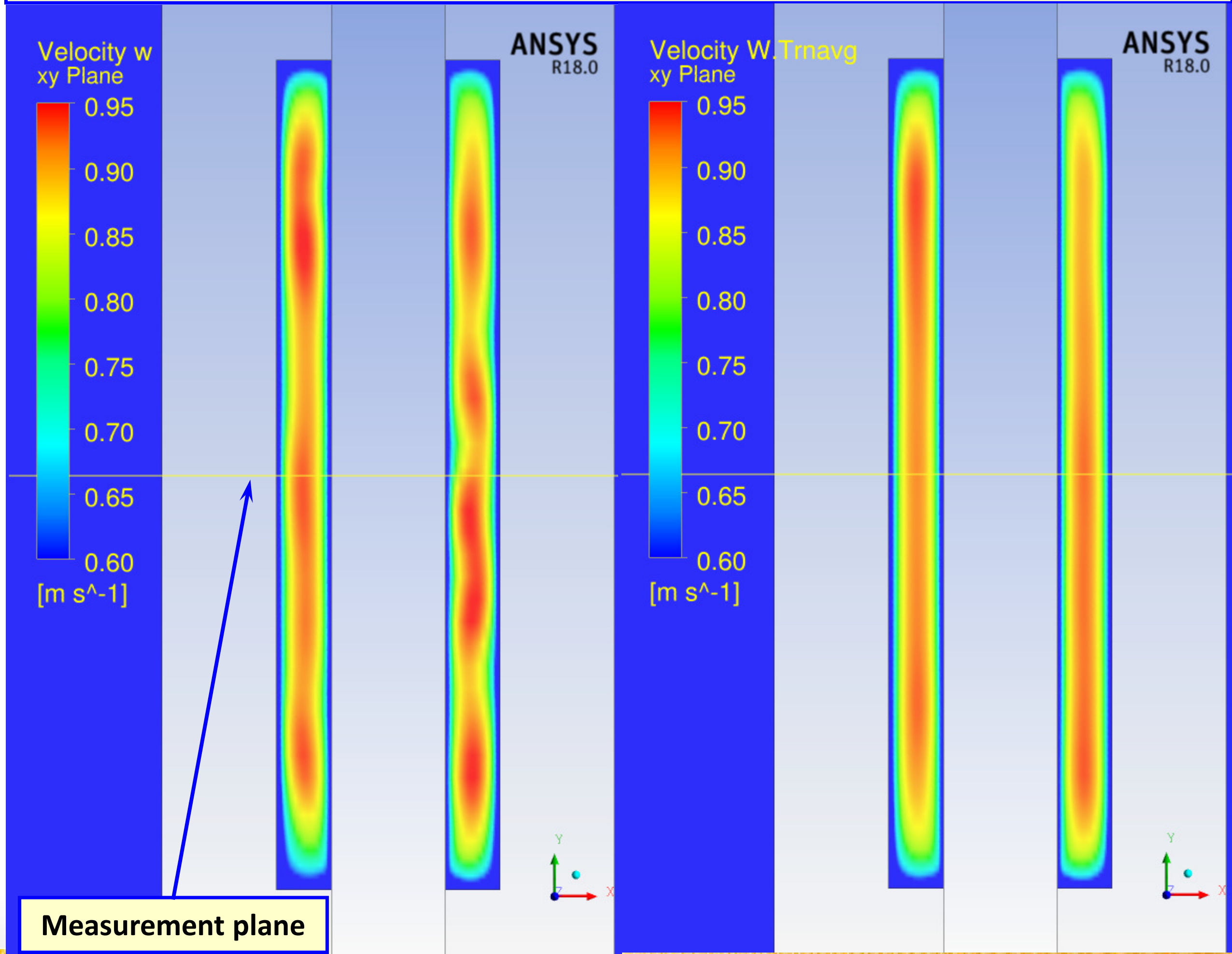
→ All further simulations are carried out as transient, time-averaged SST or SBES simulations. URANS SST simulations allow for a substantially larger timestep, since we do not have to resolve all turbulent scales in space and time (no SRS quality criteria).

Best Practice Related Analysis of the Benchmark Problem
Steady-state vs. Transient

- Comparison of instantaneous and time-averaged W-velocity distribution @ nozzle exit cross section shows:
  - Non-homogeneous velocity distribution over the whole cross section in the instantaneous velocity field.
  - This non-homogeneous vel. distribution changes over time!
  - But even slightly non-symmetric velocity distribution in the time-averaged velocity field due to the location of the flow inlets and stagnation chambers.

→ Strong requirement for time-averaging in order to compare to PIV / LDA data

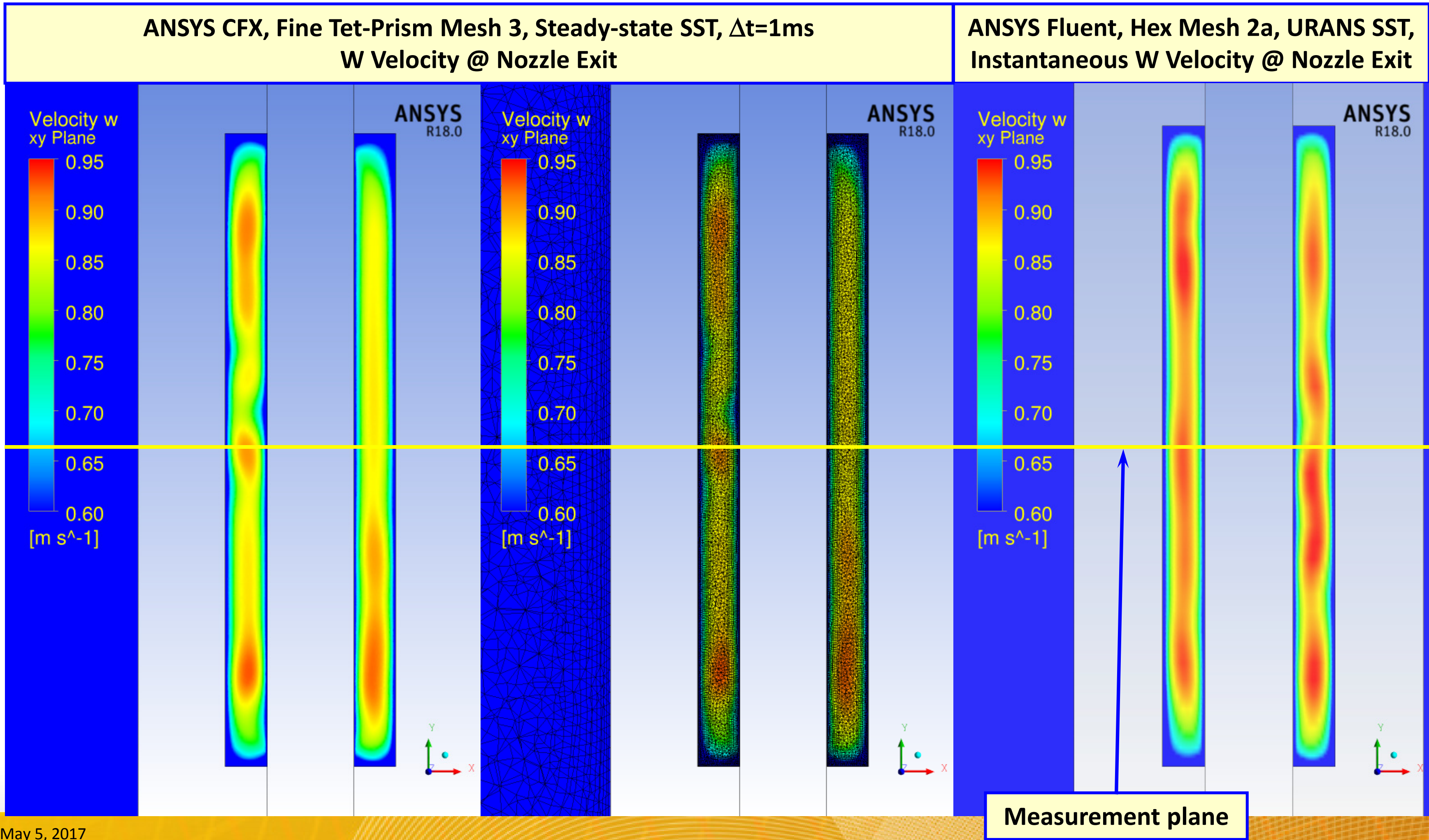
ANSYS Fluent, Mesh 2a, URANS SST, Instantaneous vs. Averaged W Velocity @ Nozzle Exit



Best Practice Related Analysis of the Benchmark Problem
Mesh Type Influence on Flow Inhomogeneities at Nozzle Exit

- In very first simulation attempts it was tried to use tet-prism meshing and steady-state SST simulation to predict the twin jet mixing.
- In this case a mesh with:
  - 58.9 Mill. Elements
  - 22.5 Mill. Nodes
  - 23.7 Mill. Tetrahedrons
  - 35.2 Mill. Prisms
  - 222 Pyramids

  has been used.
- RMS Res<10^{-4} has been achieved.
- Strong flow inhomogeneities do not allow a reasonable comparison to data.



ASME V&V Benchmark – Mesh Hierarchy and Mesh Statistics

Software Used for Mesh Generation: ANSYS ICEM CFD 17.0/ANSYS ICEM CFD 17.1

Mesh Name	Number of Nodes	Number of Elements	Minimum Orthogonal Angle [deg.]	Node Count on 'a' (Nozzle Width)	Maximum y^+ in the whole domain / Solver	Maximum y^+ on the pedestal wall / Solver	Average y^+ on the pedestal wall/Solver
Mesh 1	5.985.456	5.854.065	18.7°	15	256.86/CFX	168.43/CFX	15.79/CFX
Mesh 1+	6.606.240	6.465.838	25.7°	15	132.66/CFX	132.67/CFX	13.51/CFX
Mesh 2	60.324.396	59.650.628	24.57°	40	275.421/CFX	51.97/CFX	3.776/CFX
Mesh 2+	64.239.912	63.535.856	24.57°	40	125.30/CFX	51.10/CFX	2.59/CFX
Mesh 2a	54.898.488	54.354.804	25.8°	40	49.60/Fluent	31.86/Fluent	3.26/Fluent

Mesh 1
- Coarse mesh for the ANSYS CFX 17.1 SST solution

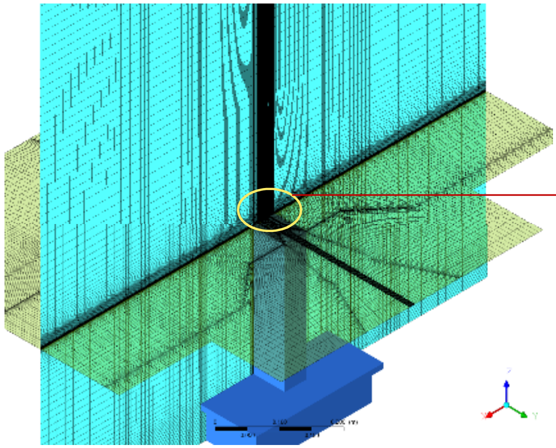
Mesh 1+
- Coarse mesh for the ANSYS CFX 17.1 SBES solution
- Coarse mesh for the ANSYS Fluent 17.1 SST solution
- Coarse mesh for the AIM Fluids 17.1 SST solution

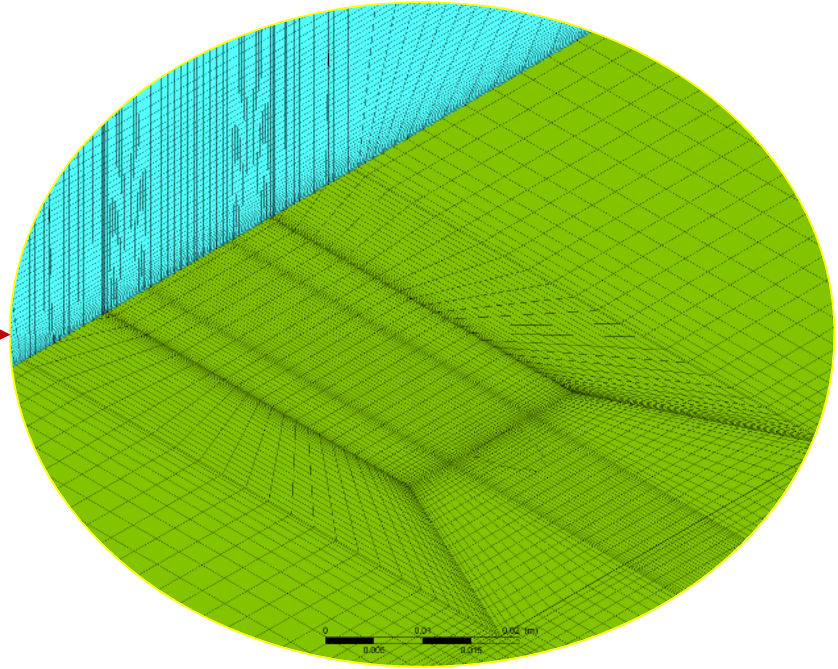
Mesh 2, Mesh2+
- Fine mesh for ANSYS CFX 17.1 SST solution (Mesh2)
- Fine mesh for ANSYS CFX 17.1 SBES solution (Mesh 2+)

Mesh 2a
- Fine mesh for the ANSYS Fluent 17.1 SST solution
- Fine mesh for the ANSYS Fluent 17.1 SBES solution
- Fine mesh for the AIM Fluids 17.1 SST solution

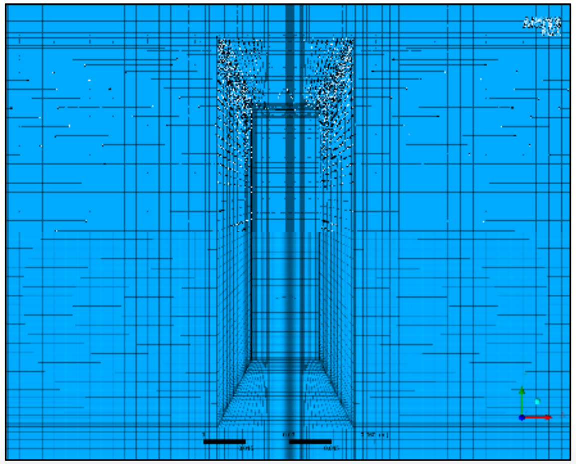
- **The transition from Mesh 1 to Mesh 1+ has led to an increase in the cell count, exclusively due to the outlet BC modification (outlet channels). We shall refer to both meshes as Mesh 1 for the rest of the V&V Study.**
- **A similar increase in cell count has taken place from Mesh 2 to Mesh 2+. We shall refer to both the meshes as Mesh 2 for the rest of the V&V study.**

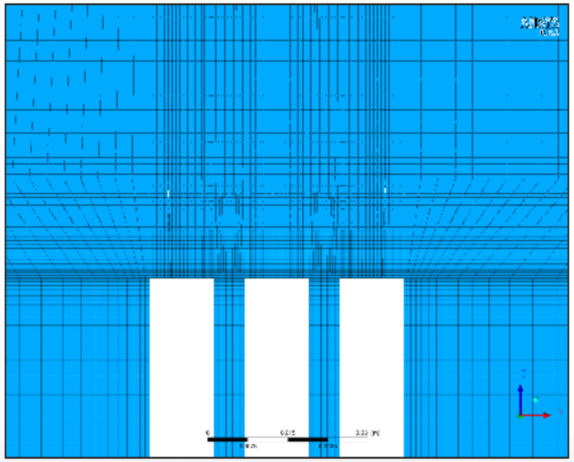
ASME V&V Benchmark – Mesh 1



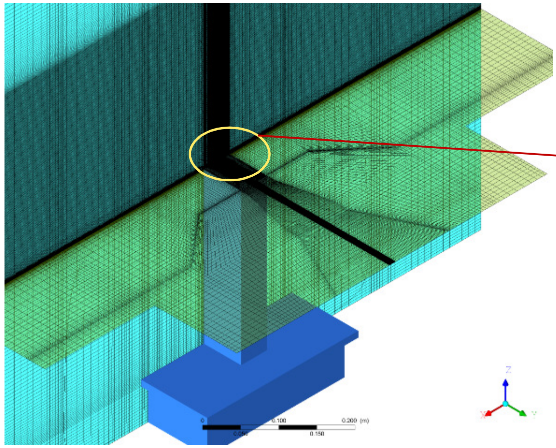


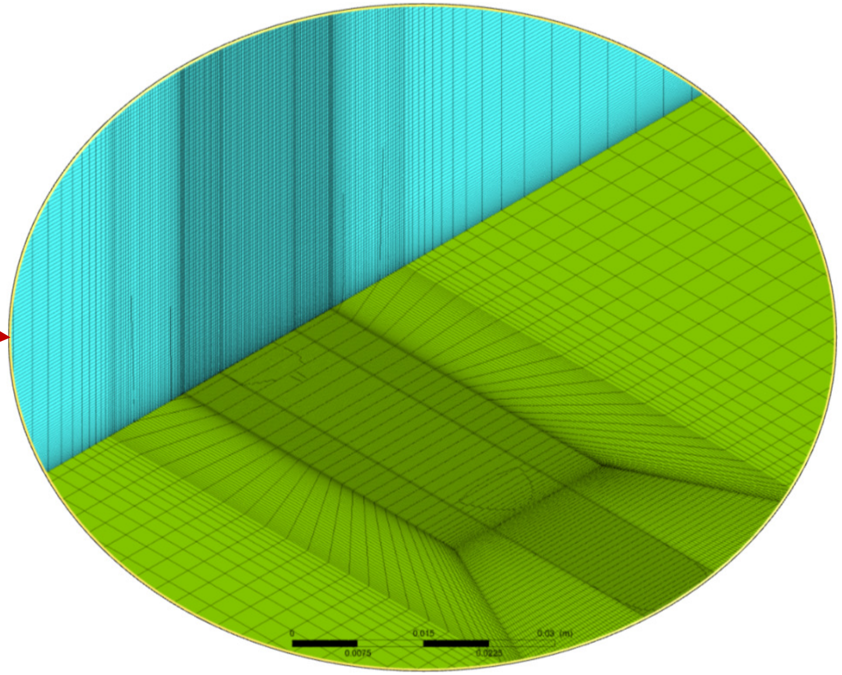
Total Number of Nodes: 5.985.456
Total Number of Elements: 5.854.065



