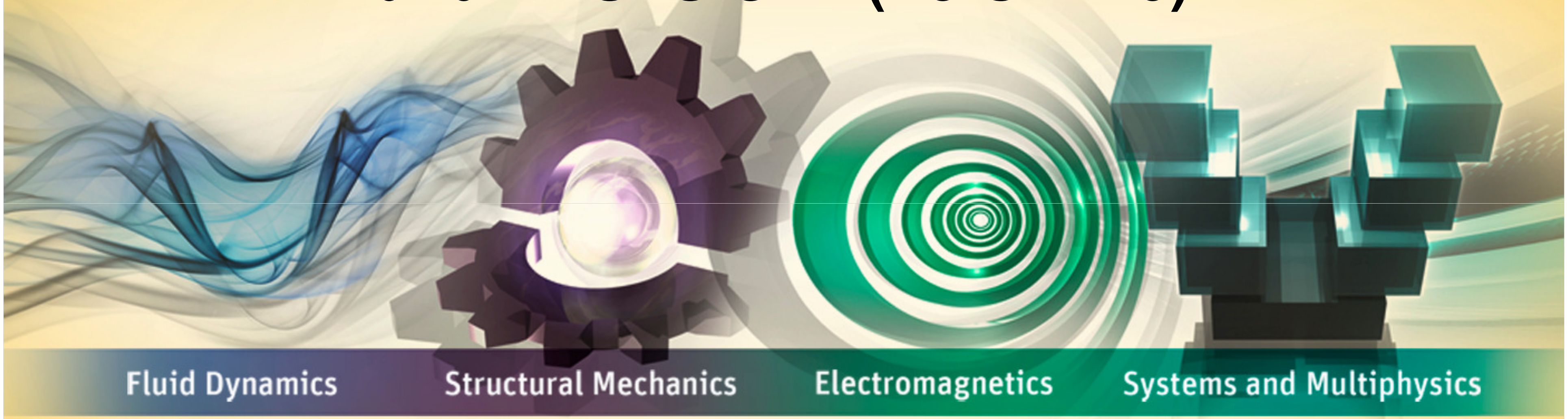
Realize Your Product Promise™

The OECD/NEA MATiS-H Benchmark – CFD Analysis of Water Flow through a 5x5 Rod Bundle with Spacer Grids using ANSYS Fluent and ANSYS CFX (Vers. 14.0)



Th. Frank (ANSYS Germany, Otterfing),
S. Jain (ANSYS Inc., Lebanon),
A.A. Matyushenko, A.V. Garbaruk
(NTS, St. Petersburg, Russia)
Thomas.Frank@ansys.com

Contents

- The OECD/NEA MATiS-H benchmark
- Outline of CFD investigation methodology
  1. Verification of inlet BC's
  2. CFD Best Practice – Investigation of reduced geometry
  3. The MATiS-H 5×5 rod bundle with split type spacer grid
  4. The MATiS-H 5×5 rod bundle with swirl type spacer grid
- CFD results and comparison to KAERI data
- Summary & Conclusions

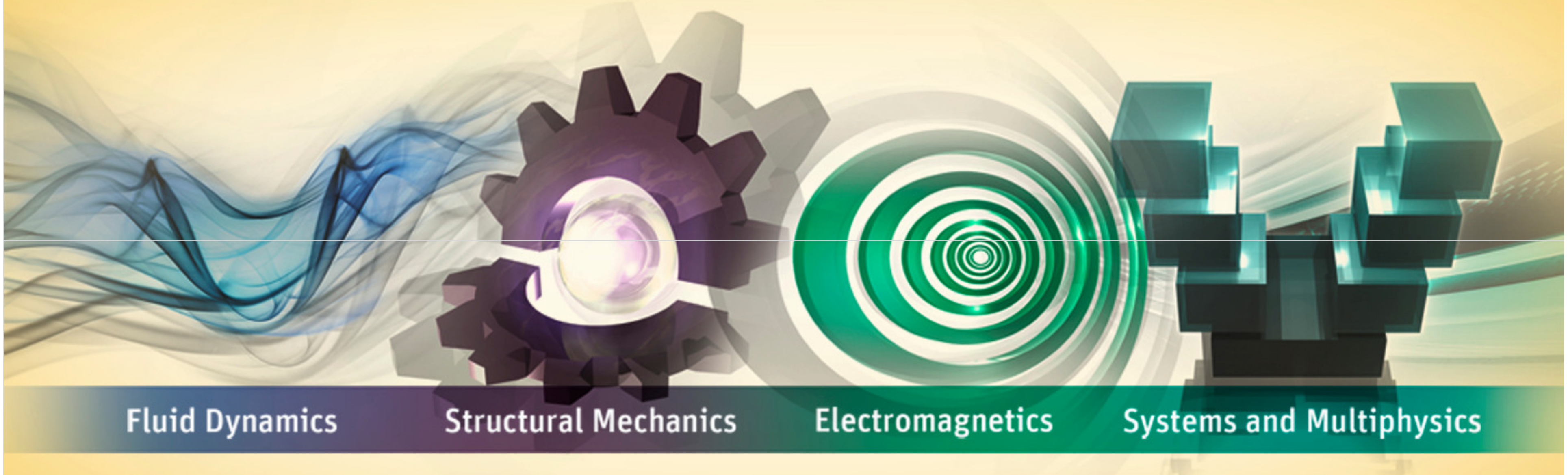
ANSYS Objectives in OECD/NEA MATiS-H Benchmark

- OECD/NEA MATiS-H Benchmark
  - Initiated by KAERI and OECD/NEA WGAMA
    WG2: Evaluate existing CFD assessment basis, identify V&V and modelling gaps
- ANSYS participation in the OECD/NEA benchmark exercise
  - Investigate both types of split & swirl spacer grids
  - Apply CFD Best Practice Guidelines as far as possible
  - Blind test simulations using ANSYS Fluent and ANSYS CFX
  - Post-test investigations using scale-resolving turbulence models (SRS)
- Demonstrate the meshing and CFD simulation capabilities of the ANSYS software for this complex and challenging Nucl. React. Eng. application
- Validate URANS and scale-resolving (SRS) turbulence models
- Compare the ANSYS Fluent and ANSYS CFX solutions
- Compare ANSYS CFD results with KAERI data

ANSYS

Realize Your Product Promise™

Benchmark Specification and MATiS Test Facility



Benchmark Specification & References

The OECD/NEA MATiS-H benchmark has been specified in:

– OECD/NEA : "*MATiS-H Benchmark - Final Benchmark Specifications", pp. 44. April* 2011 (last changed edition: 25. April 2012).

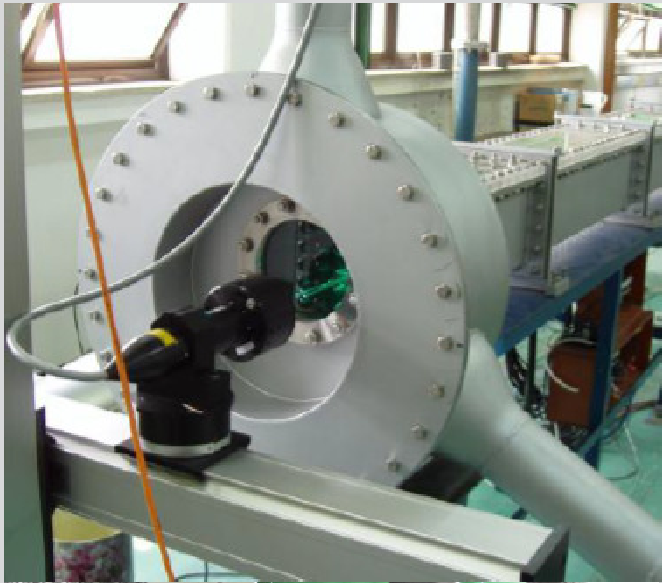
KAERI MATiS-H benchmark data are referenced in:

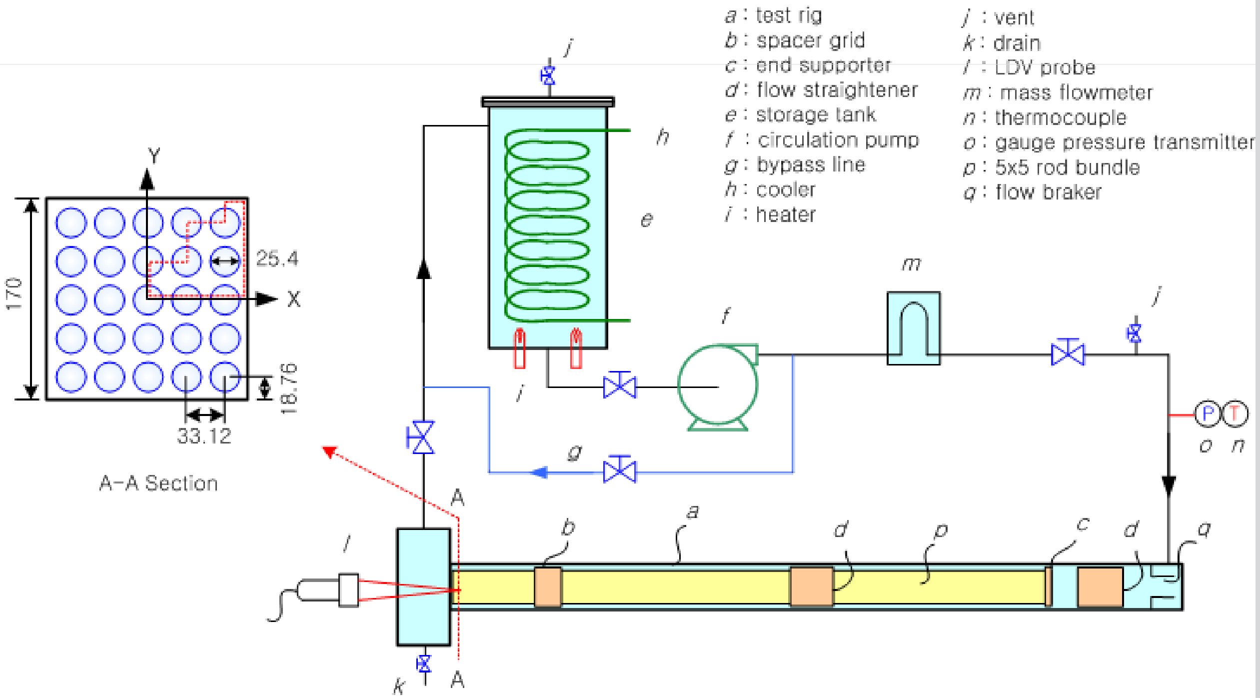
– Song C.-H. and Lee J. R. : "*OECD/NEA-KAERI Rod Bundle CFD Benchmark Exercise on Turbulent Mixing in a Rod Bundle with Spacers (MATiS-H) - Status Report on Experiments",* Open Meeting for CFD Benchmark Exercise, OECD/NEA Headquarter, 30. May 2012, Paris, France.

– Chang S.-K., Kim S. and Song C.-H.: "*OECD/NEA-MATiS-H Rod Bundle CFD Benchmark Exercise Test",* CFD4NRS-4, Conference on Experimental Validation and Application of CFD and CMFD Codes in Nuclear Reactor Technology, OECD/NEA and IAEA Workshop, 10.-12. September 2012, Daejeon, South Korea.

The KAERI MATiS-H Test Facility

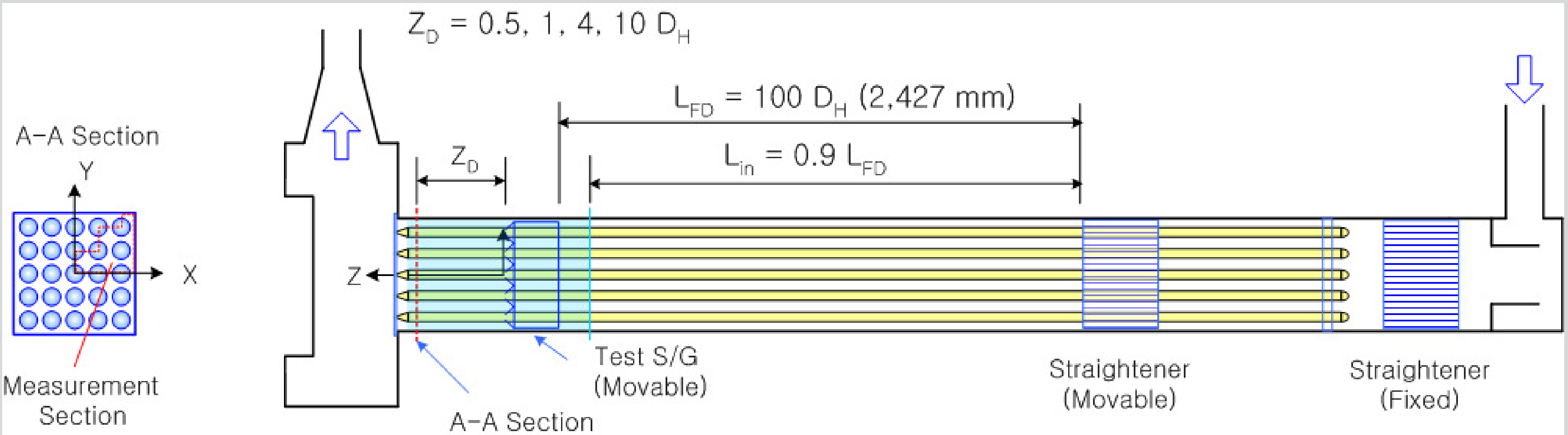
- Test facility is operated at Korea Atomic Energy Research Institute (KAERI), Daejeon, South Korea
- MATiS-H ≡ Measurements & Analysis of Turbulence in Subchannels – Horizontal





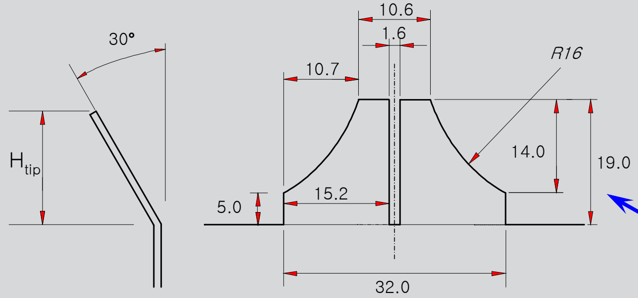
The KAERI MATiS-H Test Facility (cont.)

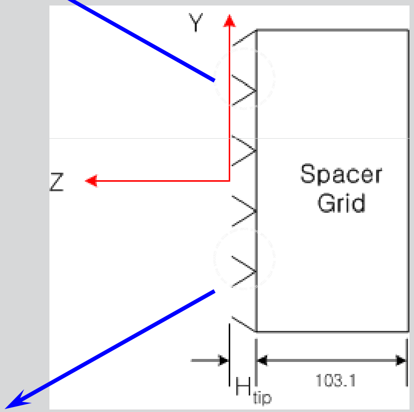
- Horizontal cold loop test section
- Inlet section and two flow straighteners designed such, that non-disturbed fully developed turbulent flow in 5×5 rod bundle is provided at inlet cross-section of investigated spacer grids
- Test spacer grid and 2nd flow straightener are movable in order to take axial LDV measurements 45+10mm upstream the end of the horizontal channel (red line)

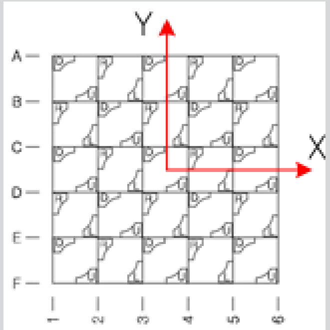


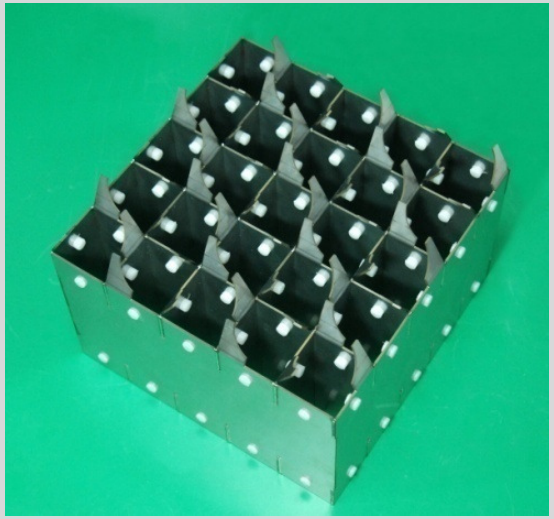
The KAERI MATiS-H Test Facility
Test Spacer Grids