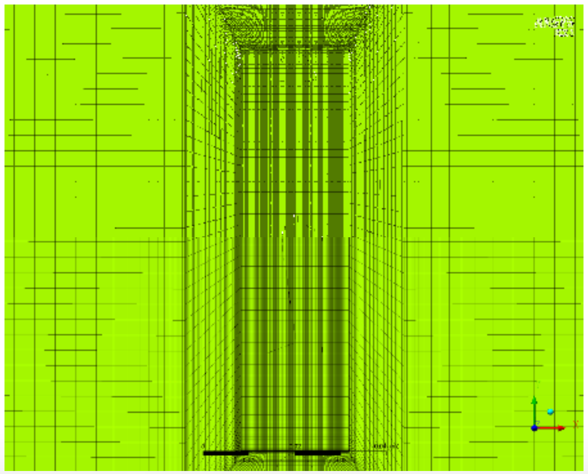


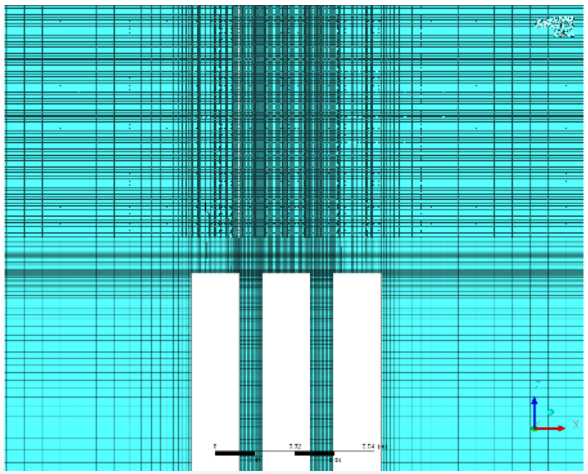
ASME V&V Benchmark – Mesh 2a



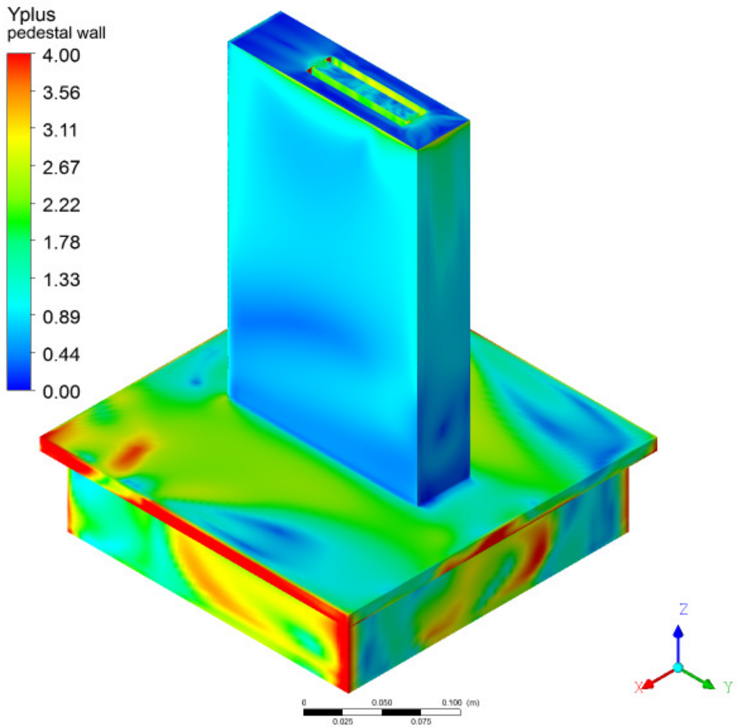


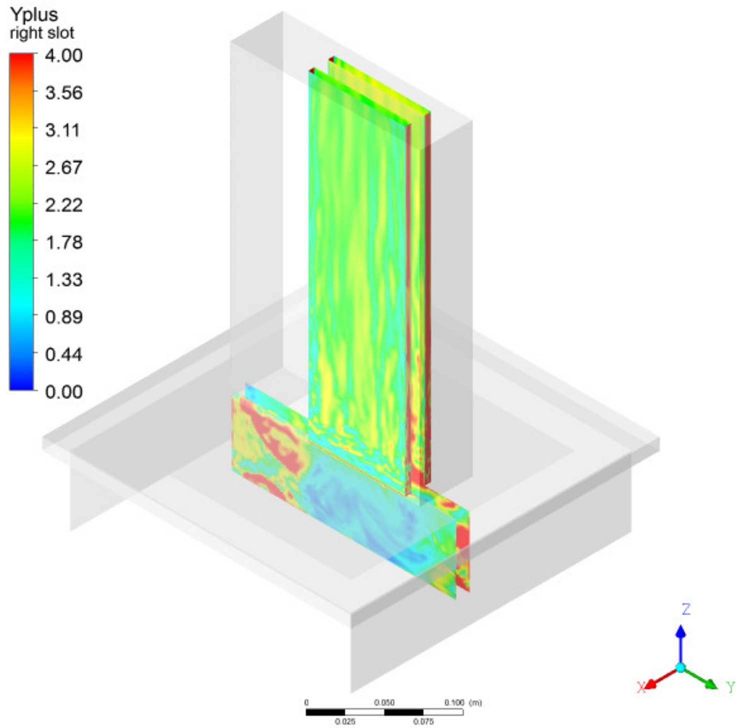
Total Number of Nodes: 54.898.488
Total Number of Elements: 54.354.804





ASME V&V Benchmark – Mesh 2a Detail
y^+ at the Pedestal Walls and the Two Nozzle Slots

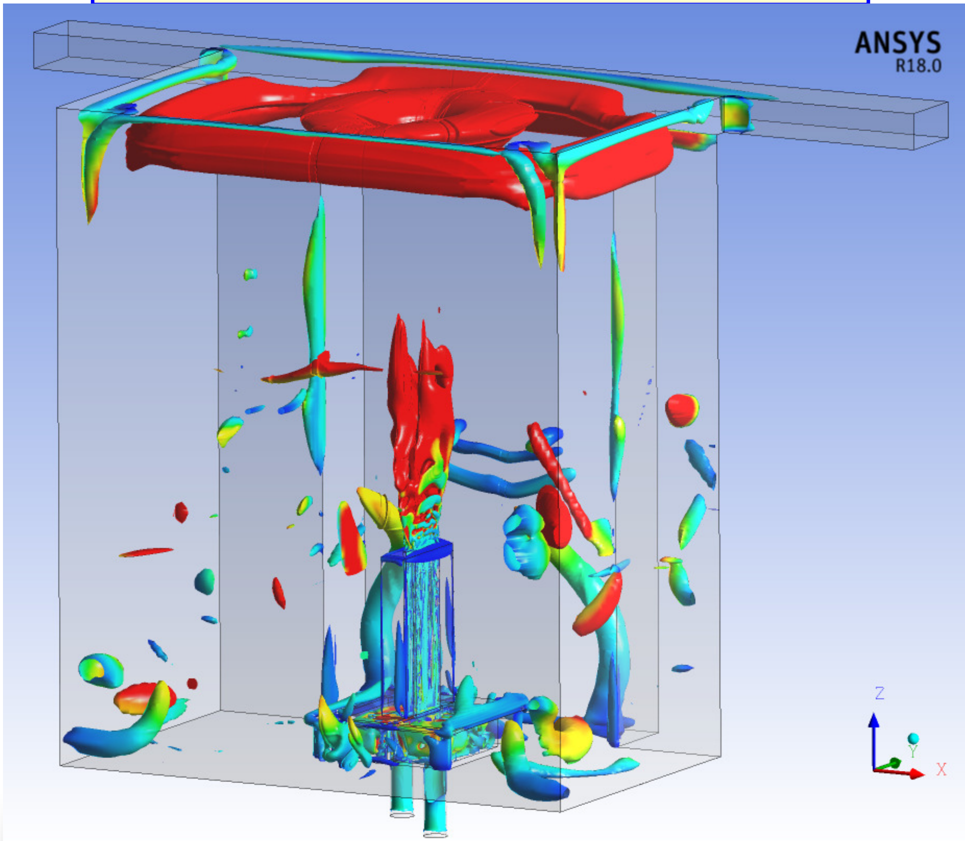




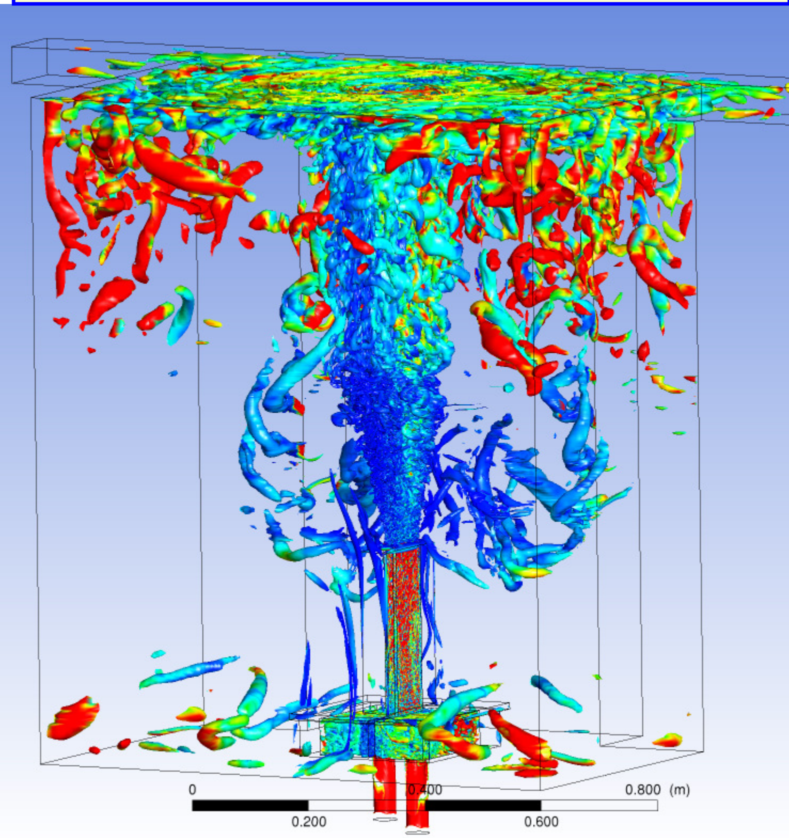
The image shows y^+ distribution on the nozzle and pedestal from ANSYS Fluent, URANS SST, Mesh2a simulation. The corresponding y^+ for AIM Fluids solver is nearly identical, The corresponding y^+ for ANSYS CFX solver will be slightly different due to different discretization (cell vs. vertex centered), approx. two times higher values.

Best Practice Related Analysis of the Benchmark Problem URANS SST vs. Stress-Blended Eddy Simulation (SBES)

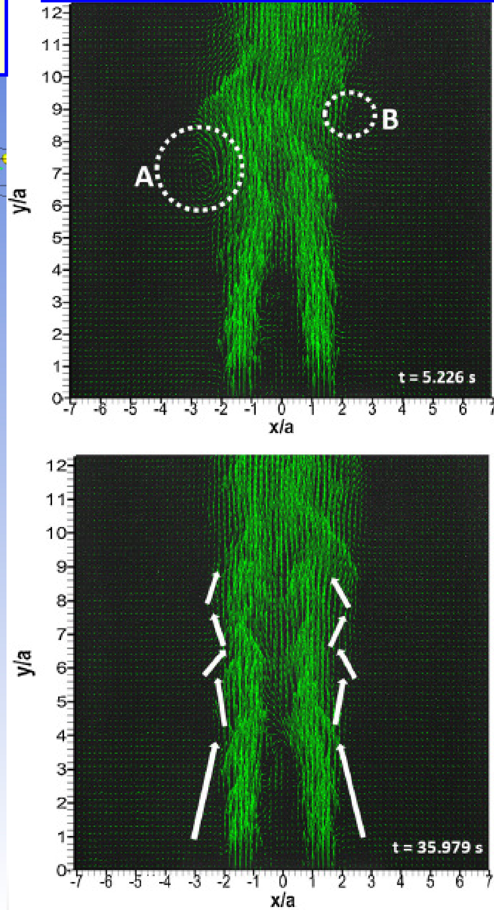
ANSYS Fluent 17.1, SST, Mesh-2a, 10.000 Timesteps, Δt=10 ms, Q-criterion (Q=0.0005) colored by EVR, EVR color range [0, 50]



ANSYS Fluent 17.1, SBES, Mesh-2a, 62.000 Timesteps, Δt=0.25 ms, Q-criterion (Q=0.001) colored by EVR, EVR color range [0, 5]



PIV Experiments by Wang, Lee, Hassan (2015)

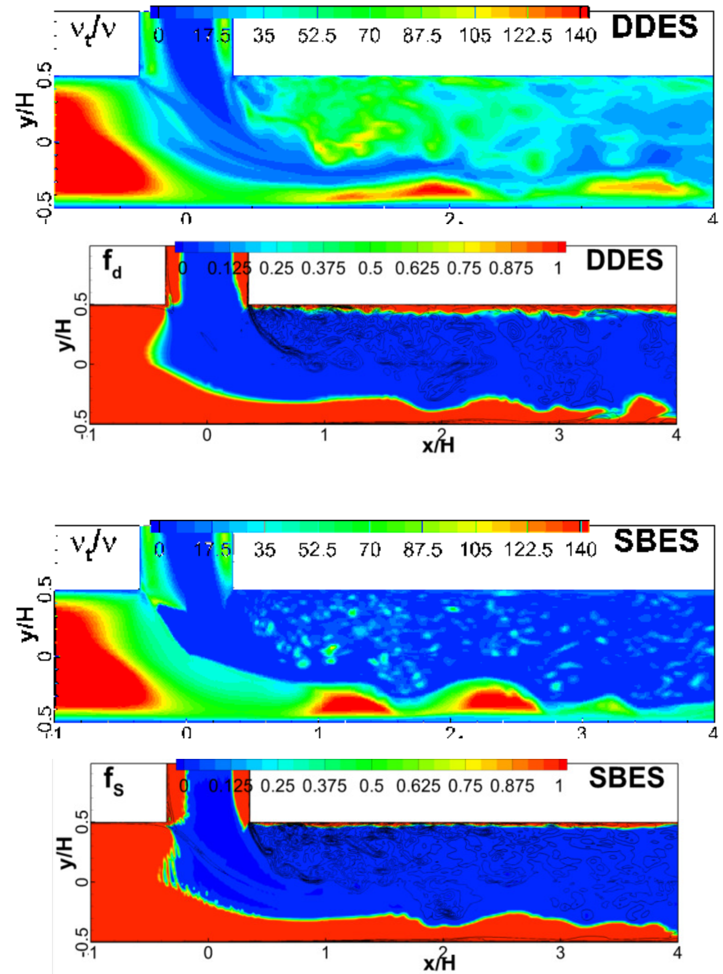


Stress Blended Eddy Simulation (SBES) in a Nutshell

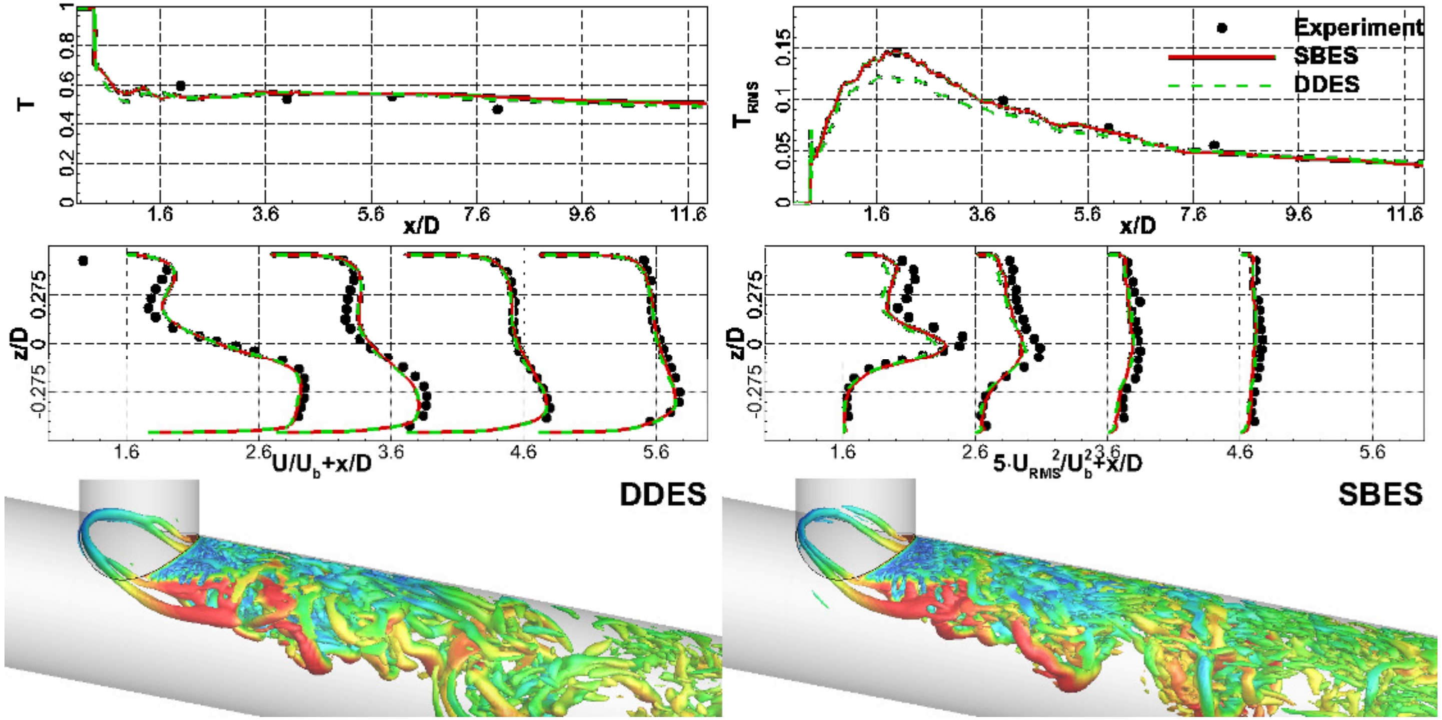
- SBES = Stress-Blended Eddy Simulation
  - A hybrid URANS-LES model similar to SAS-SST
  - Based on an improved asymptotic shielding of RANS boundary layer against LES modification
  - Able to blend the eddy-viscosity (or the Reynolds Stress) between a RANS and LES formulation

$$\nu_t^{SBES} = \nu_t^{RANS} \cdot f_{SBES} + \nu_t^{LES} \left(1 - f_{SBES}\right)$$

  - Produces substantially lower eddy-viscosity in separating shear layers (e.g. compared to SAS)
  - Allows the combination of essentially any URANS turbulence model with existing LES model in the zone, where LES is detected by the shielding function:
    - WALE model
    - Smagorinsky model, etc.

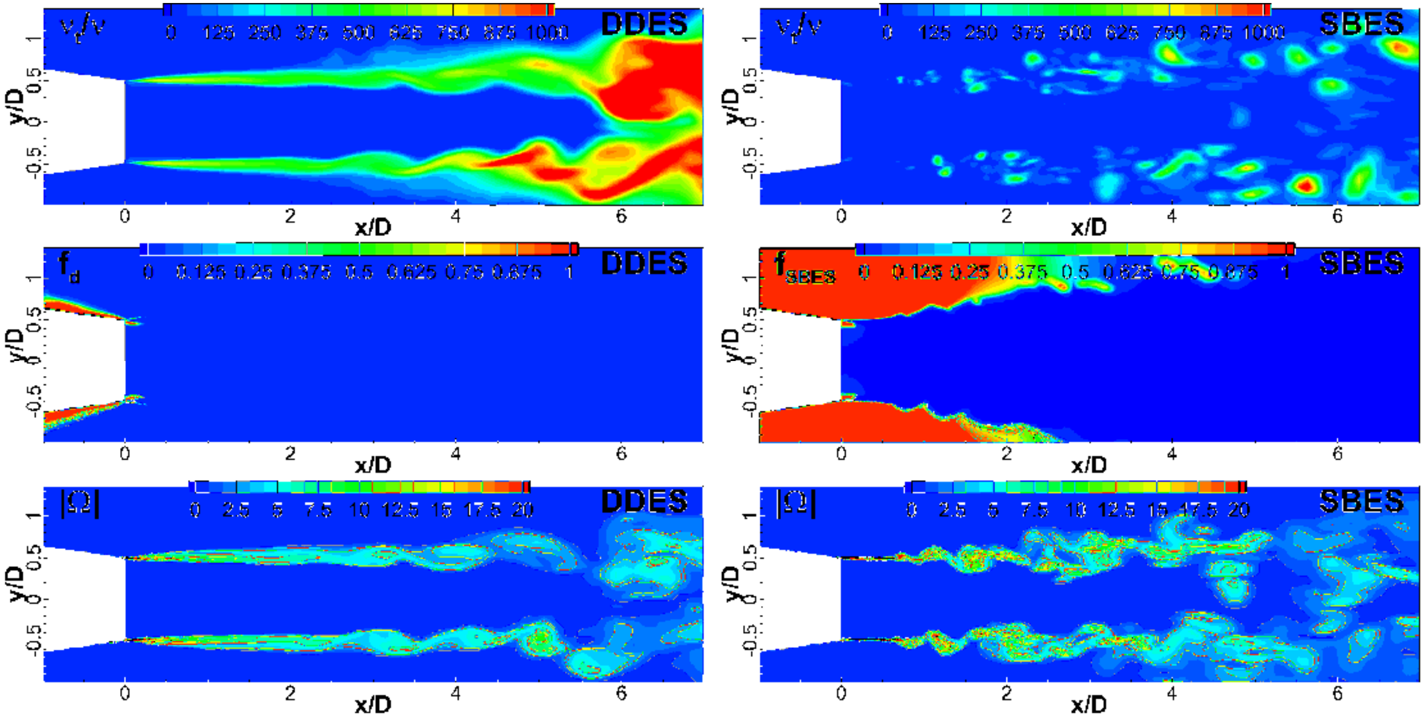


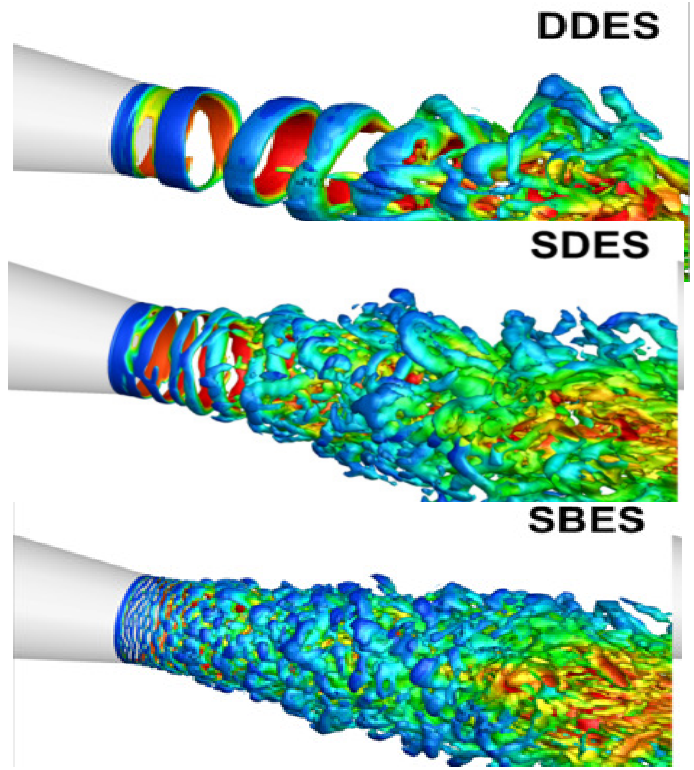
Stress Blended Eddy Simulation (SBES) in a Nutshell (cont.)
- Thermal Mixing in a T-Junction (OECD/NEA Benchmark) -



- The mean and RMS velocity profiles of SBES are close to those of DDES
- SBES provides better agreement with the experiment for the RMS wall temperature

Stress Blended Eddy Simulation (SBES) in a Nutshell (cont.)
- SRS of a Subsonic Round Jet -

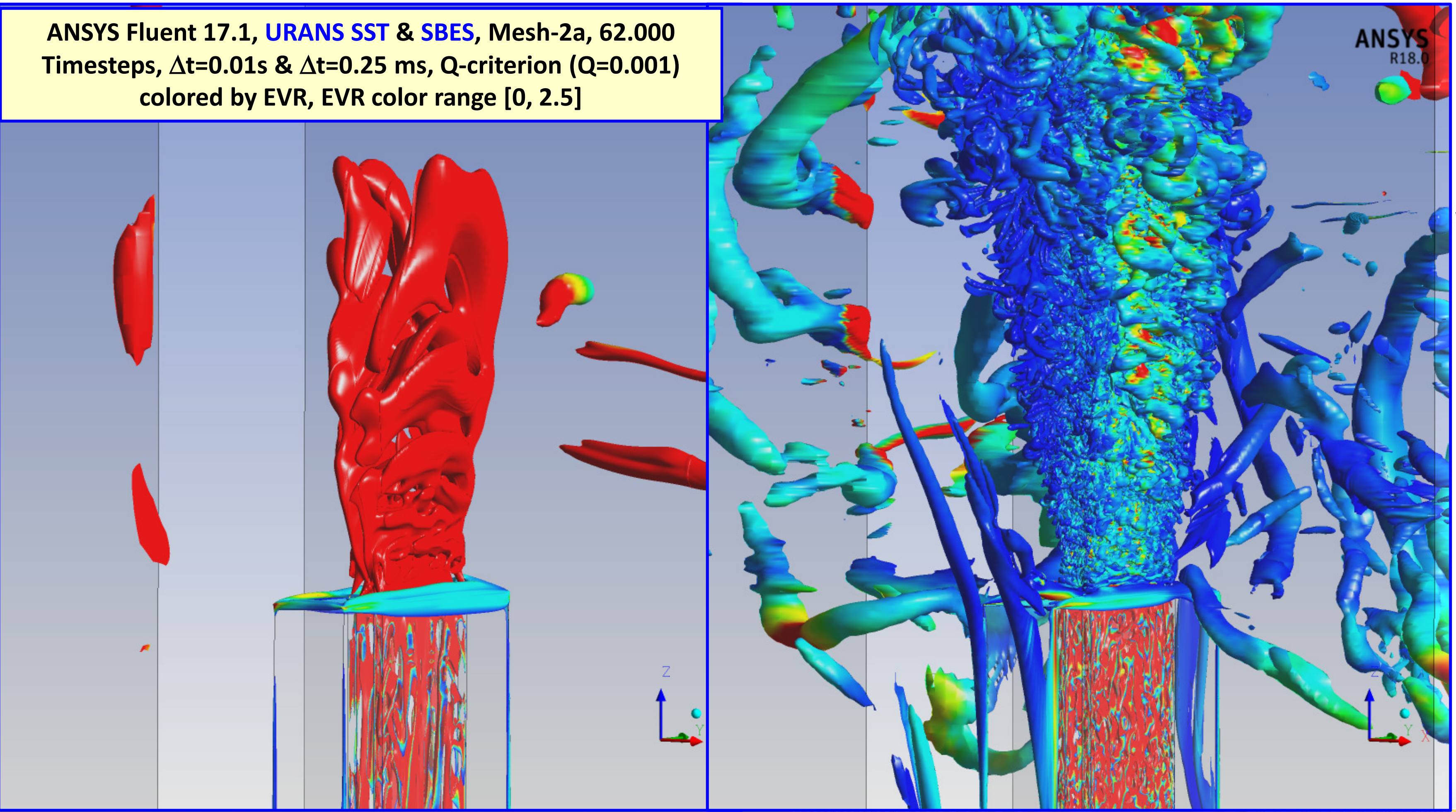




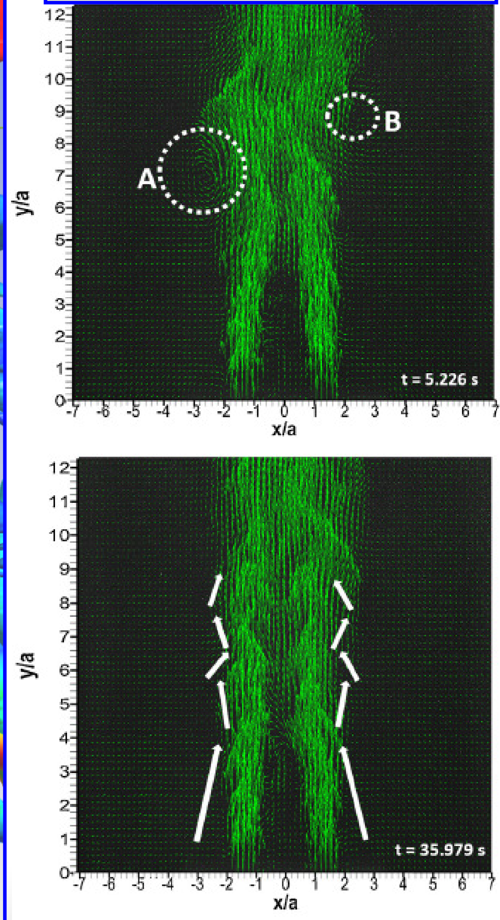
- DDES has larger eddy viscosity in the mixing layer which suppresses the development of 3D turbulence
- SBES shows less eddy viscosity and fast development of turbulent scales

Best Practice Related Analysis of the Benchmark Problem URANS SST vs. Stress-Blended Eddy Simulation (SBES)

ANSYS Fluent 17.1, URANS SST & SBES, Mesh-2a, 62.000 Timesteps, Δt=0.01s & Δt=0.25 ms, Q-criterion (Q=0.001) colored by EVR, EVR color range [0, 2.5]

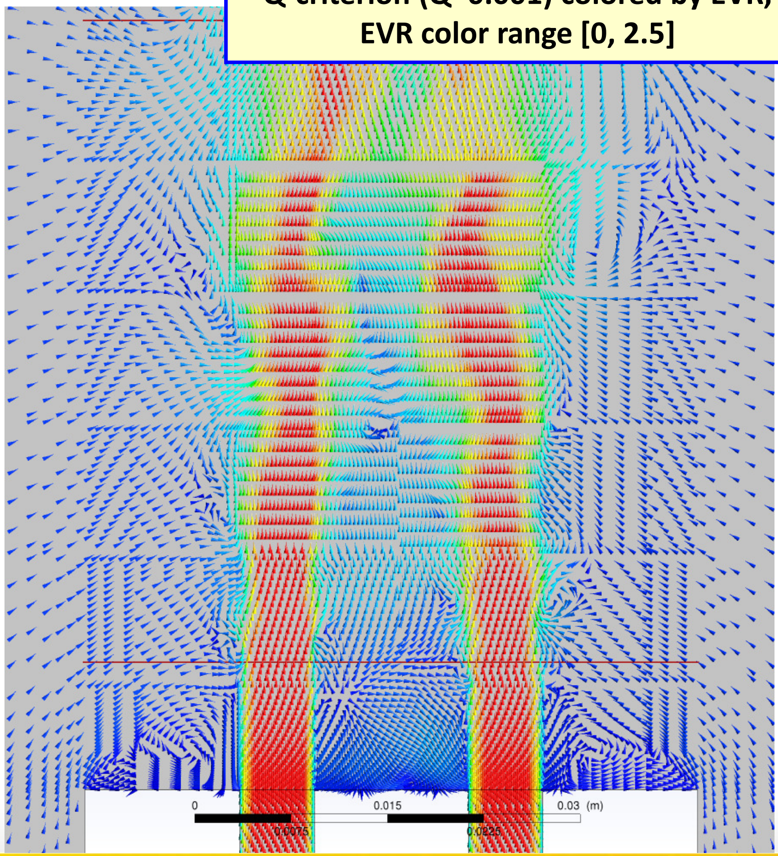


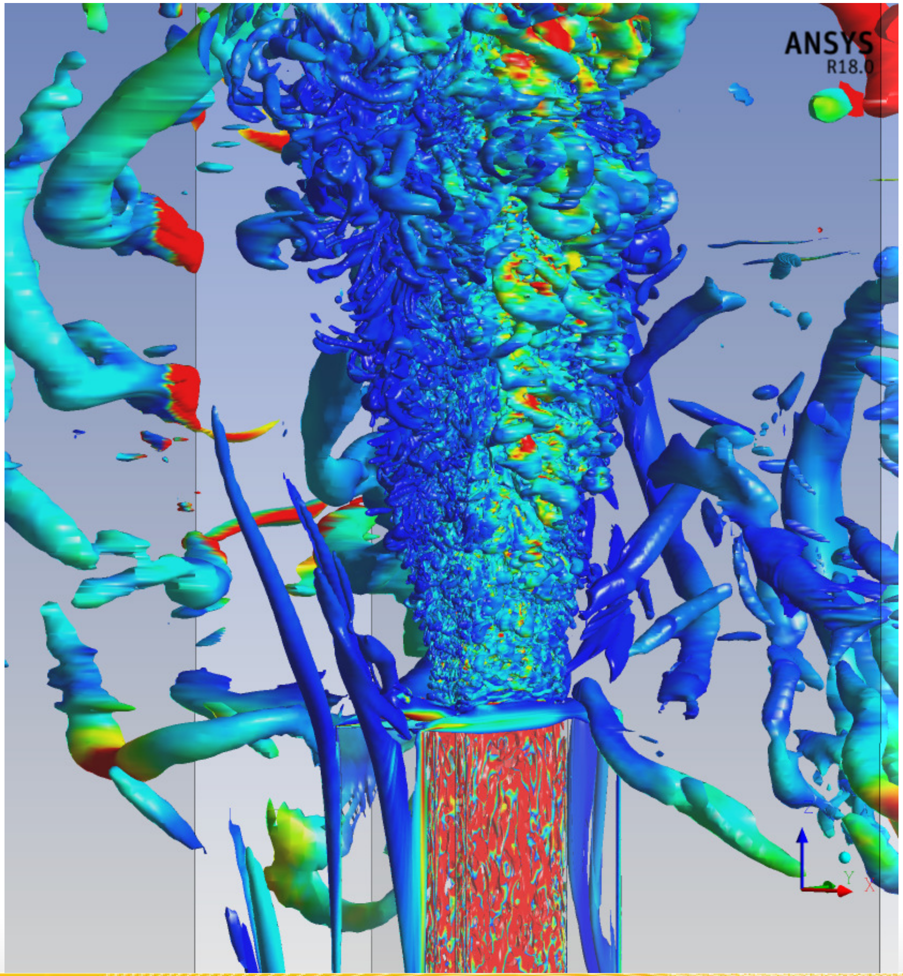
PIV Experiments by Wang, Lee, Hassan (2015)



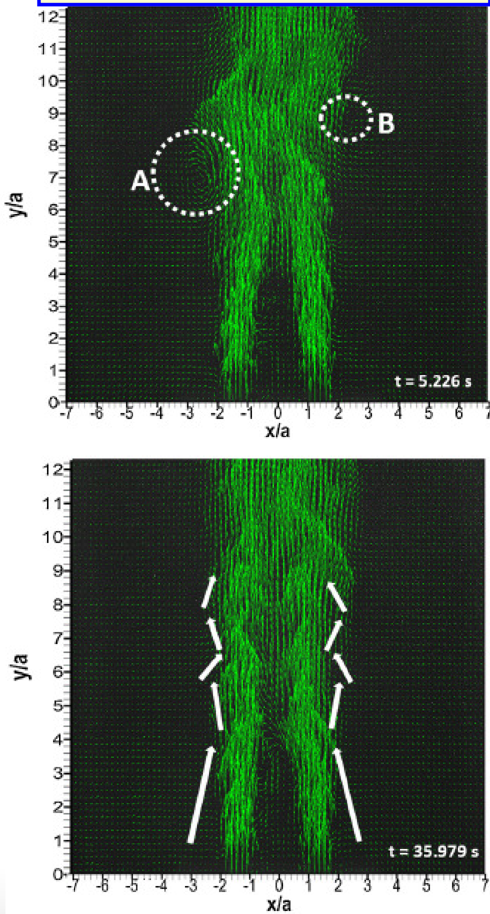
Best Practice Related Analysis of the Benchmark Problem URANS SST vs. Stress-Blended Eddy Simulation (SBES)

ANSYS Fluent 17.1, SBES, Mesh-2a, 62.000 Timesteps, Δt=0.25 ms, Q-criterion (Q=0.001) colored by EVR, EVR color range [0, 2.5]





PIV Experiments by Wang, Lee, Hassan (2015)



ASME V&V Benchmark – CFD Setup used with ANSYS CFX 17.1

ANSYS CFD Solver: ANSYS CFX 17.1

Mesh	Mesh 1		Mesh 2	
Analysis Type	Steady-state	Transient	Transient	
Turbulence model	SST	SBES	SST	SBES
Solver Settings				
Advection Scheme	High resolution	Bounded Central Difference	High Resolution	High resolution
Turbulence Numerics	High resolution	High resolution	High Resolution	High resolution
Physical timescale/Timestep	0.1s	5×10^{-4} s	0.1s / 0.01s	2.5×10^{-4} s
Convergence control	---	Min No. of co-eff loops: 1 Max No. of co-eff loops: 3	Min No. of co-eff loops: 1 Min No. of co-eff loops: 10	Min No. of co-eff loops: 1 Max No. of co-eff loops: 3
Flow Development time steps	---	11000	10000	2000 (initialized with Mesh1 SBES)
Transient Averaging time steps	---	20000	5000	20000
Transient Scheme	---	Second Order Backward Euler	Second Order Backward Euler	Second Order Backward Euler
Conservation Target	0.001	0.001	0.001	0.001
Residual Target	RMS Res<10^{-5}	Max Res<10^{-3}	RMS Res<10^{-5}	Max Res<10^{-3}

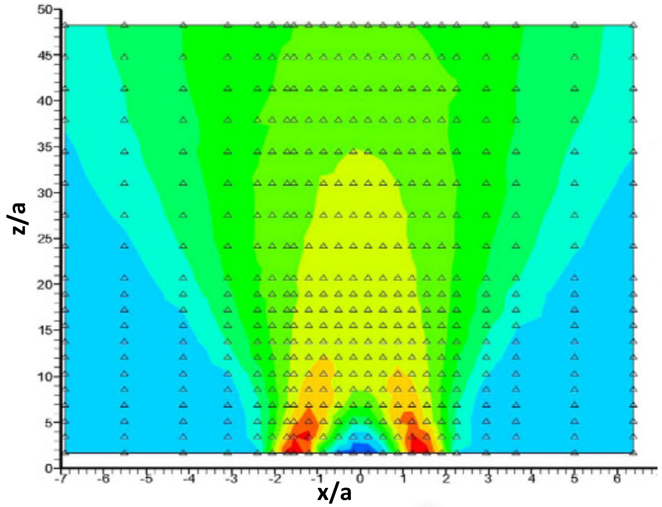
ASME V&V Benchmark – CFD Setup used with ANSYS Fluent 17.1

ANSYS CFD Solver: ANSYS Fluent 17.1

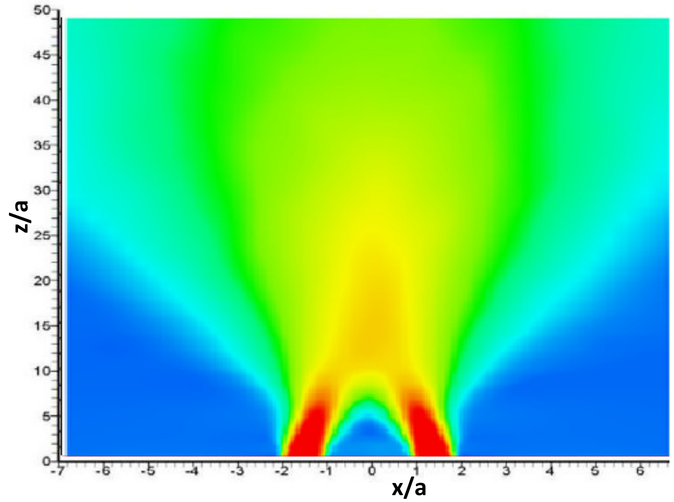
Mesh	Mesh 1	Mesh 2a	
Analysis Type	Transient	Transient	
Turbulence model	SST	SST	SBES
Solver Settings			
Gradient Scheme	LSCB	LSCB	LSCB
Pressure-Velocity Coupling	Coupled Solver	Coupled Solver	SIMPLEC
Pressure Scheme	Second Order	Second Order	Second Order
Advection Scheme	Second Order Upwind	Second Order Upwind	Bounded Central Difference
Turbulence Numerics	First Order Upwind	First Order Upwind	First Order Upwind
Physical Timestep	0.01s /0.01s	0.01s / 0.01s	2.5×10^{-4} s
Convergence control	Max. Iterations per Timestep: 5	Max. Iterations per Timestep: 5	Max. Iterations per Timestep: 4
Flow Development time steps	5000	5000	10000
Transient Averaging time steps	5000	5000	50000
Transient Scheme	Second Order Implicit	Second Order Implicit	Second Order Implicit
Residual Target	RMS Res$<10^{-7}$	RMS Res$<10^{-6}$	RMS Res$<10^{-5}$

Based on experience gained with the ANSYS CFX solver and the SBES turbulence model, this very expensive simulation has only performed on the adjusted fine grid level of Mesh2a.