Split-type Spacer Grid



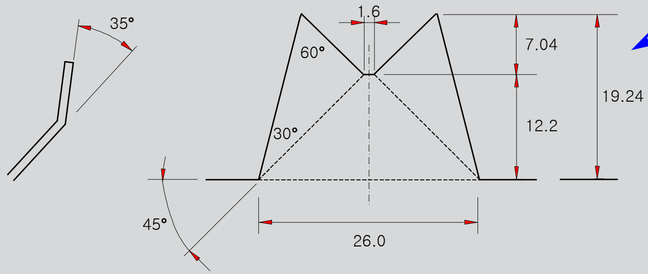


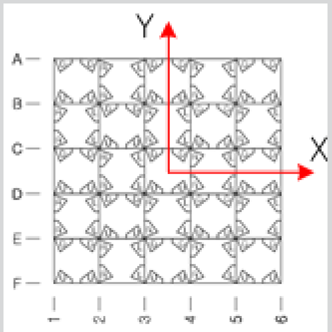




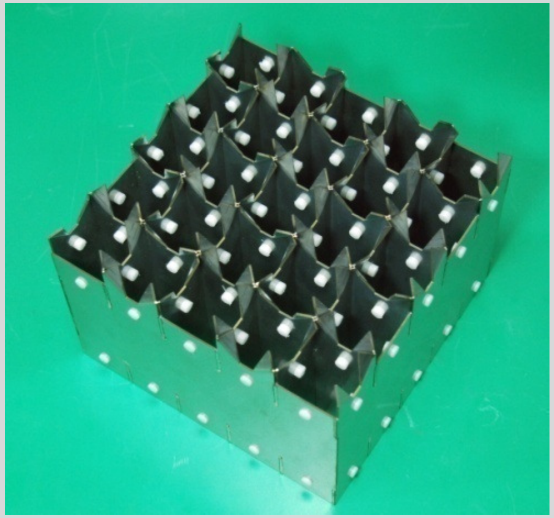
Split-type spacer grid

Swirl-type Spacer Grid





H_{tip} = 16.45 mm (Split-type)
17.97 mm (Swirl-type)



Swirl-type spacer grid

The KAERI MATiS-H Test Facility Test Conditions

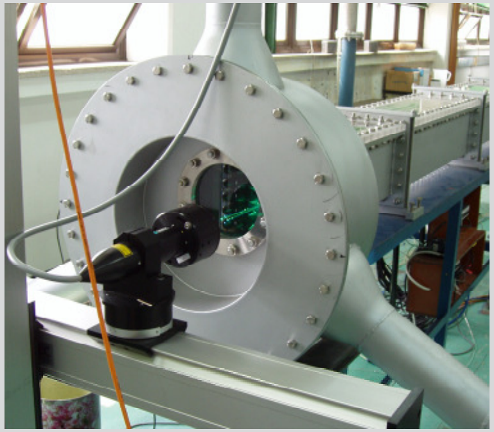
- **Fluid:** Water at T_{Water} = 35 °C
  ρ_F = 994.03 kg/m^3
  μ_F = 719.6 μPa s
- **Pressure:** p = 1.5bar
- **Mass flow rate:** $\dot{m}$ = 24.2 kg/s
- **Axial bulk velocity:** W_{bulk} ~ 1.5 m/s
- **Reynolds number:** Re ~ 50250

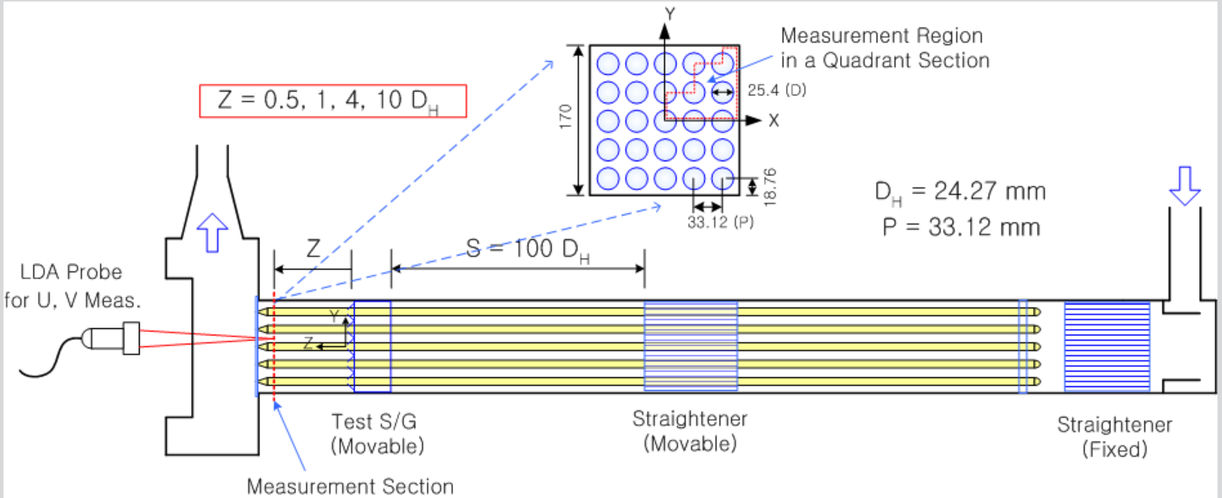
$$Re = \frac{\rho \cdot W_{Bulk} \cdot D_H}{\mu}$$

$$D_H = 4 \cdot \frac{flow\ area}{wetted\ perimeter} = 24.27 mm$$

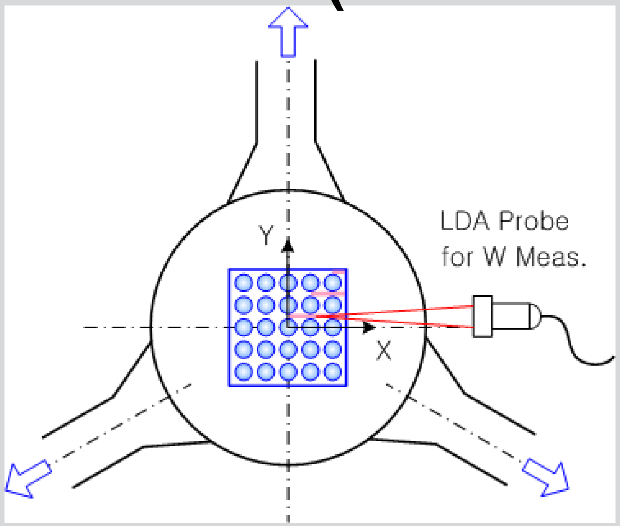
The KAERI MATiS-H Test Facility Measurements & Target Variables

- **Setup for Lateral Velocity Measurements (u,v and RMS)**

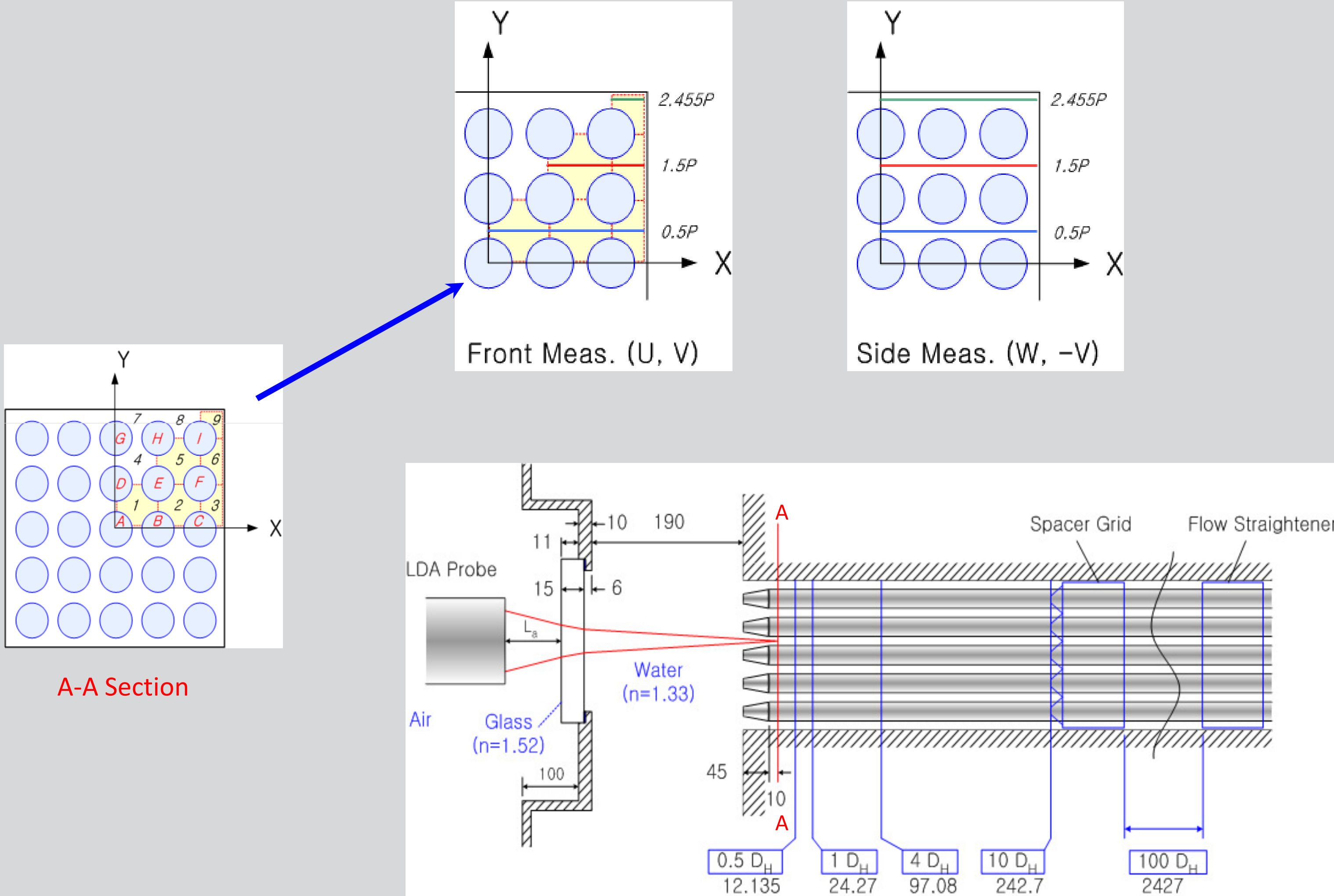




- **Setup for Axial Velocity Measurements (w and RMS)**



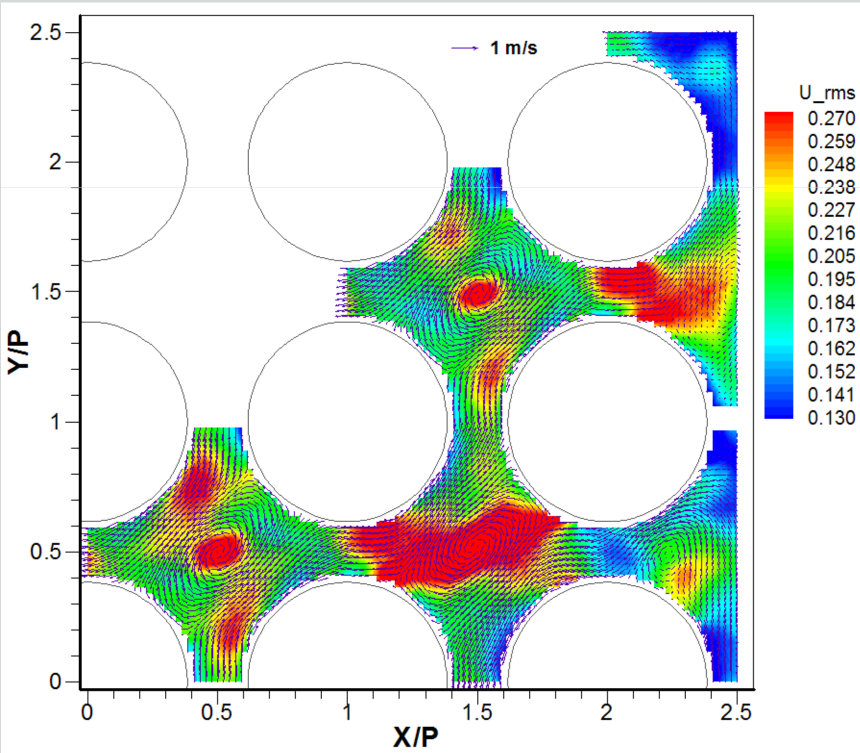
The KAERI MATiS-H Test Facility
Measurement Region for Downstream Tests



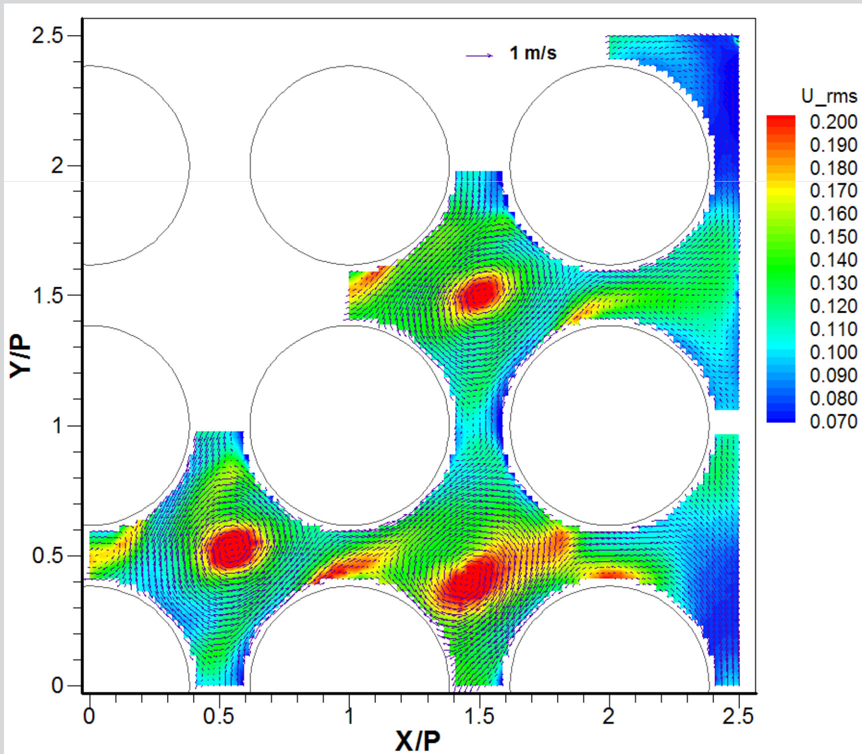
  August 9, 2013

The KAERI MATiS-H Test Facility Example of Measurements

- Images show an example of the lateral velocity map for the split type spacer grid at two different distances downstream of the tips of spacer guide vanes
- Vorticity maps are obtained as well