Qualitative Comparison: Streamwise Velocity, SBES Turbulence Model

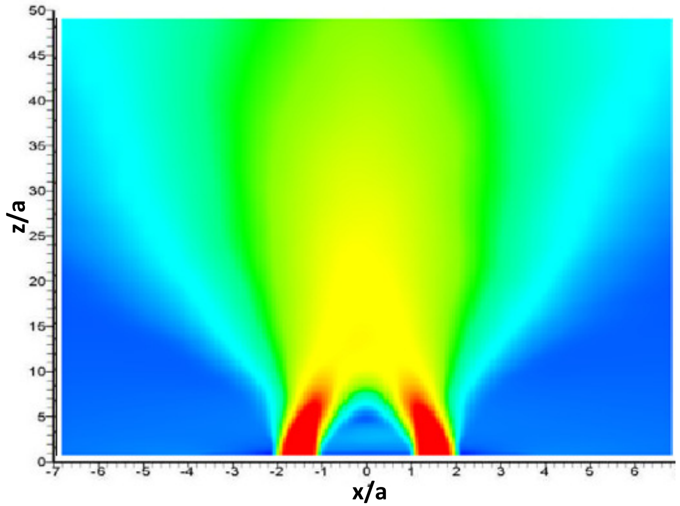


LDA Measurements

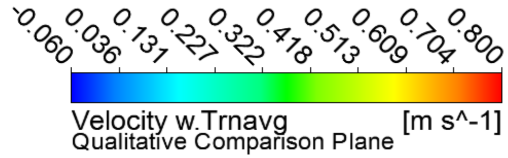


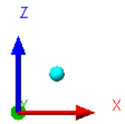
ANSYS CFX 17.1 SBES

W_{max}=0.8816 m/s

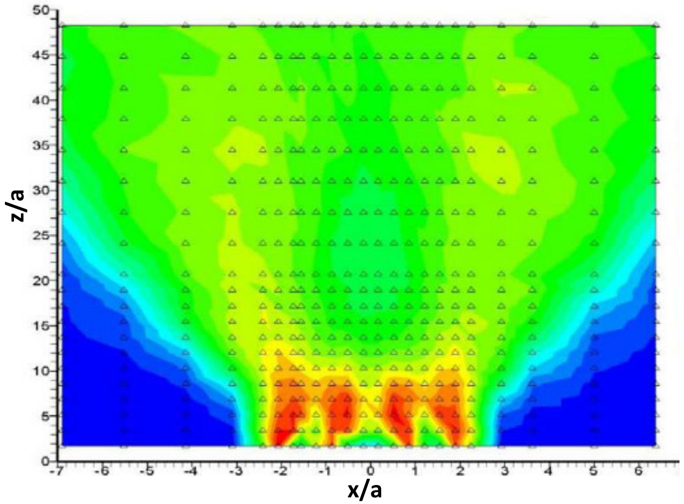


ANSYS Fluent 17.1 SBES

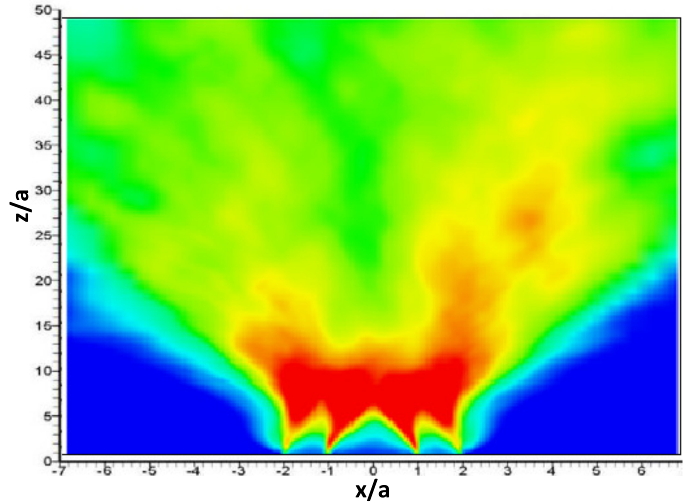




Qualitative Comparison: Streamwise turbulence intensity $\sqrt{\overline{w'^2}}/W_{max}$, SBES Turbulence Model

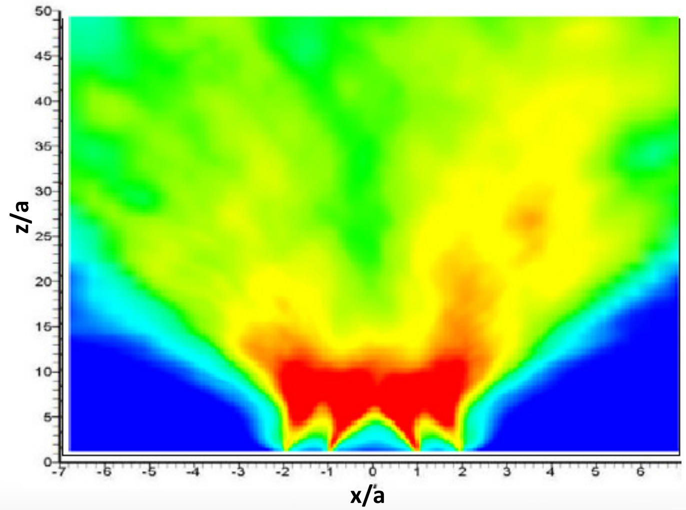


LDA Measurements

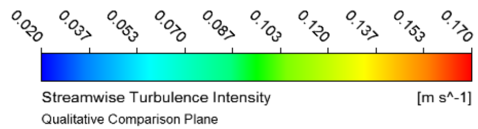


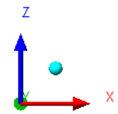
ANSYS CFX 17.1

- "Streamwise turbulence intensity" corresponds to the streamwise fluctuation z-velocity, non-dimensionalized by the maximum streamwise velocity at the nozzle cross section.
- W_{max}=0.8816 m/s

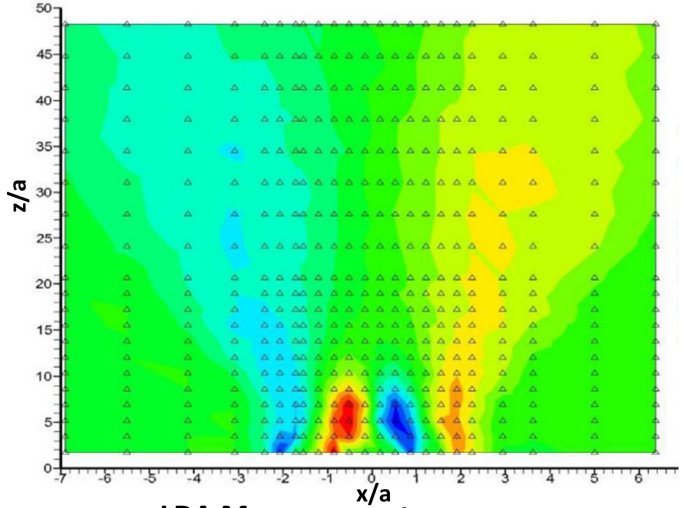


ANSYS Fluent 17.1

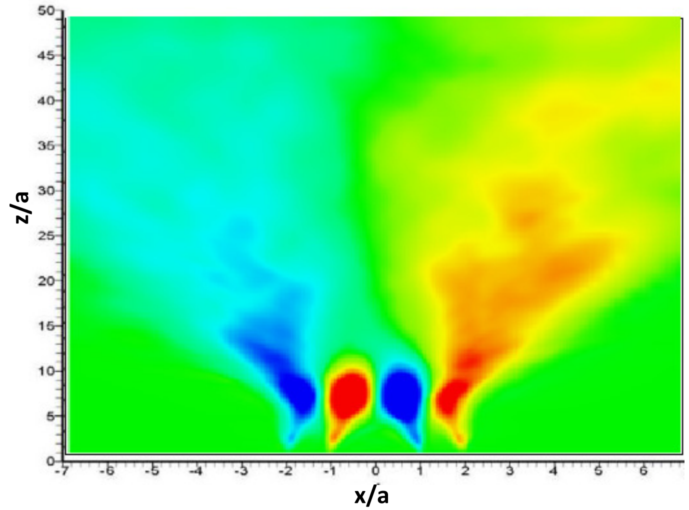




Qualitative Comparison: Normalized Reynolds Stress Tensor Component $\overline{u'w'}/W_{max}^2$ SBES Turbulence Model



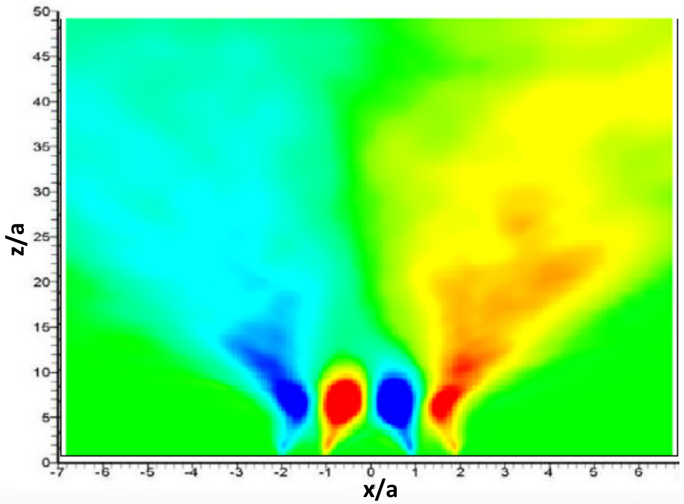
LDA Measurements



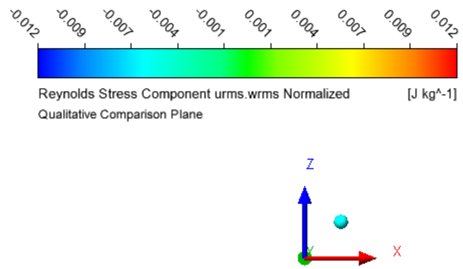
ANSYS CFX 17.1 SBES

- For CFD data, “Normalized Reynolds Stress Tensor Component” corresponds to the Reynolds Stress Tensor Component, non-dimensionalized by the square of maximum streamwise velocity at the nozzle cross section.
- For the LDA data it is named as “Reynolds Stress Component” and corresponds to the equally non-dimensionalized mixed Reynolds stress as well.

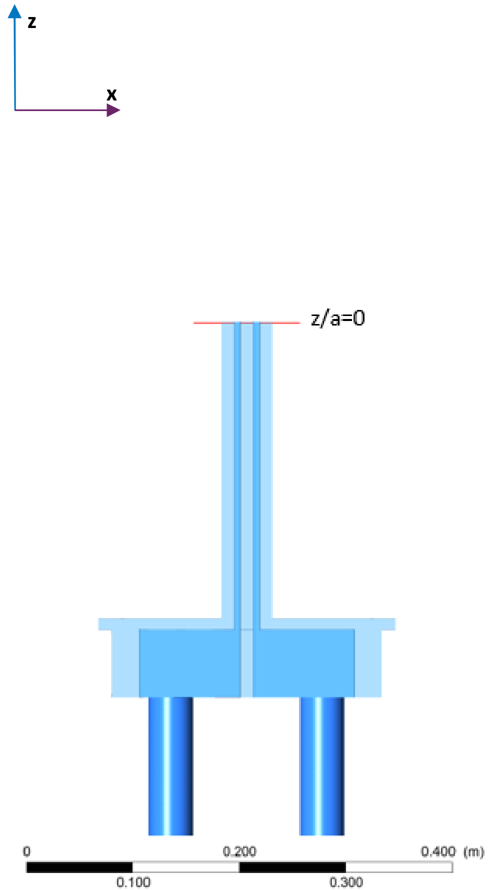
- W_{max}=0.8816 m/s



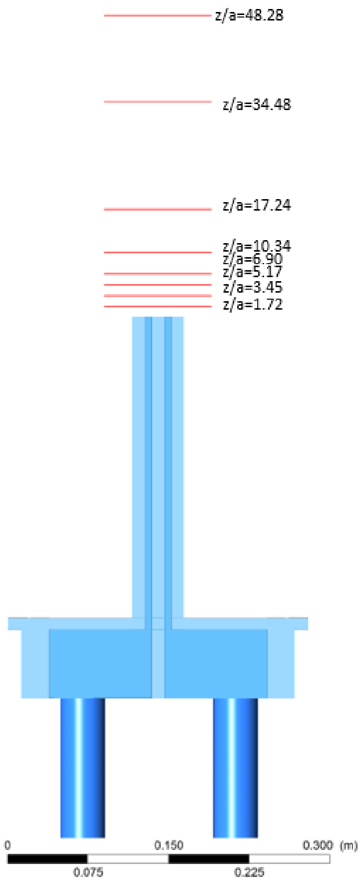
ANSYS Fluent 17.1 SBES



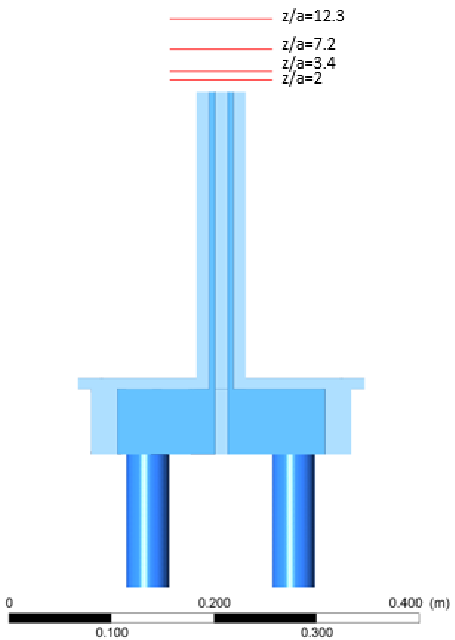
Measurement Locations (LDA & PIV Profile Measurements)



ASME Benchmark Specification

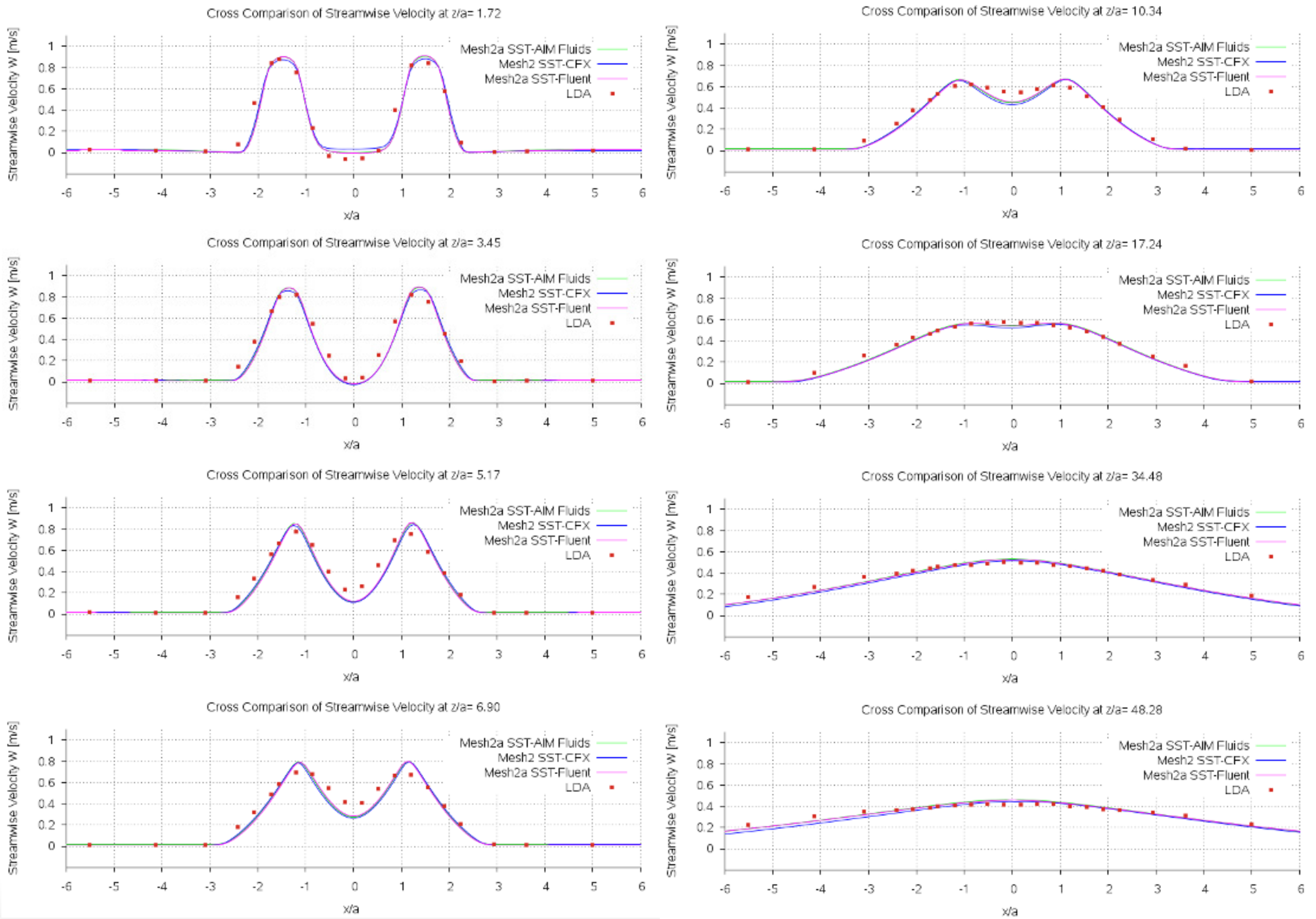


Laser Doppler Anemometry Measurements

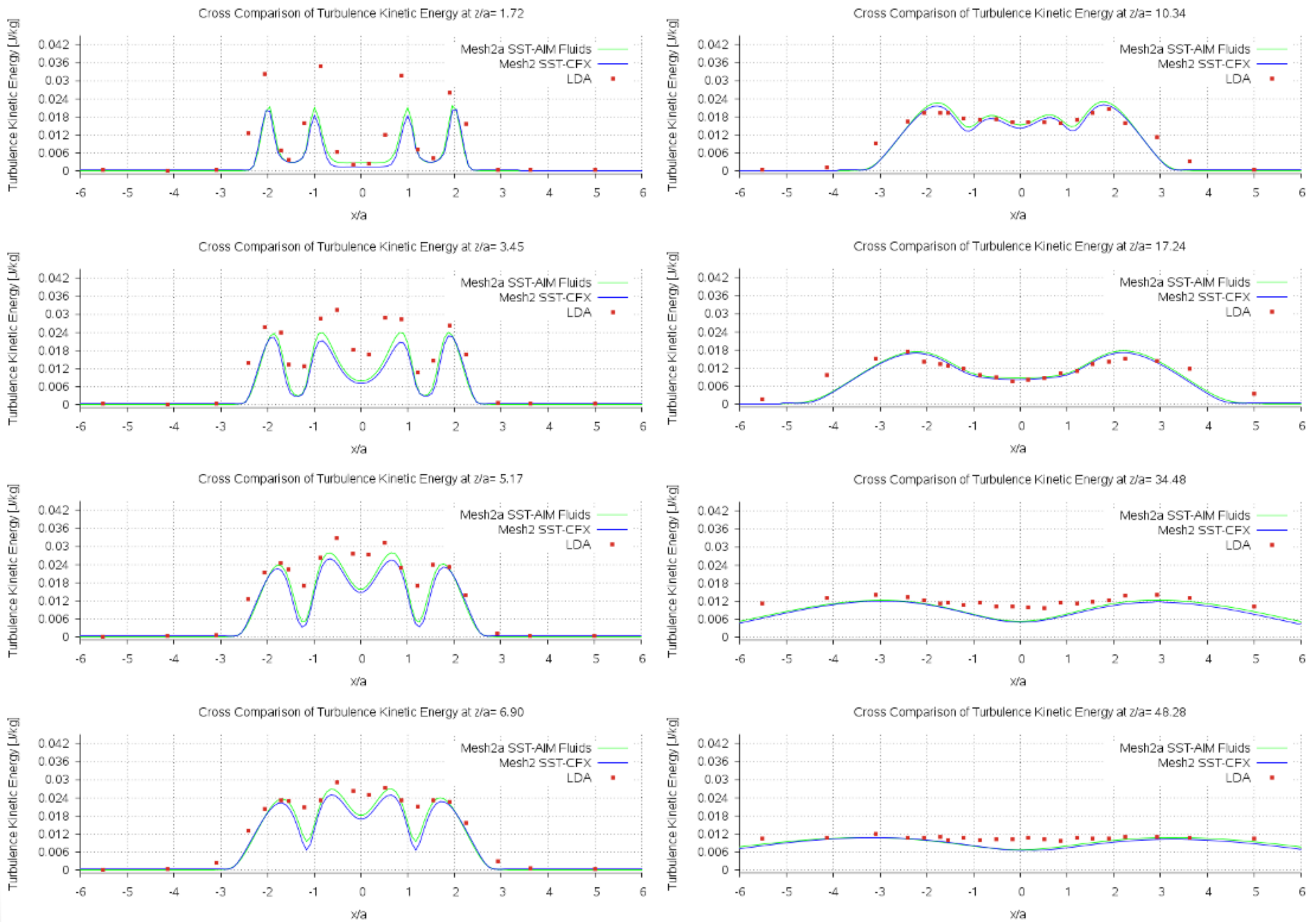


Particle Image Velocimetry Measurements

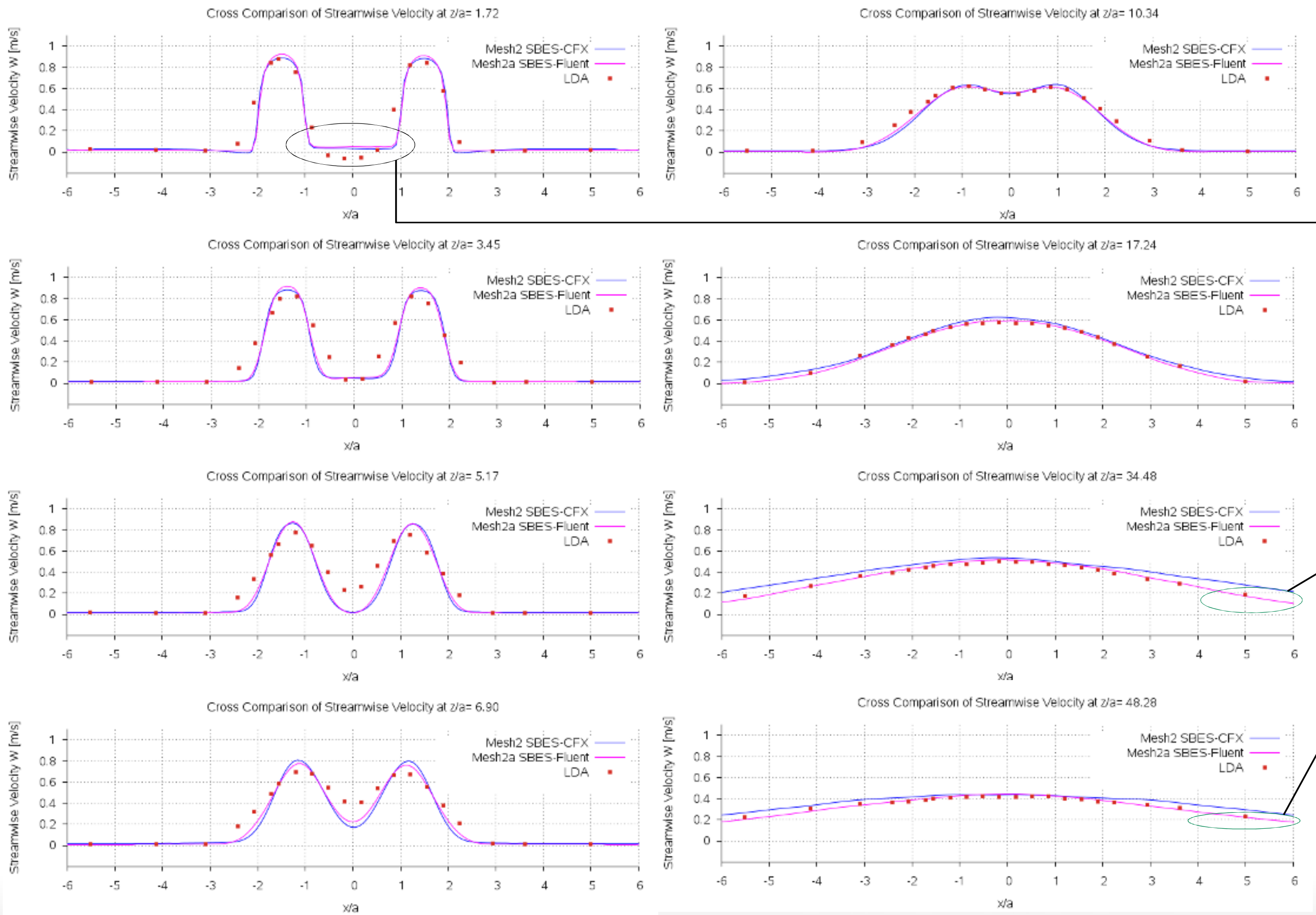
ANSYS CFD Solver Comparison: Streamwise Velocity (SST Model)



ANSYS CFD Solver Comparison: Turbulence Kinetic Energy (SST Model)



ANSYS CFD Solver Comparison: Streamwise Velocity for SBES Turbulence Model



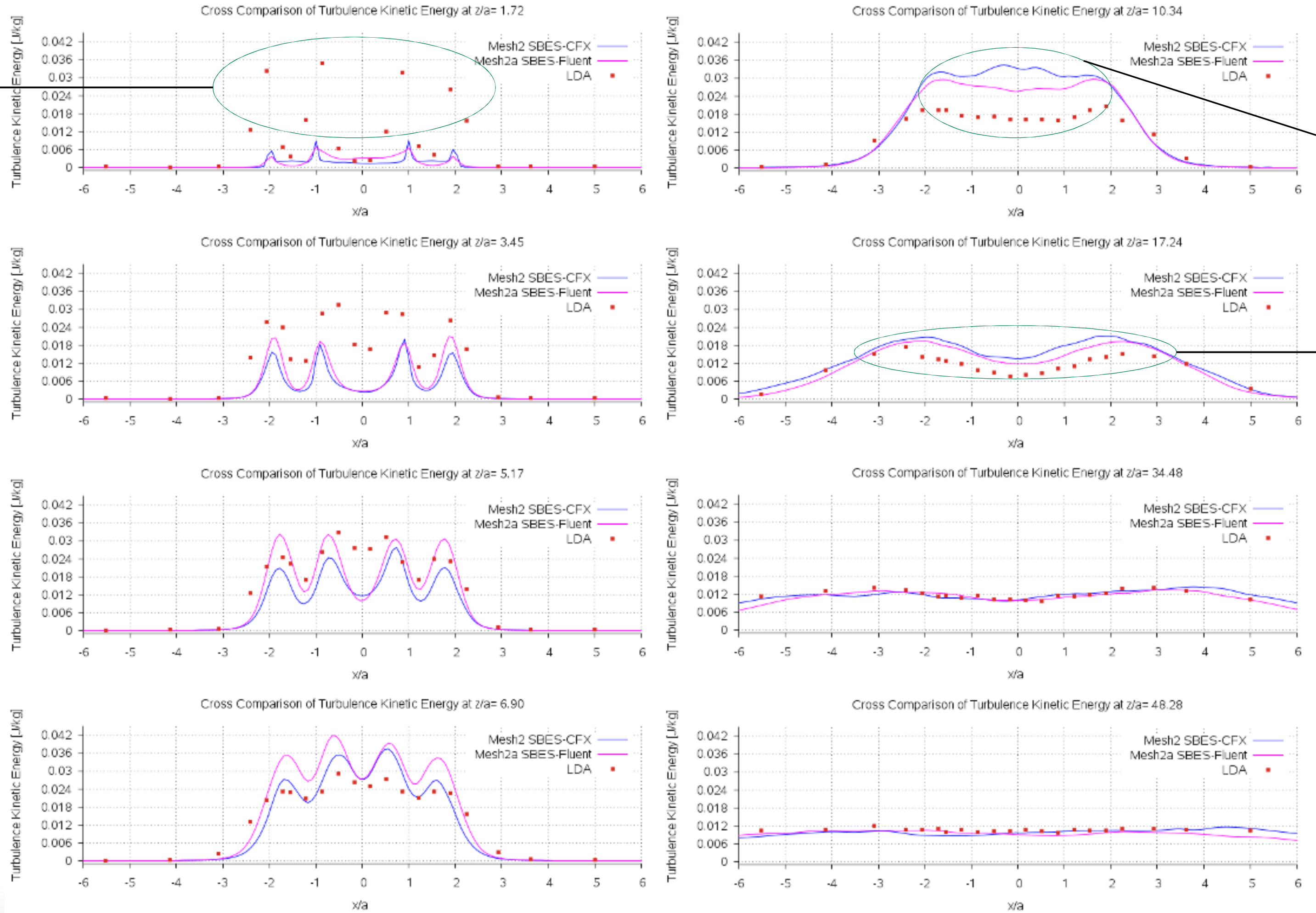
Under prediction of flow recirculation intensity

Effects of:

- Longer time averaging
- Mesh refinement

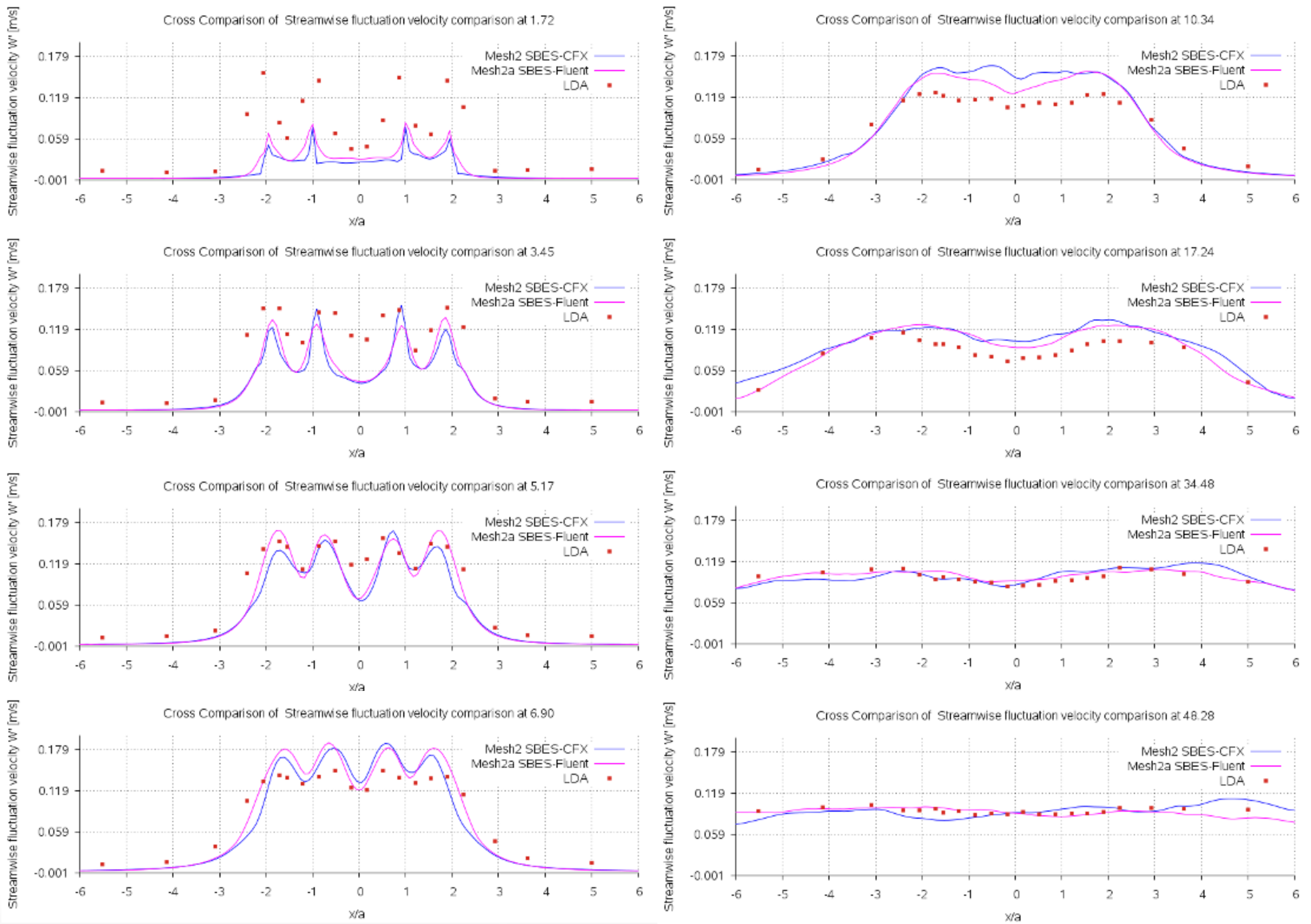
ANSYS CFD Solver Comparison: Resolved Turbulence Kinetic Energy (SBES)

- Turbulence on channel side walls not reflected in resolved turbulence kinetic energy
- Mesh still under-resolved in nozzle channels in order to get resolved turbulent scales at nozzle exits