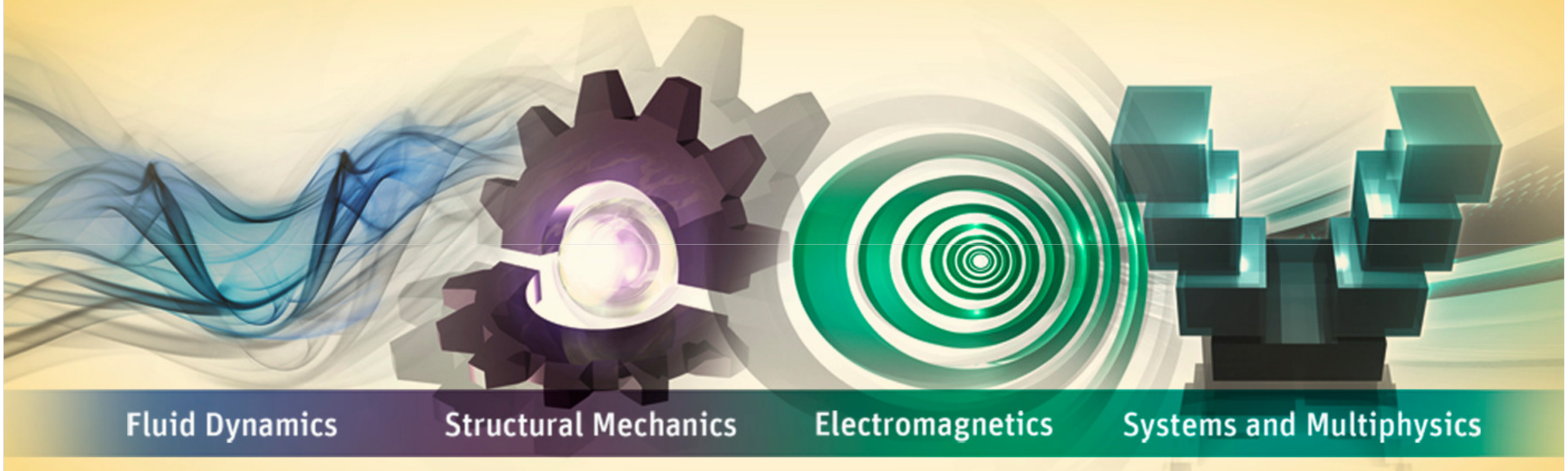
Z = 0.5 D_H



Z = 4 D_H

Realize Your Product Promise™

CFD Investigation Methodology

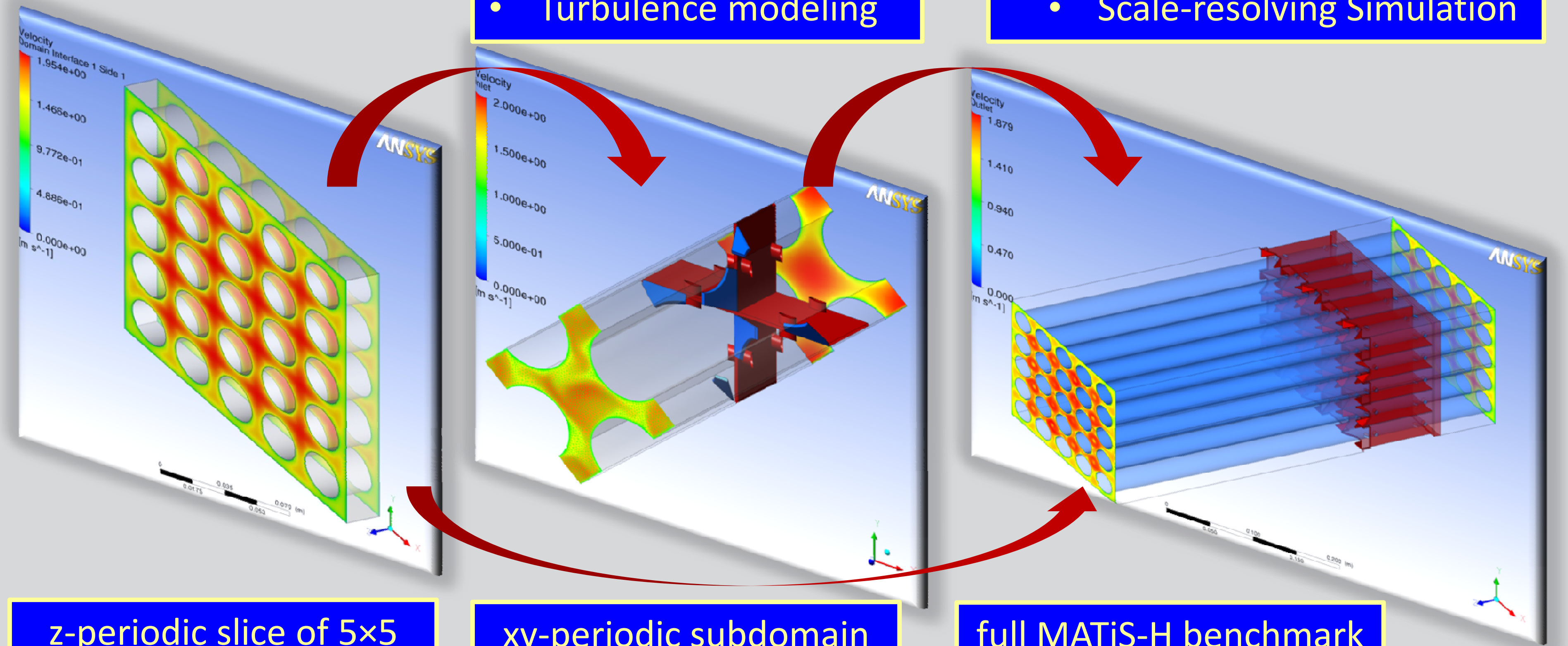


CFD Investigation Methodology

Provision & Validation of Inlet BC's

CFD Best Practice:
- Mesh dependence
- Numeric parameters
- Convergence
- Turbulence modeling

Final MATiS-H Benchmark Simulations:
- URANS SST-CC model
- URANS RSM models
- Scale-resolving Simulation



z-periodic slice of 5×5 rod bundle

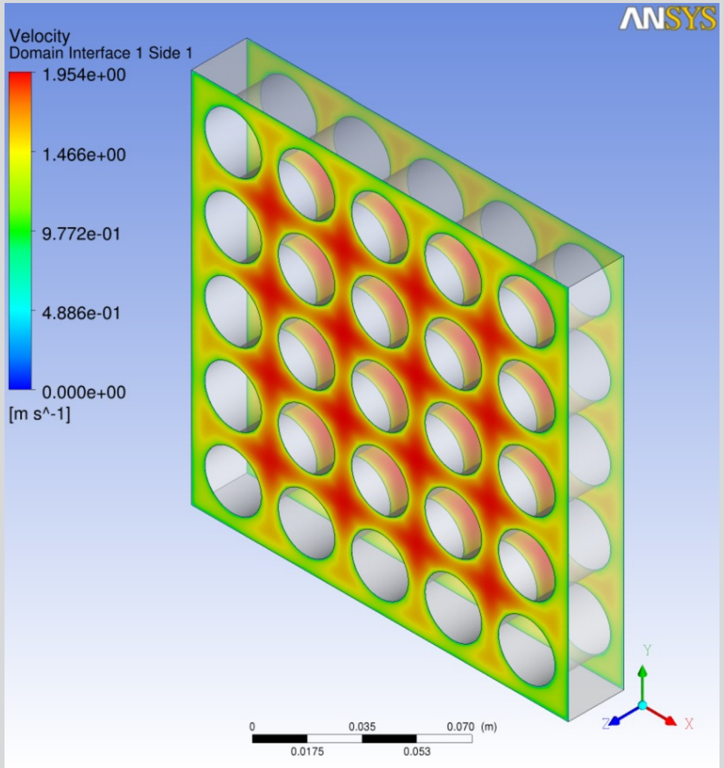
xy-periodic subdomain with representative split type spacer element

full MATiS-H benchmark geometry with test spacers (split & swirl)

CFD Investigation Methodology

Step 1: Provision & Validation of Inlet BC's

- **Objective:**
  - Generate inlet BC's for full MATiS-H geometry simulations
  - Compare CFD solutions on structured & unstructured meshes
  - Compare ANSYS Fluent and ANSYS CFX
  - Compare to KAERI data

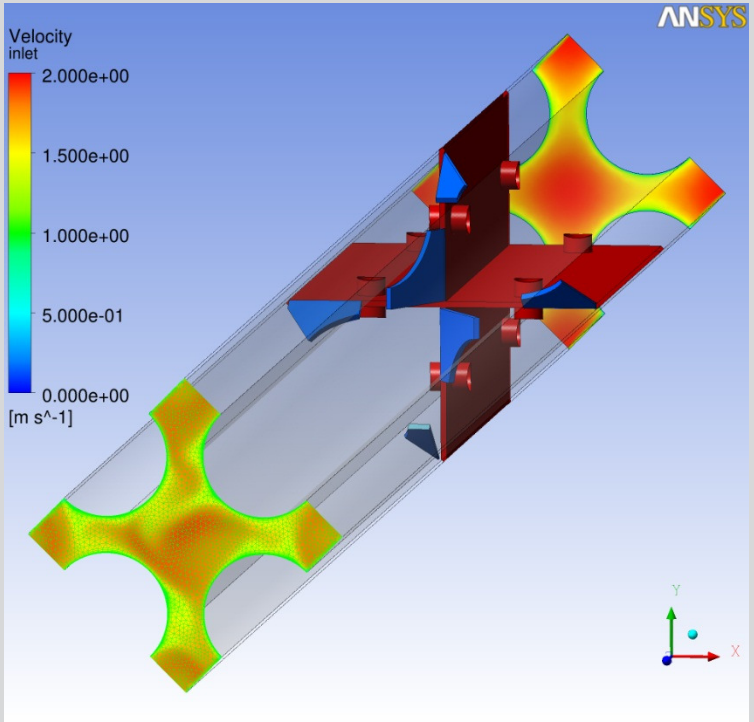


- **Geometry:**
  - Thin slice through the 5×5 rod bundle without spacer
  - z-periodic boundary conditions on top/bottom
  - Prescribed mass flow rate of 24.2 kg/s

CFD Investigation Methodology

Step 2: Investigations related to CFD Best Practices

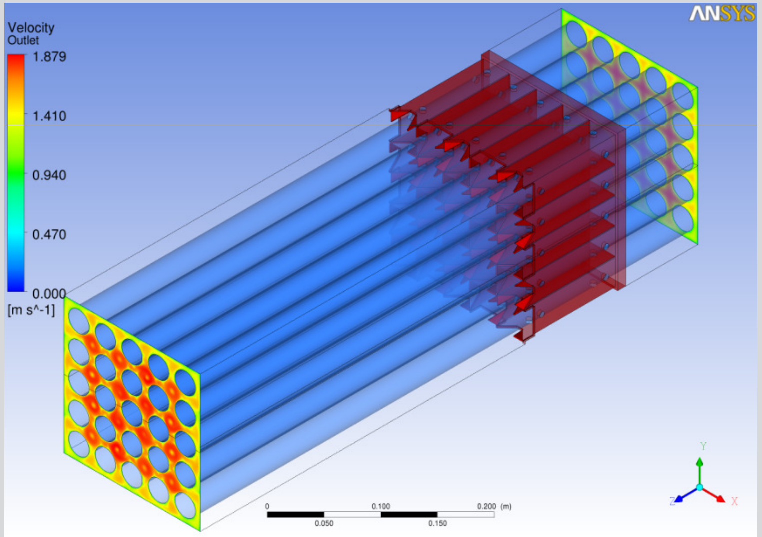
- **Objective:**
  - Investigate flow through representative spacer grid element in accordance with CFD Best Practice Guidelines
  - Limit the comp. effort for these investigations to be feasible
  - Identify required meshing approach and mesh resolution
  - Identify numeric parameters for reliable convergence & required accuracy level
  - Compare time-averaged URANS solutions (SST-CC, ωRSM) to SRS simulations (SAS-SST, WMLES)
- **Geometry:**
  - Reduced computational domain with representative split type spacer grid element including buttons & guide vanes
  - Cut out of the full geometry
  - xy-periodic boundary conditions
  - reduction factor for computational effort in comparison to full geometry is approx. 12.5