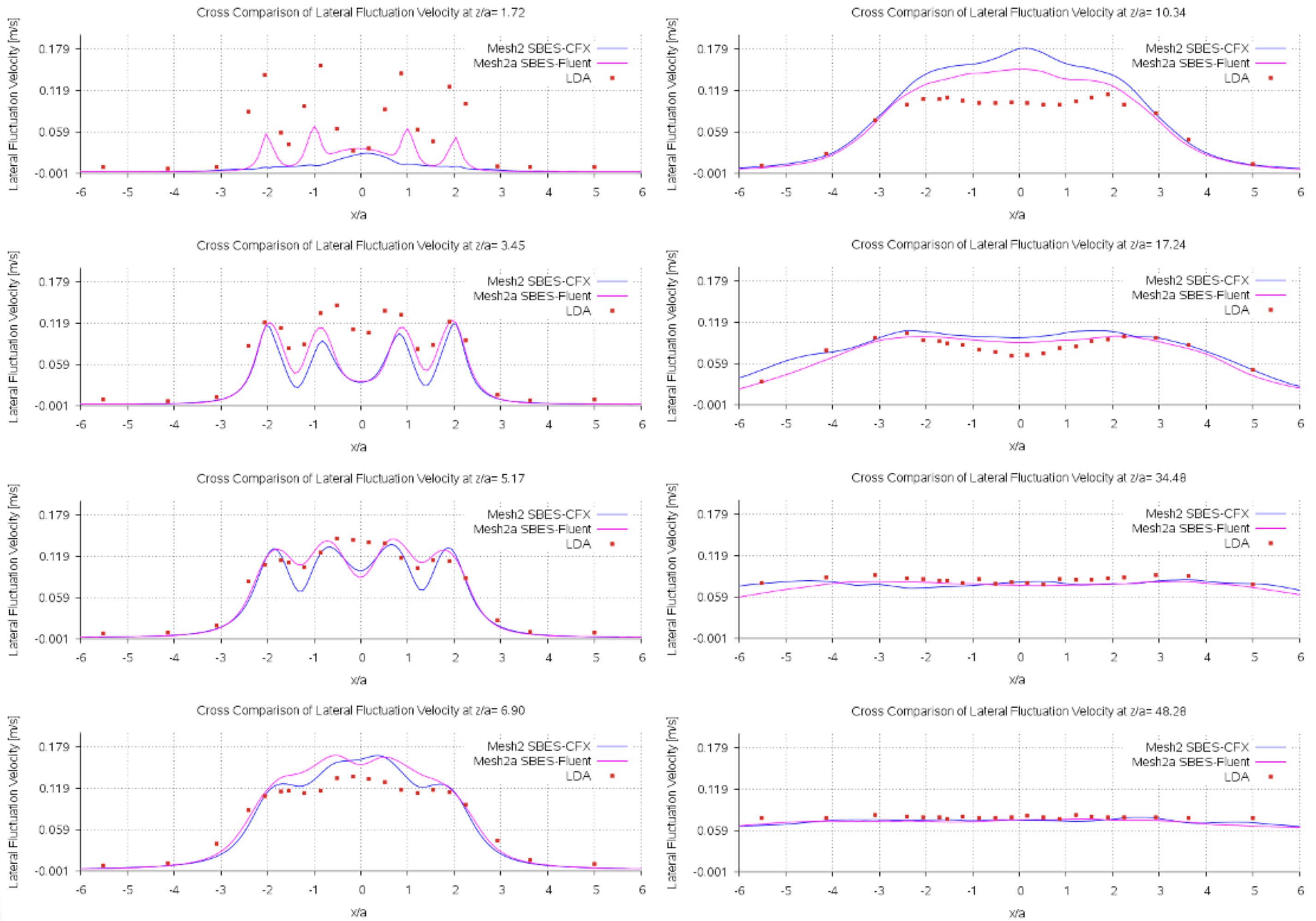


Effects of:
- Longer time averaging
- Mesh refinement

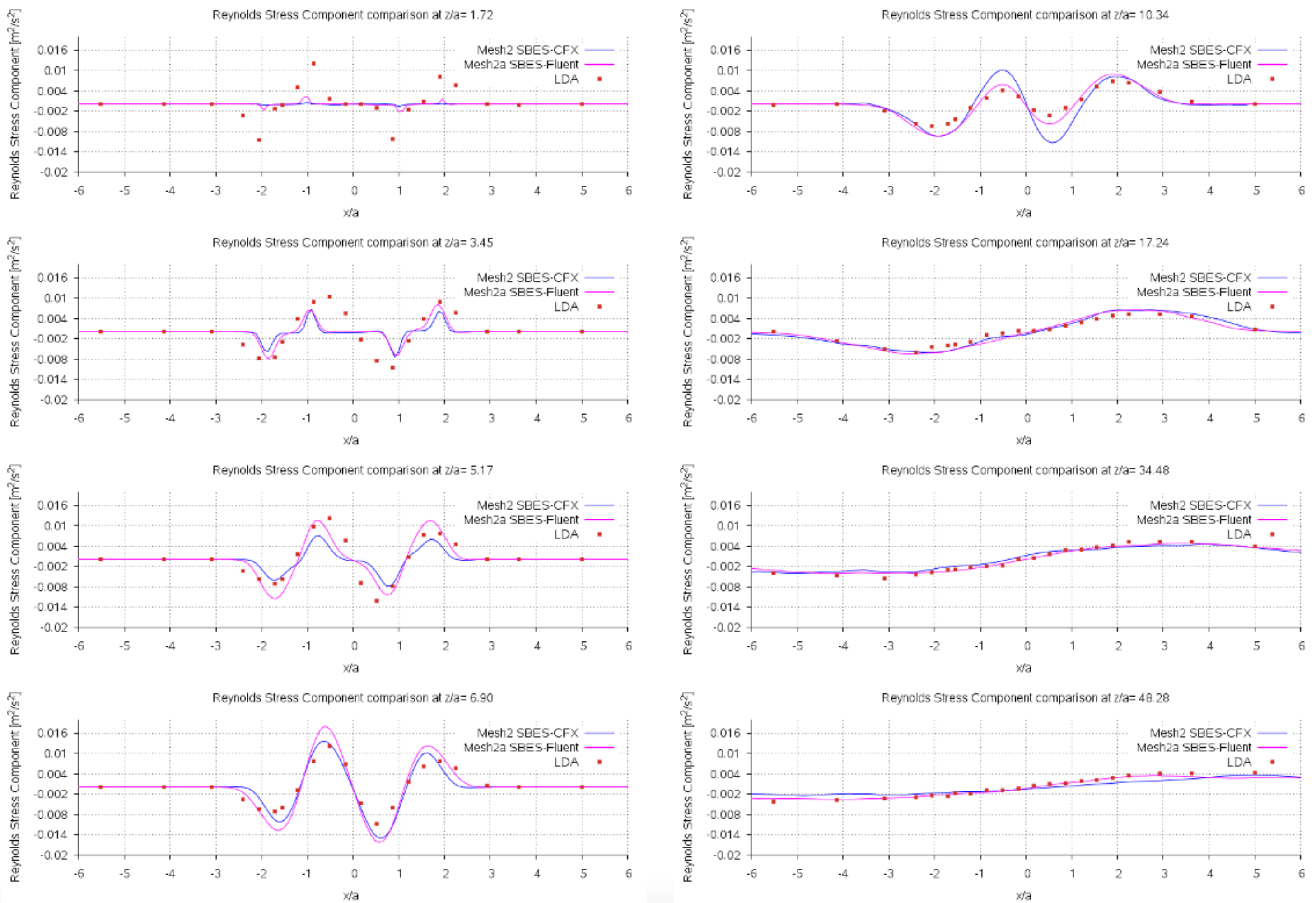
ANSYS CFD Solver Comparison: Streamwise Fluctuation Velocity (SBES)



ANSYS CFD Solver Comparison: Lateral Fluctuation Velocity (SBES)



ANSYS CFD Solver Comparison: Reynolds Stress Tensor Component (SBES)

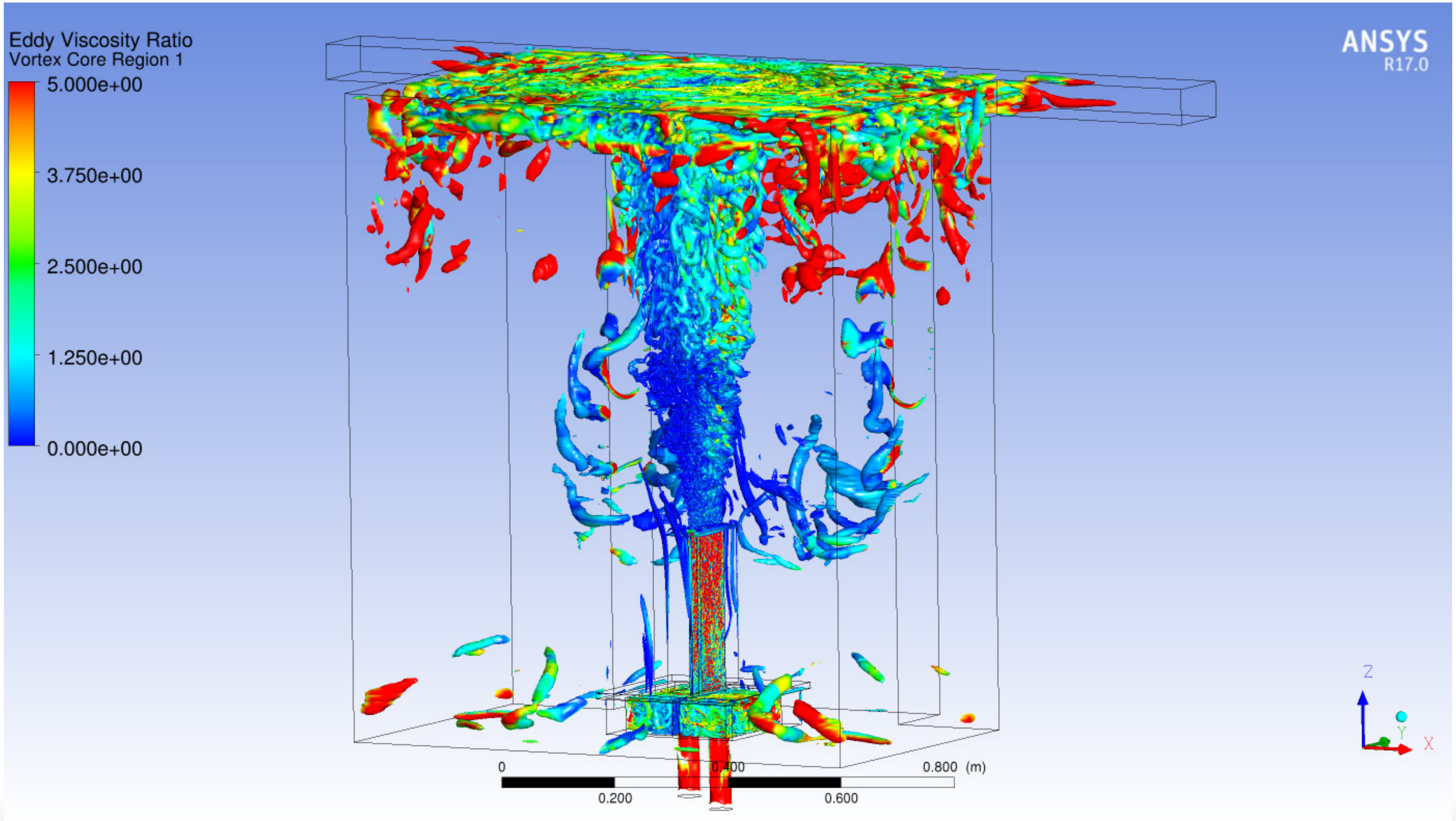


SBES Simulation: Q-Criterion Pseudo-Movie

- ANSYS Fluent 17.1, SBES Simulation
- Fine mesh 2a
- 1 Frame / 500 timesteps, Δt=0.25ms
- Q-criterion (Q=0.001)
- Colored by Eddy Viscosity Ratio
  EVR $\in$ [0, 5]

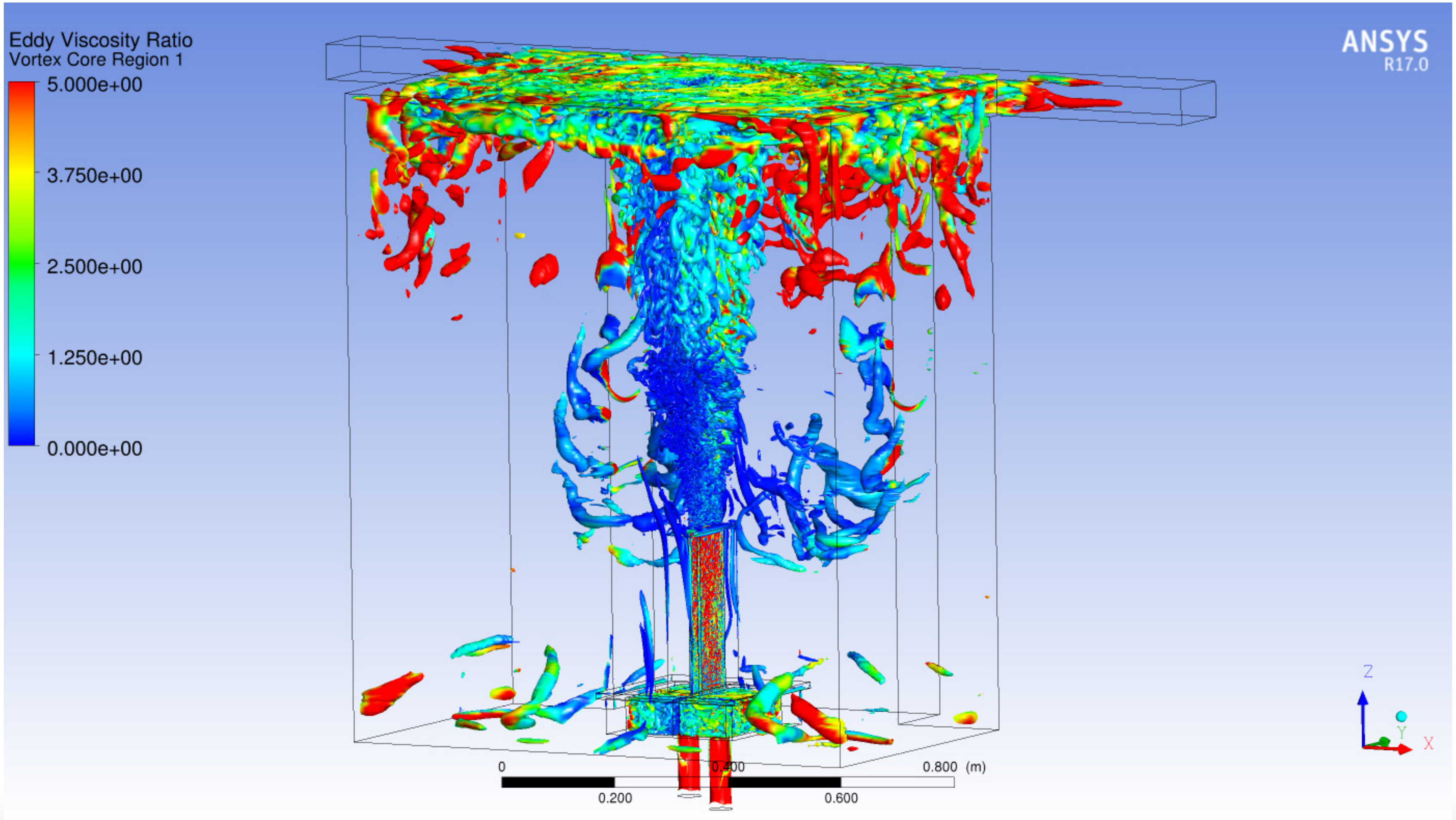
Time Development of Resolved Turbulent Scales from SBES

Timestep=53000



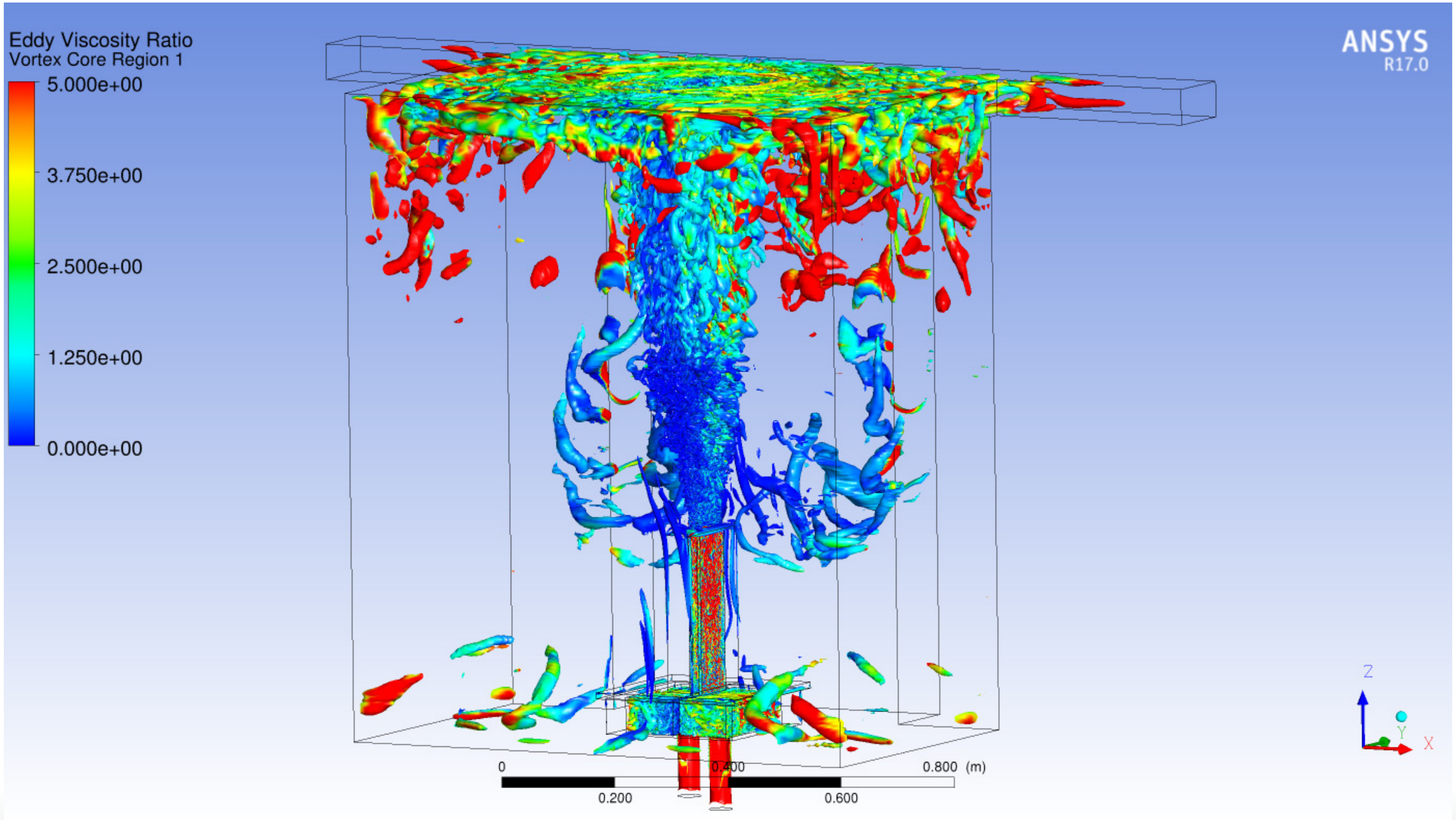
Time Development of Resolved Turbulent Scales from SBES