CFD Investigation Methodology

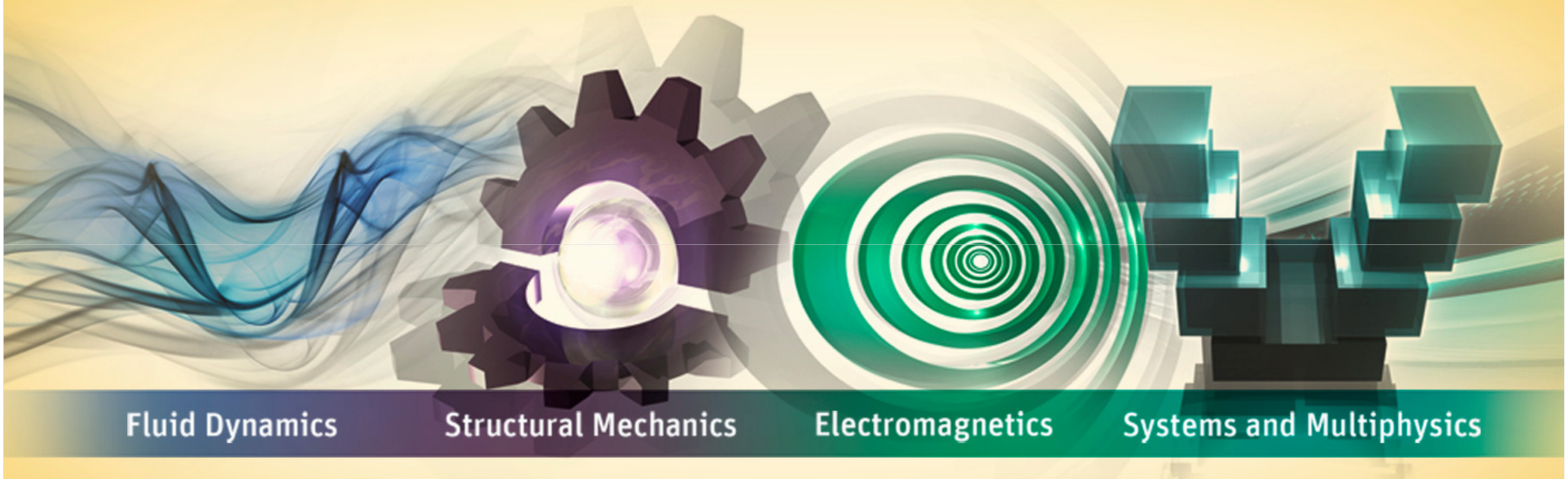
Step 3: Final MATiS-H Benchmark Simulations

- **Objective:**
  - Provide ANSYS CFD solutions for full MATiS-H geometry and for both types of investigated spacer grids (split & swirl type spacer)
  - Compare ANSYS Fluent and ANSYS CFX
  - Compare time-averaged URANS and SRS
  - Compare to KAERI data



- **Geometry:**
  - **Split S/G:** full 5×5 rod bundle with split type spacer grid
    Swirl S/G: half 5×5 rod bundle with swirl type spacer grid and application of 180 degree rotational periodicity
  - Inlet BC profiles for u,v,w and turbulence properties from Step 1
  - Simulation approach, mesh resolution and numeric parameters determined from Step 2

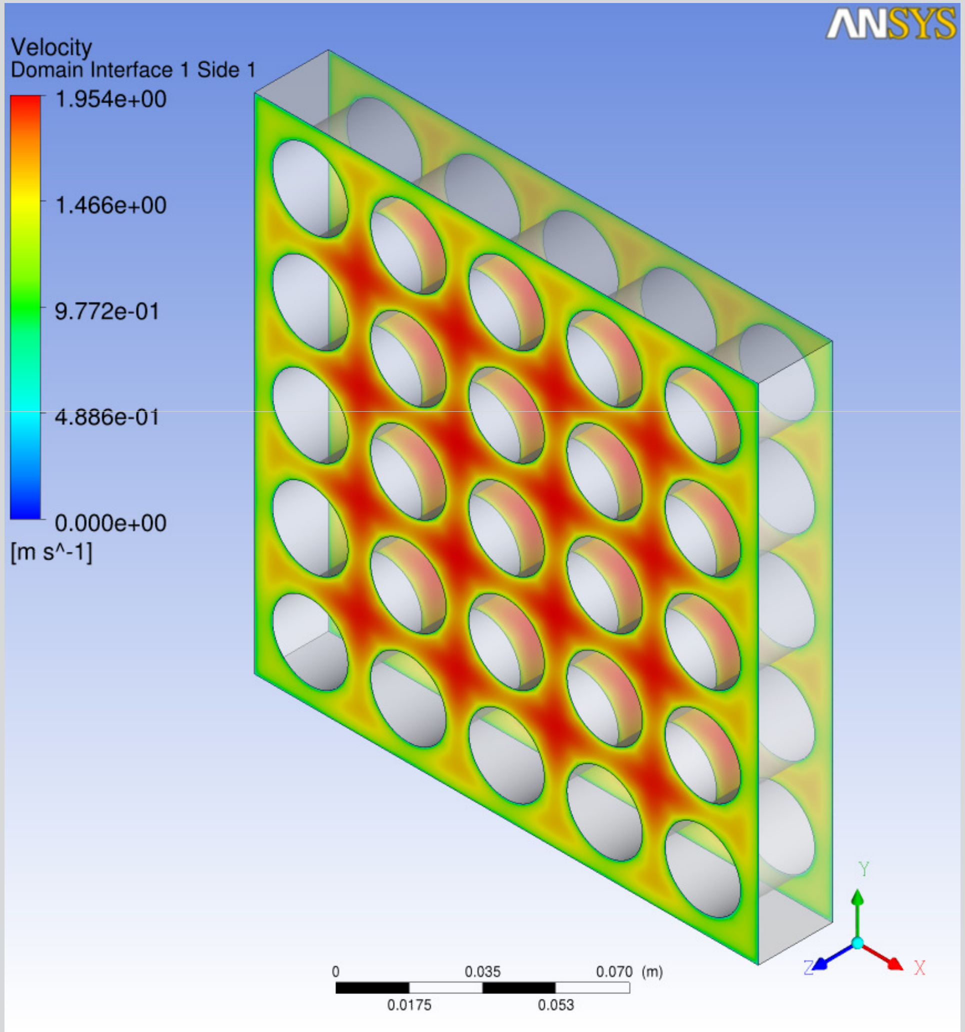
Step 1: Provision & Validation of Inlet BC’s



Investigated Geometry and Flow Conditions

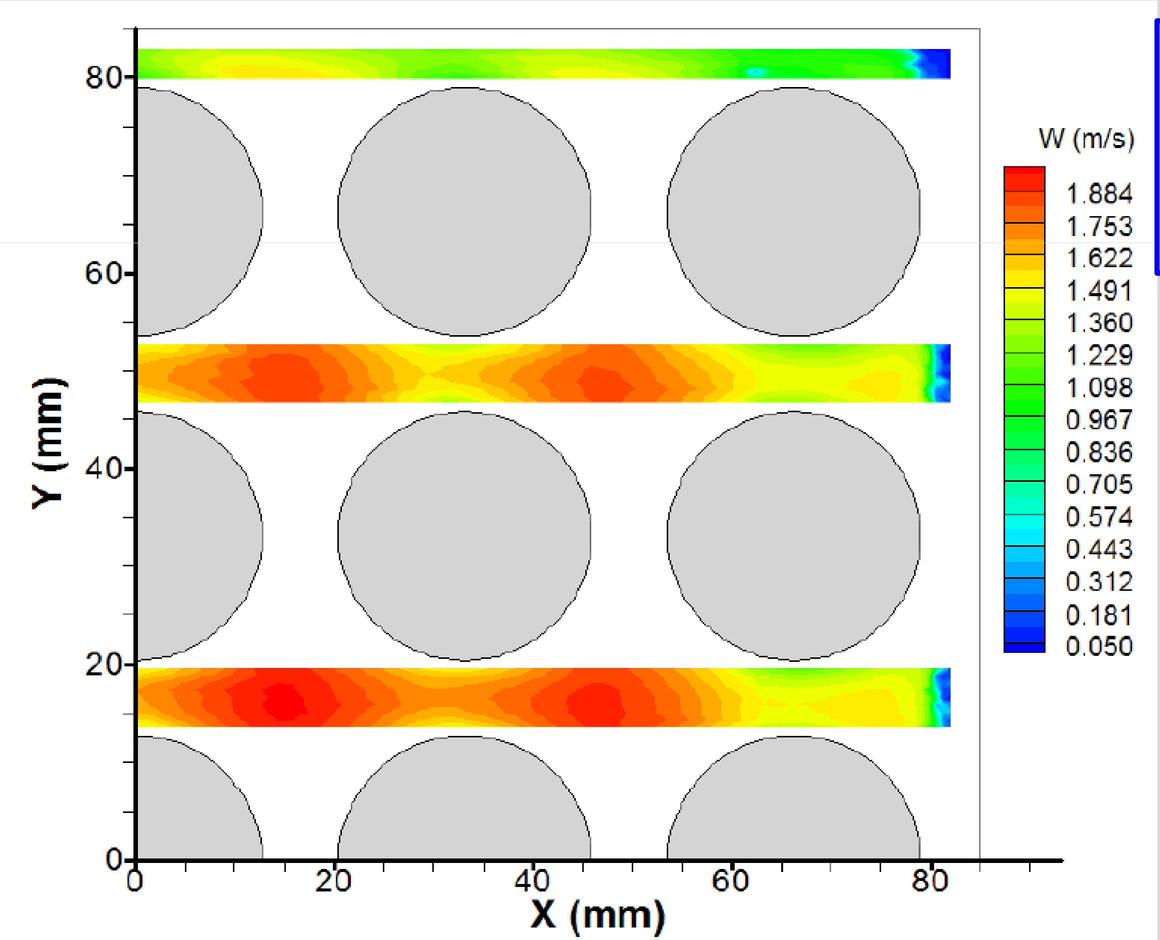
- **MATIS-H 5x5 rod bundle without spacer grid**
- **z-periodic geometry length = 0.045mm**
- **Mass flow rate: 24.2 kg/s**
- **Water @ 35°C**
  - **ρ=994.03 kg/m^3**
  - **μ=0.0007196 Pa s**
- **Reynolds number:**
  - **CFD: Re=50 828**
  - **Exp.: Re=50 250±2% =50 250±1005**