Timestep=53500



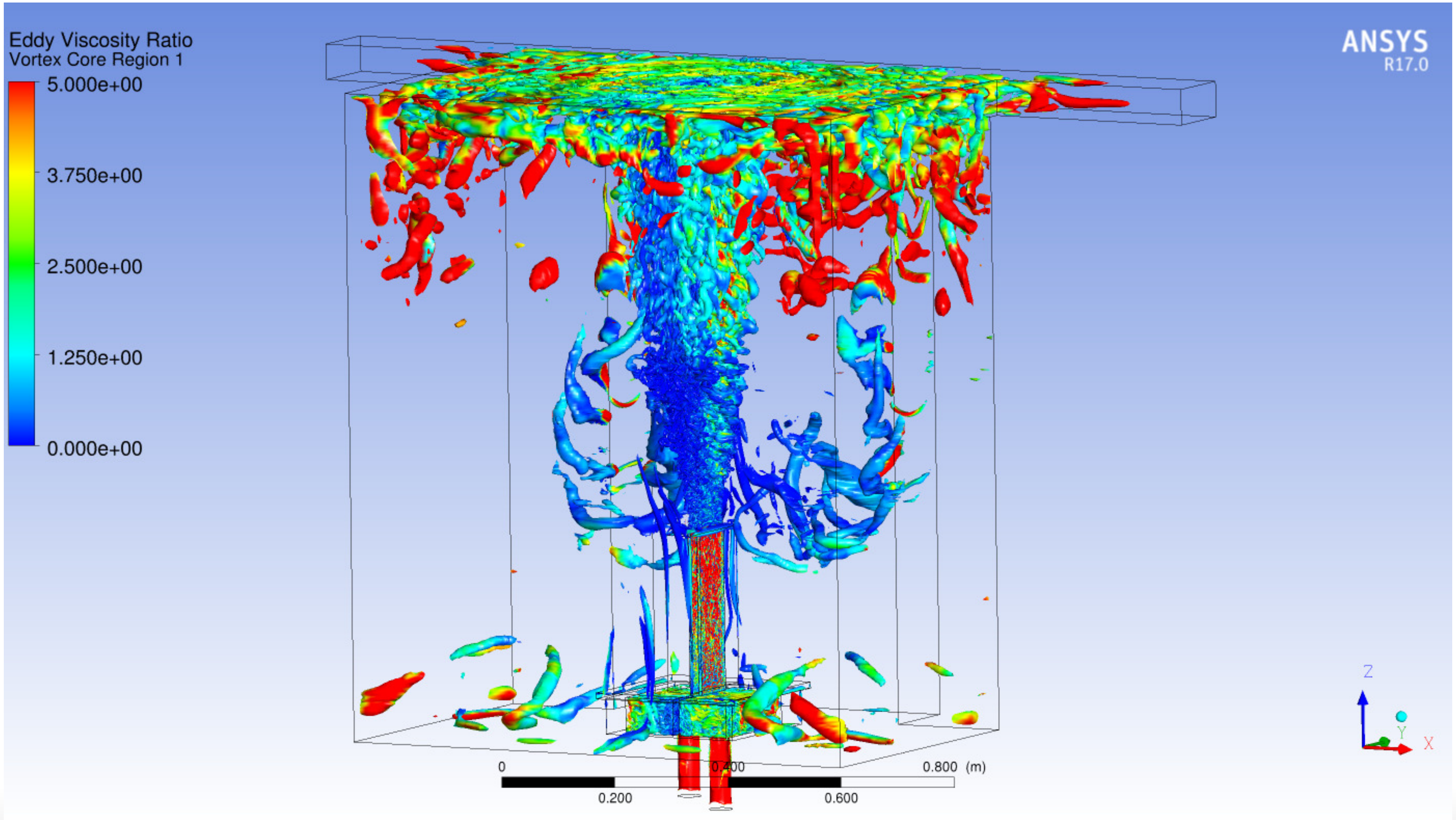
Time Development of Resolved Turbulent Scales from SBES

Timestep=54000



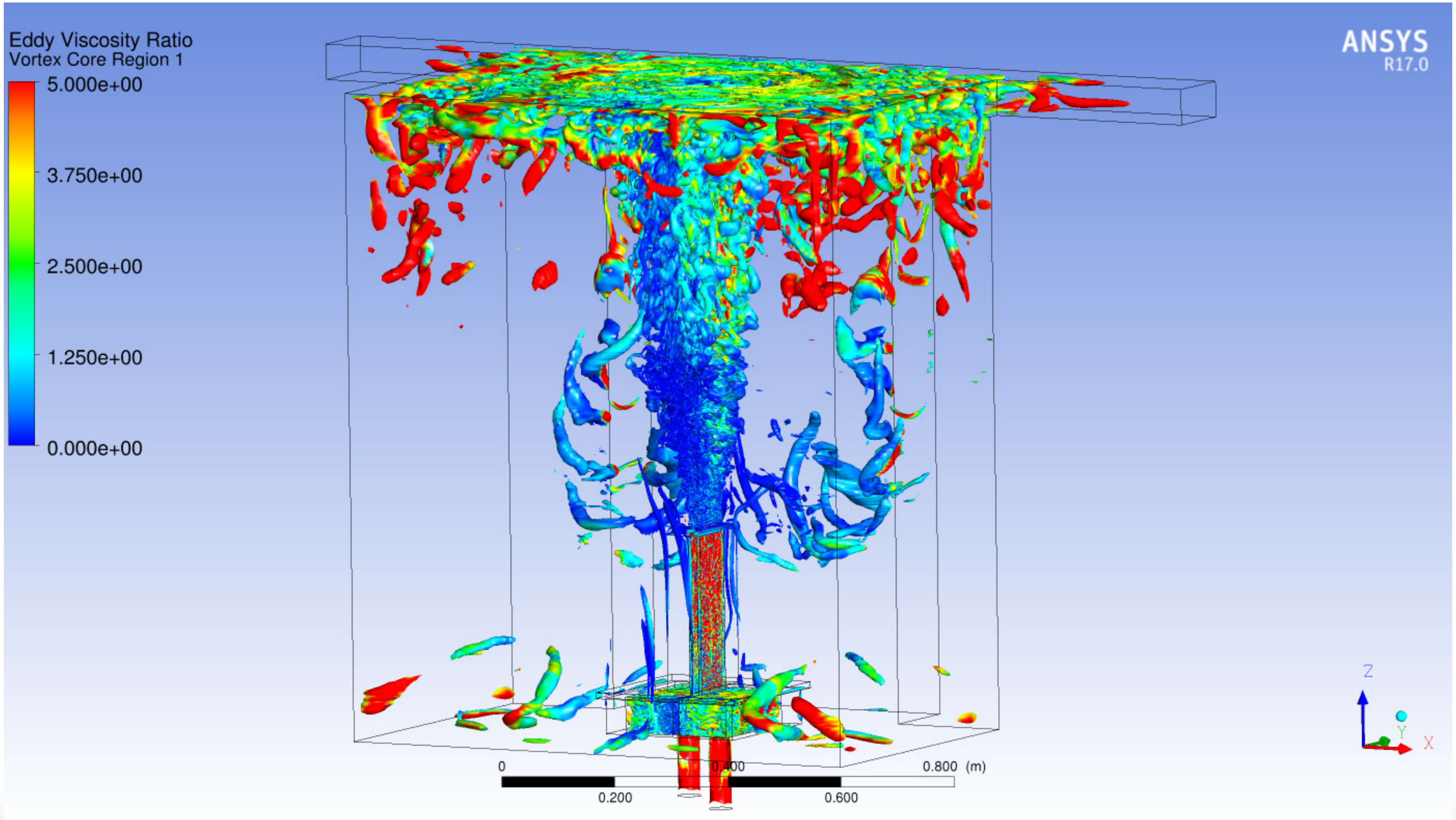
Time Development of Resolved Turbulent Scales from SBES

Timestep=54500



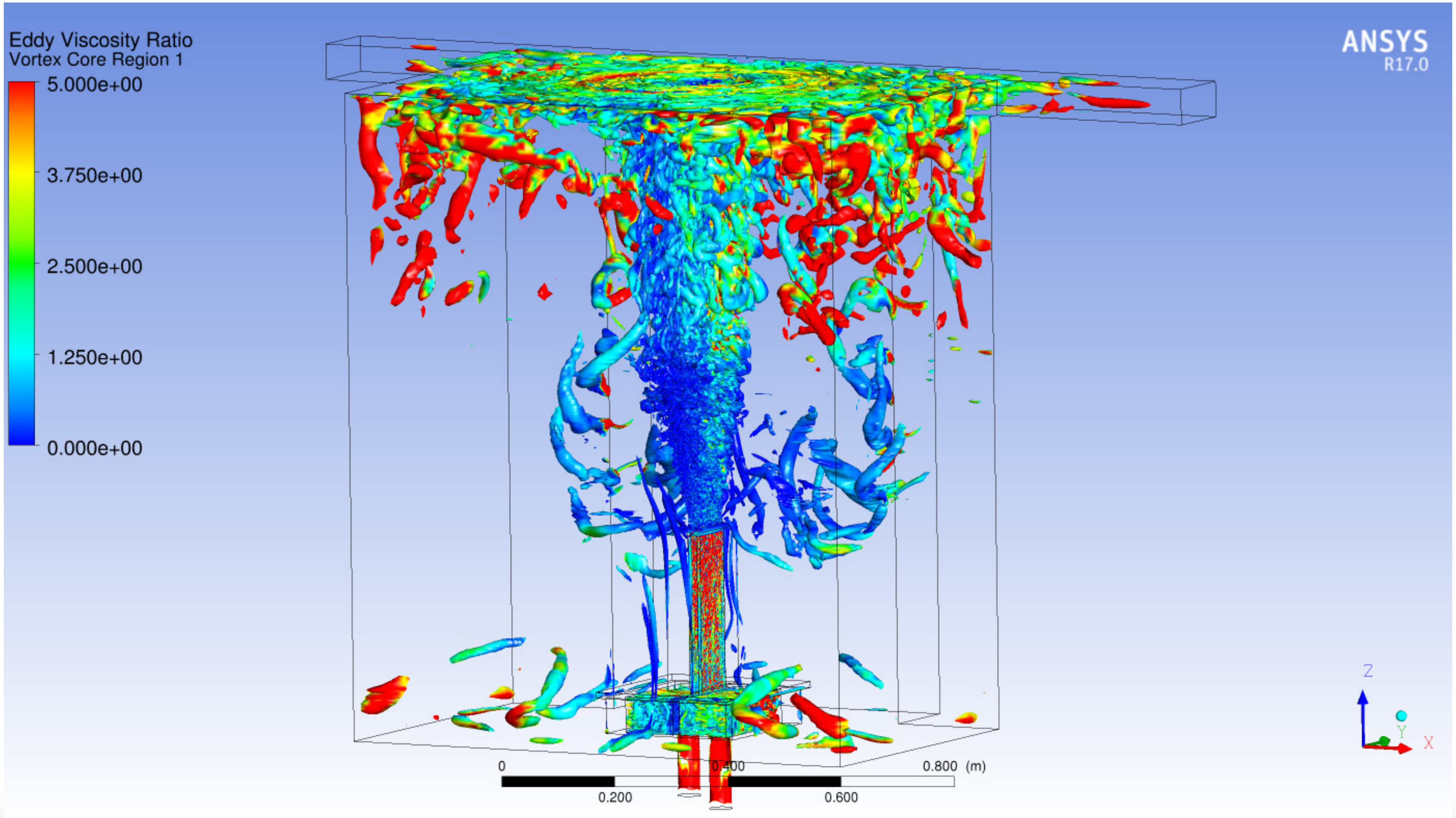
Time Development of Resolved Turbulent Scales from SBES

Timestep=55000



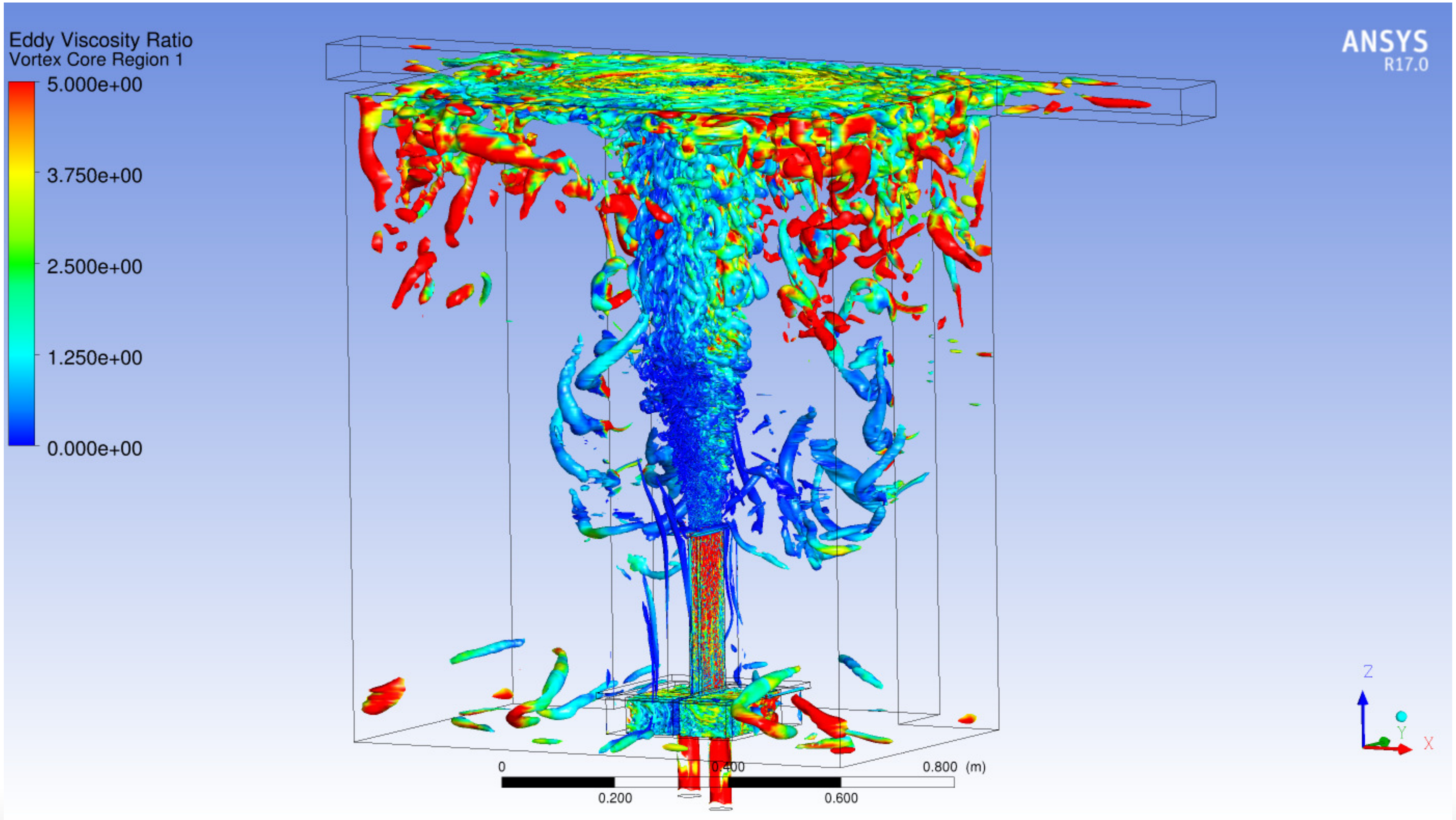
Time Development of Resolved Turbulent Scales from SBES

Timestep=55500



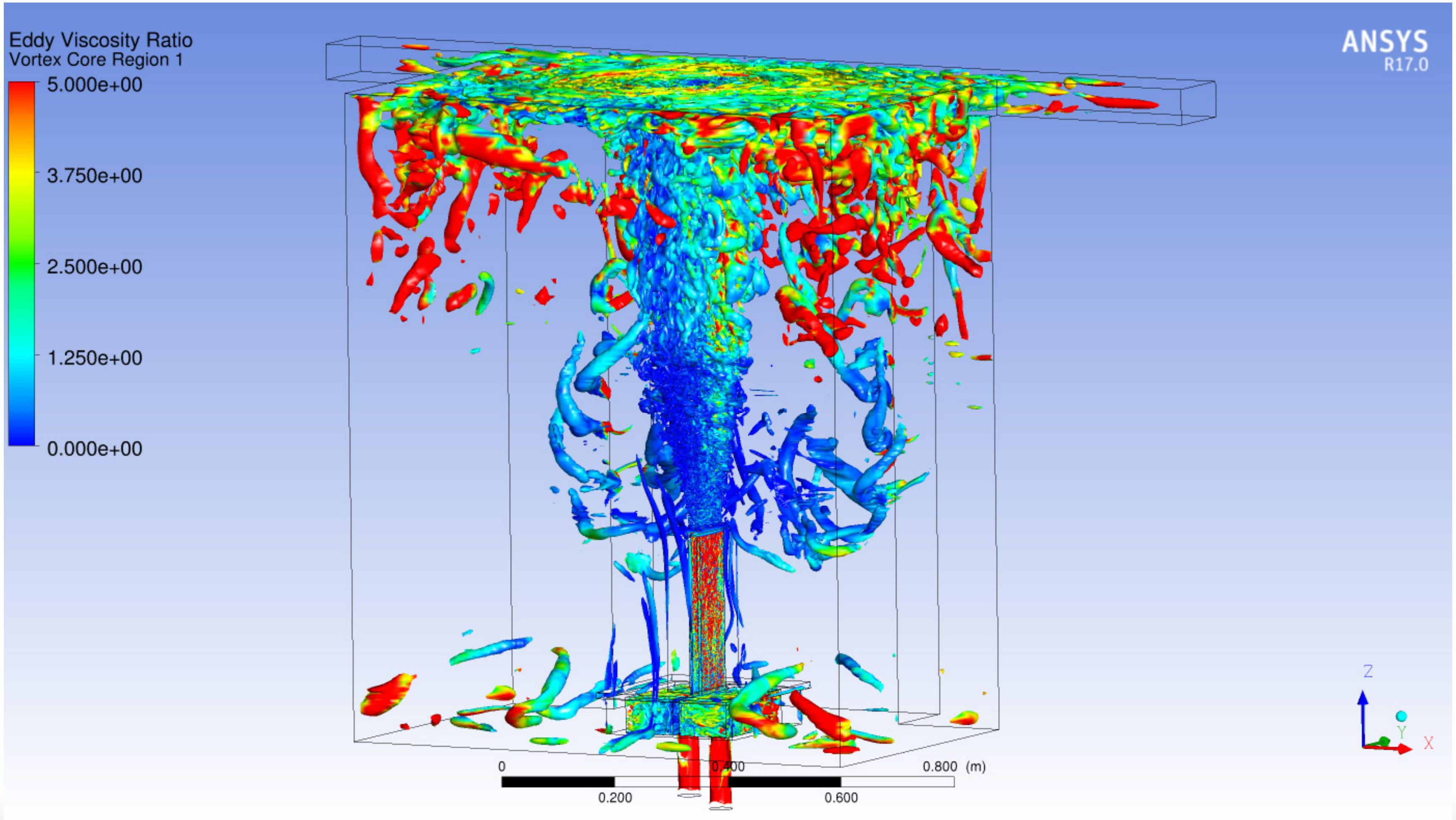
Time Development of Resolved Turbulent Scales from SBES