calculated from bulk velocity w_{bulk}=1.5m/s and hydraulic diameter D_H



KAERI Measurements of Inlet BC's

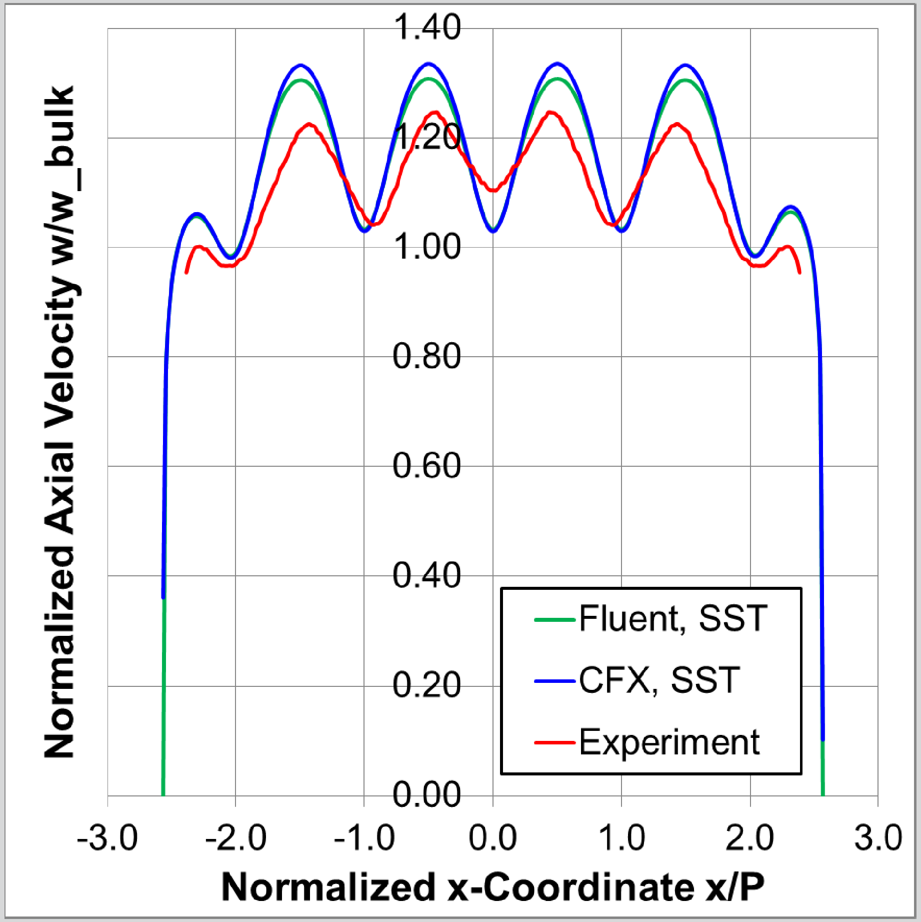
- **Measurements in 3 horizontal cross sections through the 5x5 rod bundle**



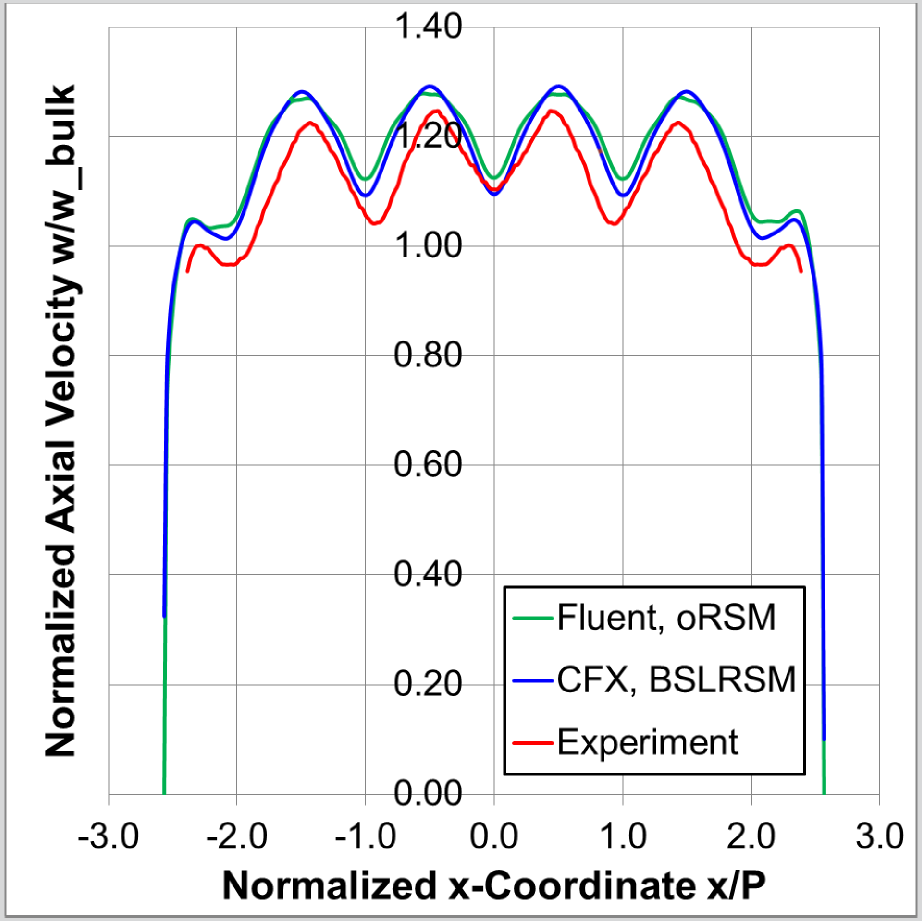
CFD-data comparison at:
- y/P=0.5
- y/P=1.5
- y/P=2.455