Timestep=58500



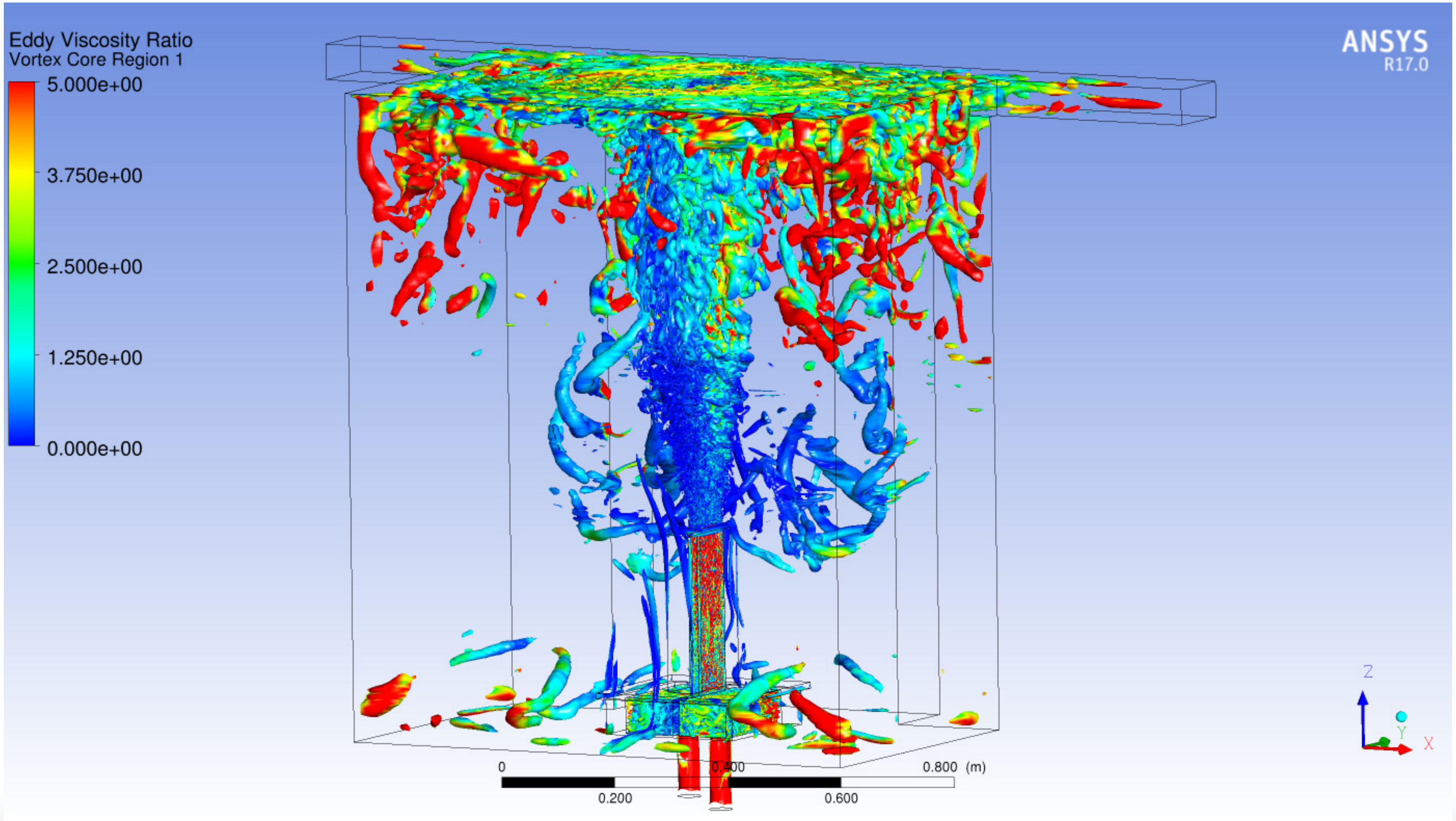
Time Development of Resolved Turbulent Scales from SBES

Timestep=59000



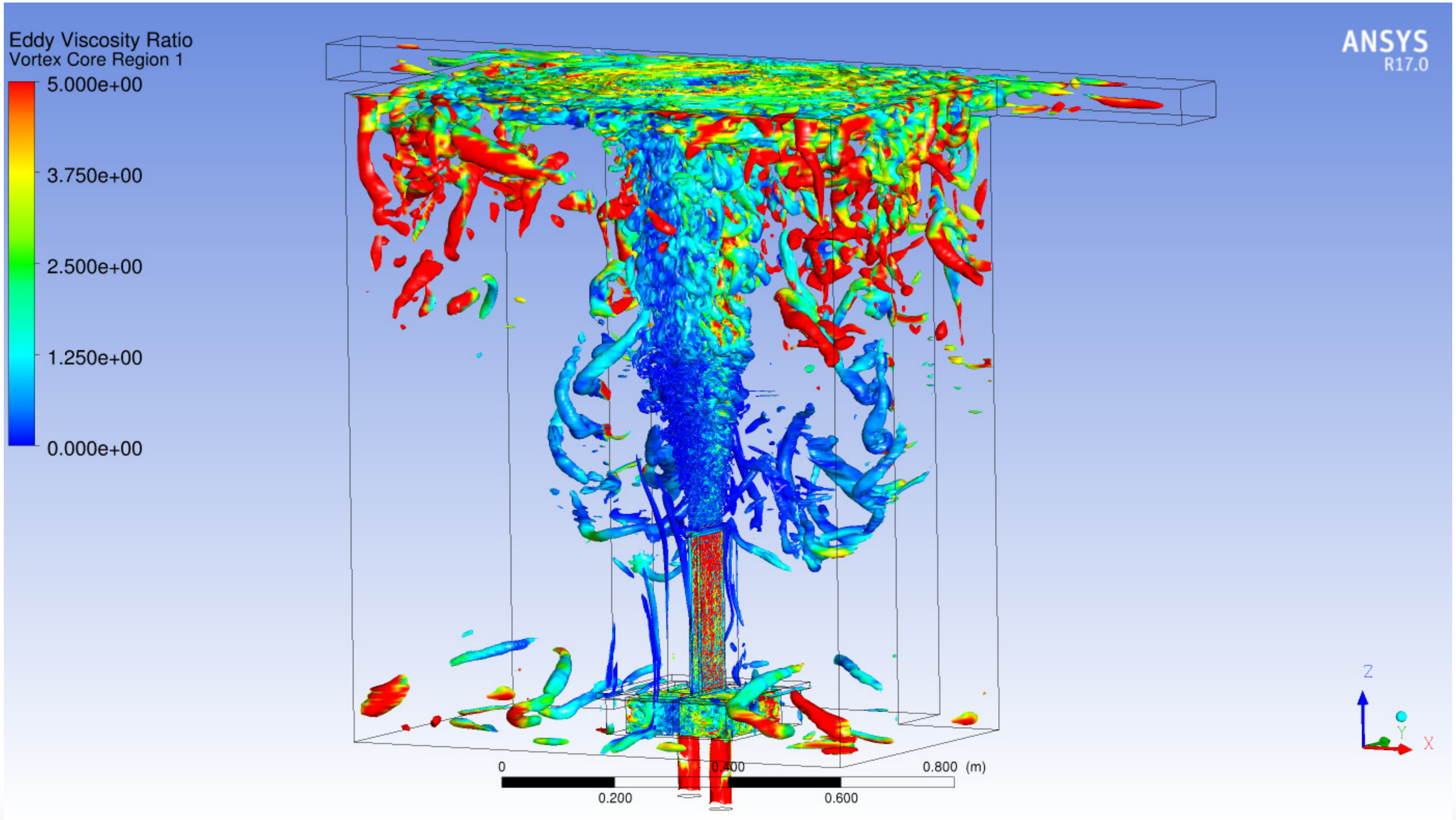
Time Development of Resolved Turbulent Scales from SBES

Timestep=60000



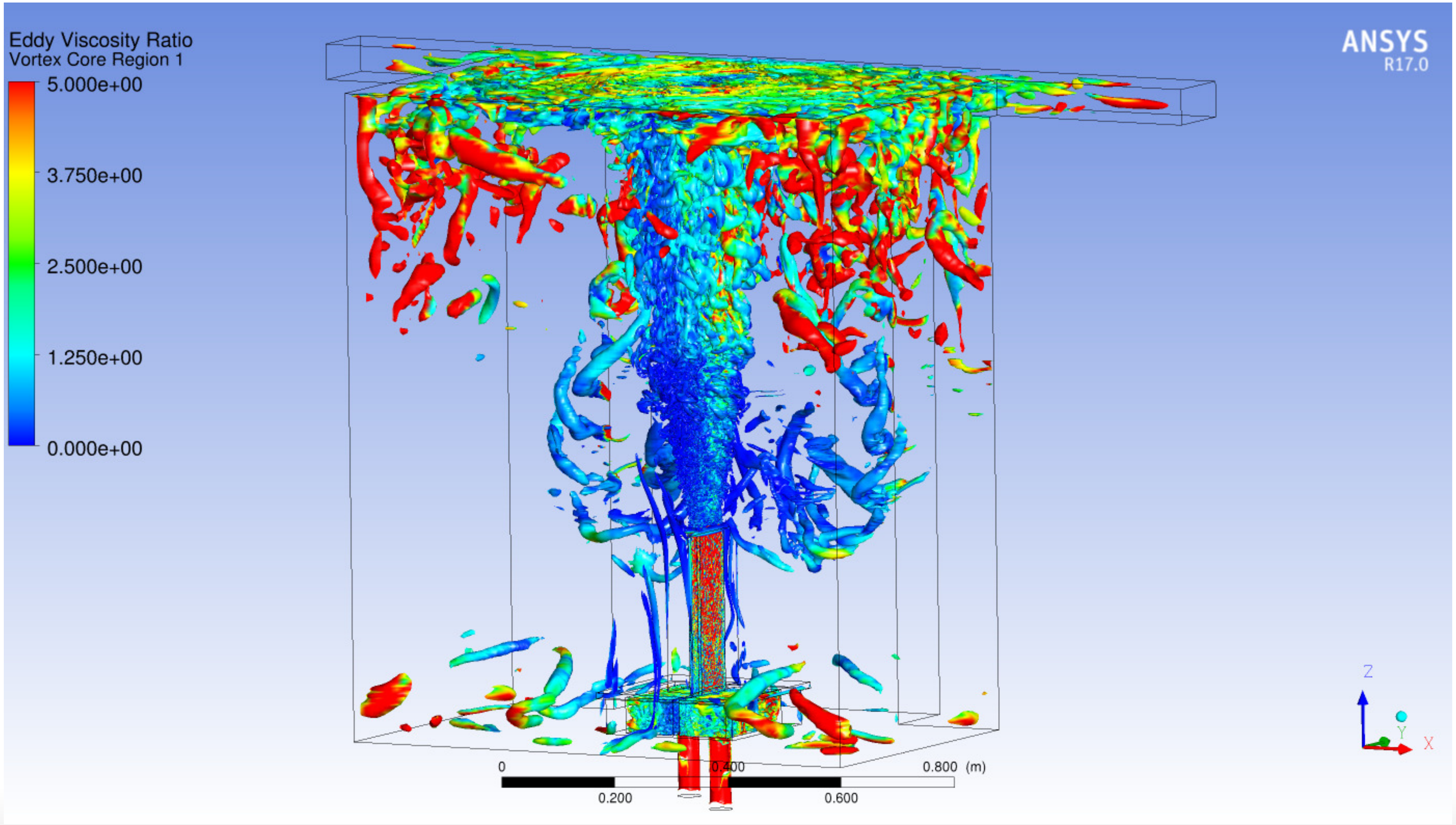
Time Development of Resolved Turbulent Scales from SBES

Timestep=60500



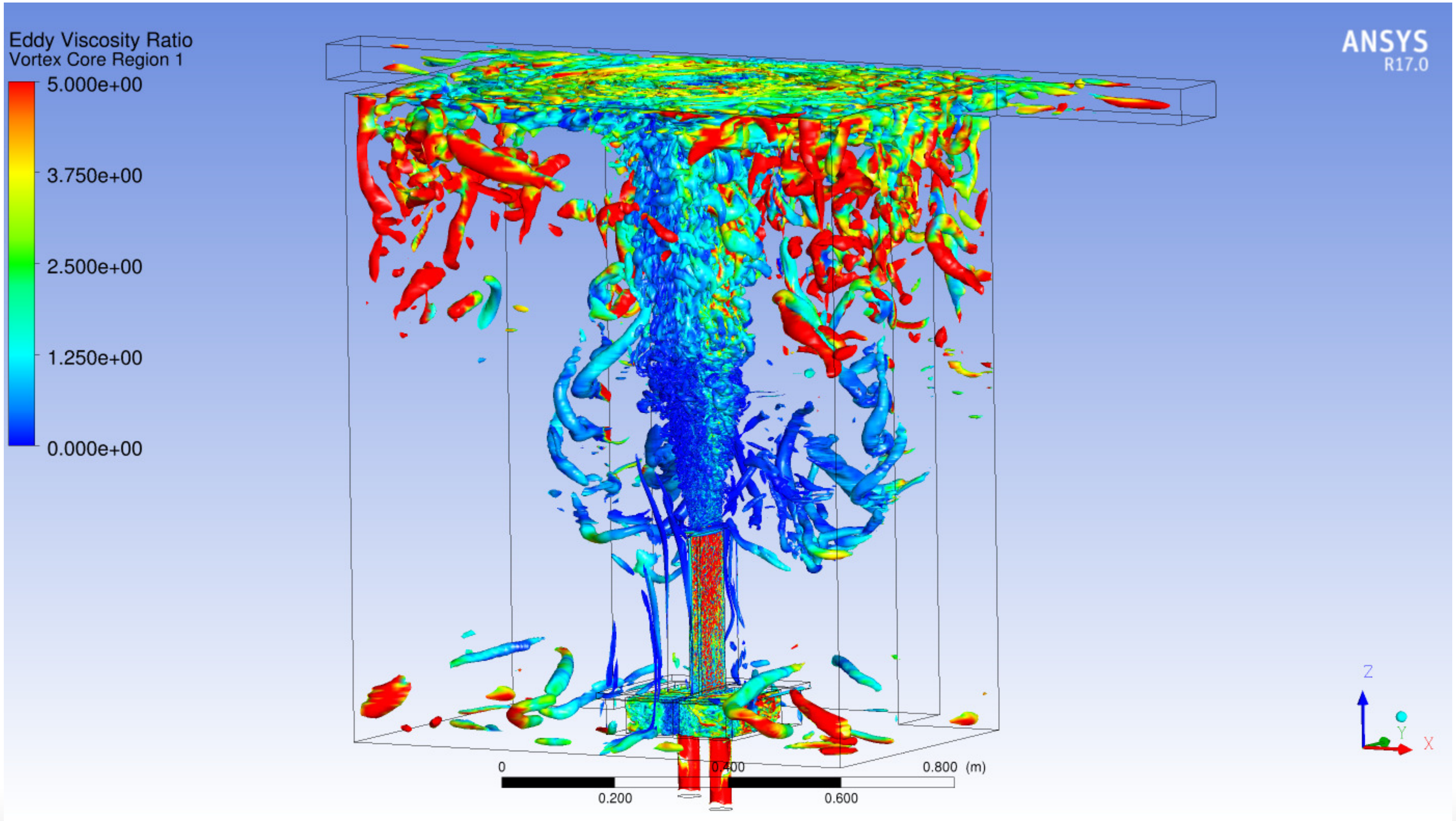
Time Development of Resolved Turbulent Scales from SBES

Timestep=61000



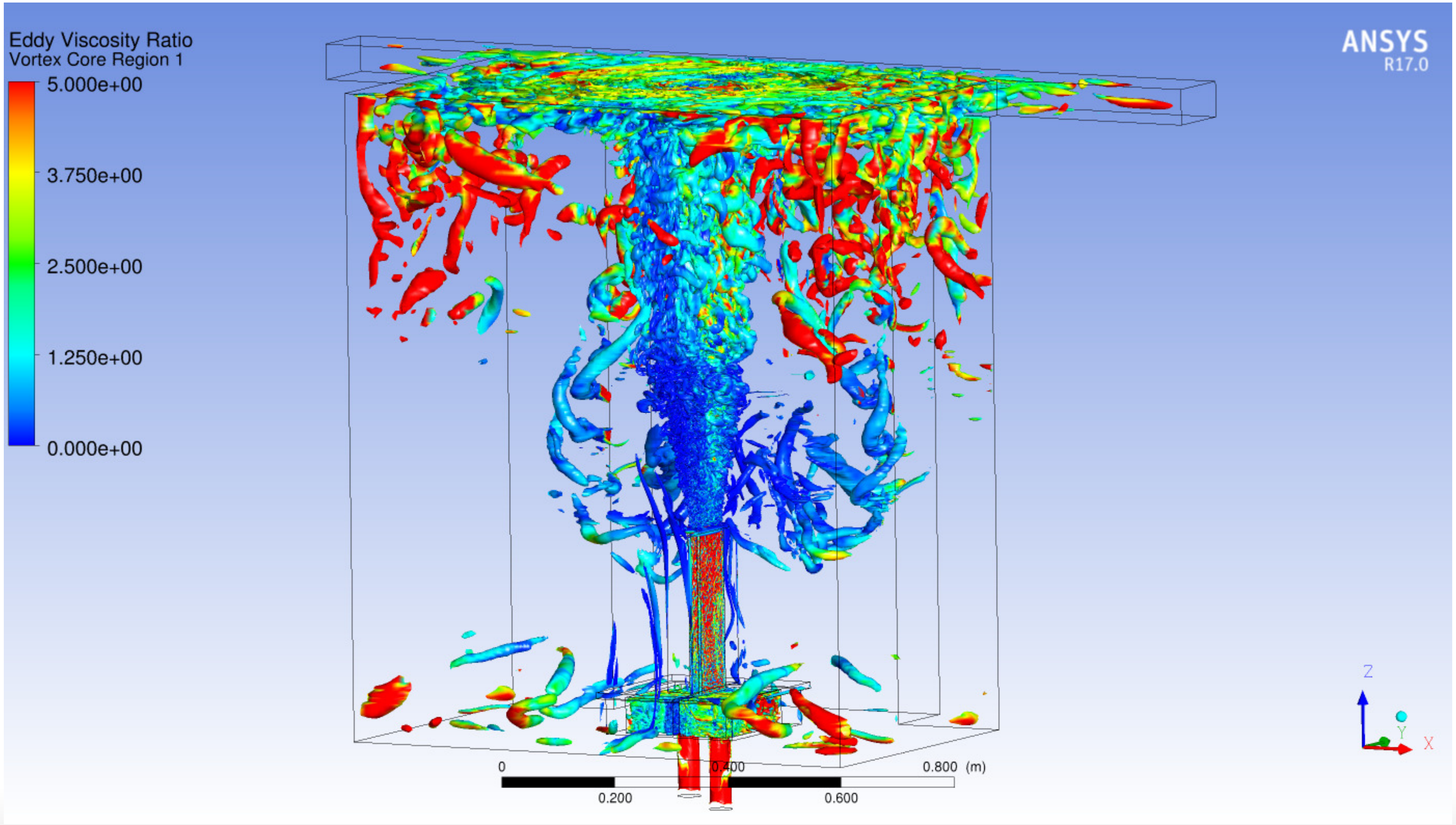
Time Development of Resolved Turbulent Scales from SBES

Timestep=61500



Time Development of Resolved Turbulent Scales from SBES

Timestep=62000



Summary & Conclusion

- The ASME V&V 30 Benchmark Problem 1 has been investigated using three different ANSYS CFD solvers, applying URANS SST model and the newly developed SBES (**S**tress **B**lended **E**ddy **S**imulation) scale-resolving turbulence model. Almost grid independent results have been obtained for both turbulence models.
- In the investigation of the ASME V&V 30 Benchmark Problem 1 all applicable CFD best practices have been applied in order to reduce the level of uncertainty, numerical and systematic errors for the obtained CFD result.
- The CFD results compare very well with the provided LDA & PIV measurements by Prof. Y.A. Hassan & team from the Texas A&M University and University of Tennessee, USA.
- Similar level of CFD solution accuracy has been obtained with both URANS SST and SBES model simulations. Minor differences can be observed depending on individual measurement cross-section.
- Even further increased accuracy could be obtained by using SBES and by resolving turbulent scales inside the nozzle channels.