Comparison to KAERI Data
Normalized W Velocity along Y=1.5P

SST turbulence model



ω-based RSM turbulence models



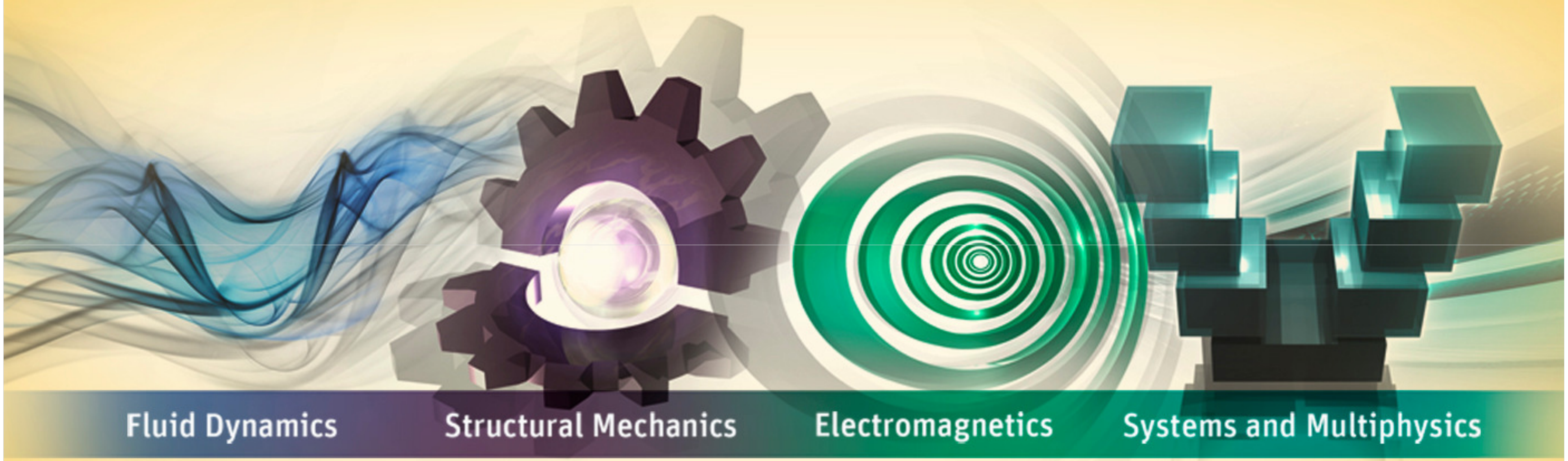
Conclusions

- ω-based RSM models seem to be the more accurate model for rod bundle flows; in SST model the missing secondary flows lead to overpredicted minima and maxima in the axial velocities at the center locations of subchannels
- Comparison of axial velocities in satisfactory agreement with data
- Good agreement between RANS solutions in Fluent and CFX
- Issues found in near-wall KAERI data
  → corrected by KAERI by clipping the data where the measurements were not reliable
- Cross-sectional distributions of u, v, w, k, ω (SST) and Reynolds stresses for RSM models are extracted from these precursor simulations and later used as inlet boundary conditions for the MATiS-H benchmark simulations in Step 3

ANSYS

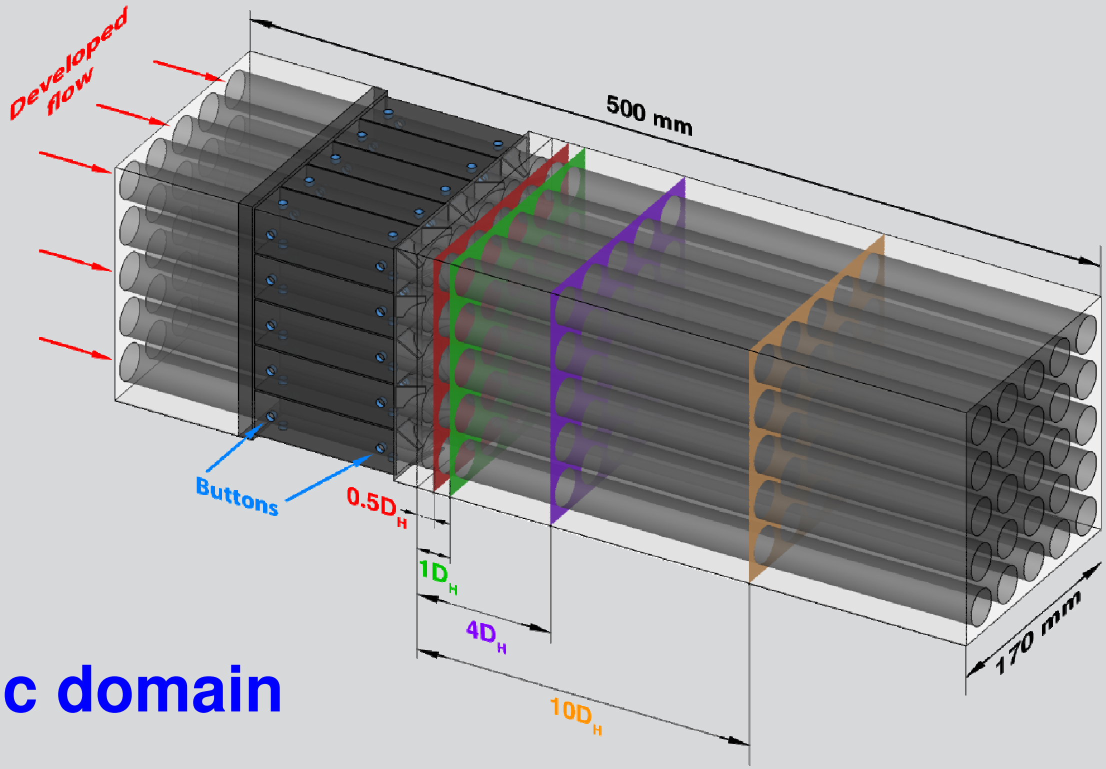
Realize Your Product Promise™

Step 2: Investigations related to CFD Best Practices



MATiS-H (CFD) Problem Definition

- 5×5 rod bundle geometry with spacer grid requires mesh resolution of about 100M mesh elements
- Resulting computational requirements will not allow for application of CFD Best Practice Guidelines
- Need to reduce the application to a feasible size to investigate:
  - Simulation approach
  - Numerics
  - Mesh resolution
  - Turbulence modeling



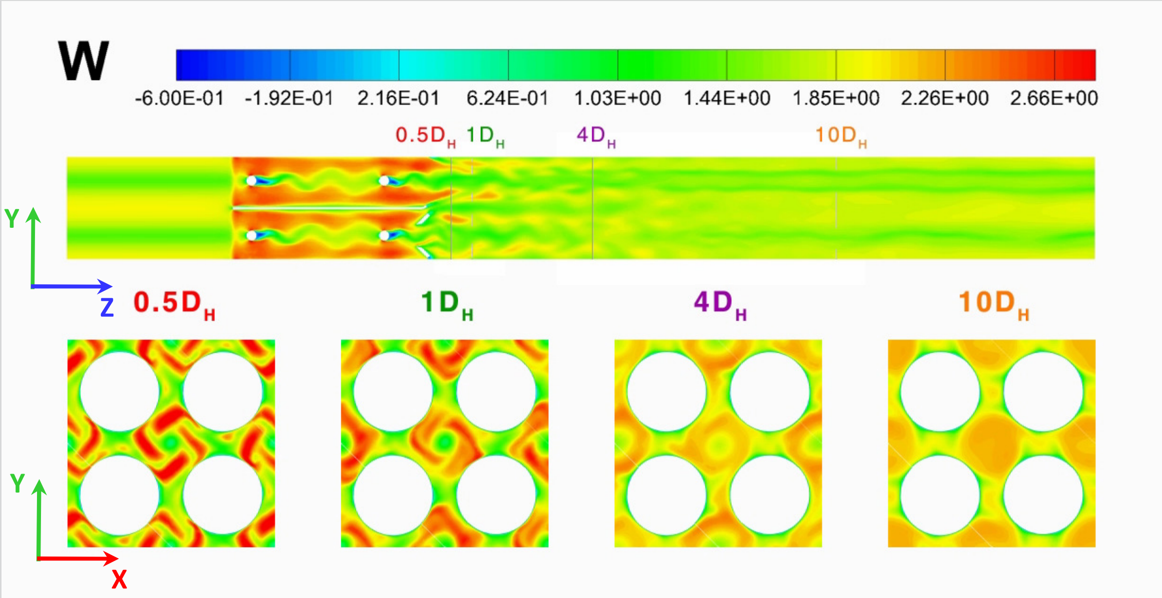
→ Reduced xy-periodic domain

Precursor Simulations
– Transient Flow Structures –

- **Expected unsteady flow around buttons**
  - Buttons ≡ cylinder in cross flow → von Karman vortex street

$$\mathbf{Re_d = W_{bulk} \cdot d/\nu \approx 10^4}$$

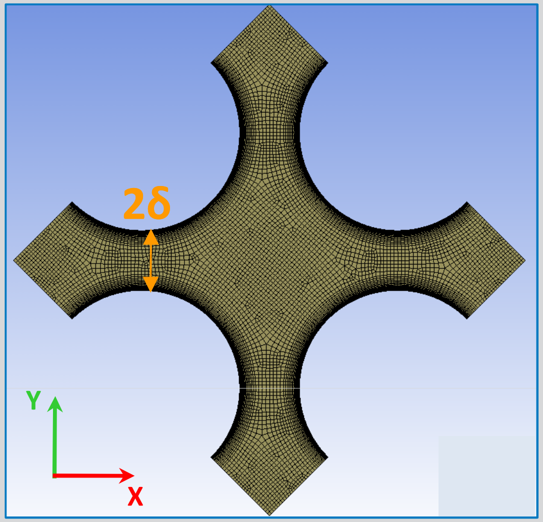
  - Button diameter: d = 6 mm
- **Consequently transient simulation with statistical time-averaging of results should be applied to this application**
  - URANS – Unsteady RANS
  - SRS – Scale Resolving Simulations



- **Shows oscillating flow patterns**
- **Intensive vortex is created downstream the spacer guide vanes**
  - Main vortex in the channel center
  - Additional vortices spawned from each of the buttons, interacting with guide vanes of the spacer

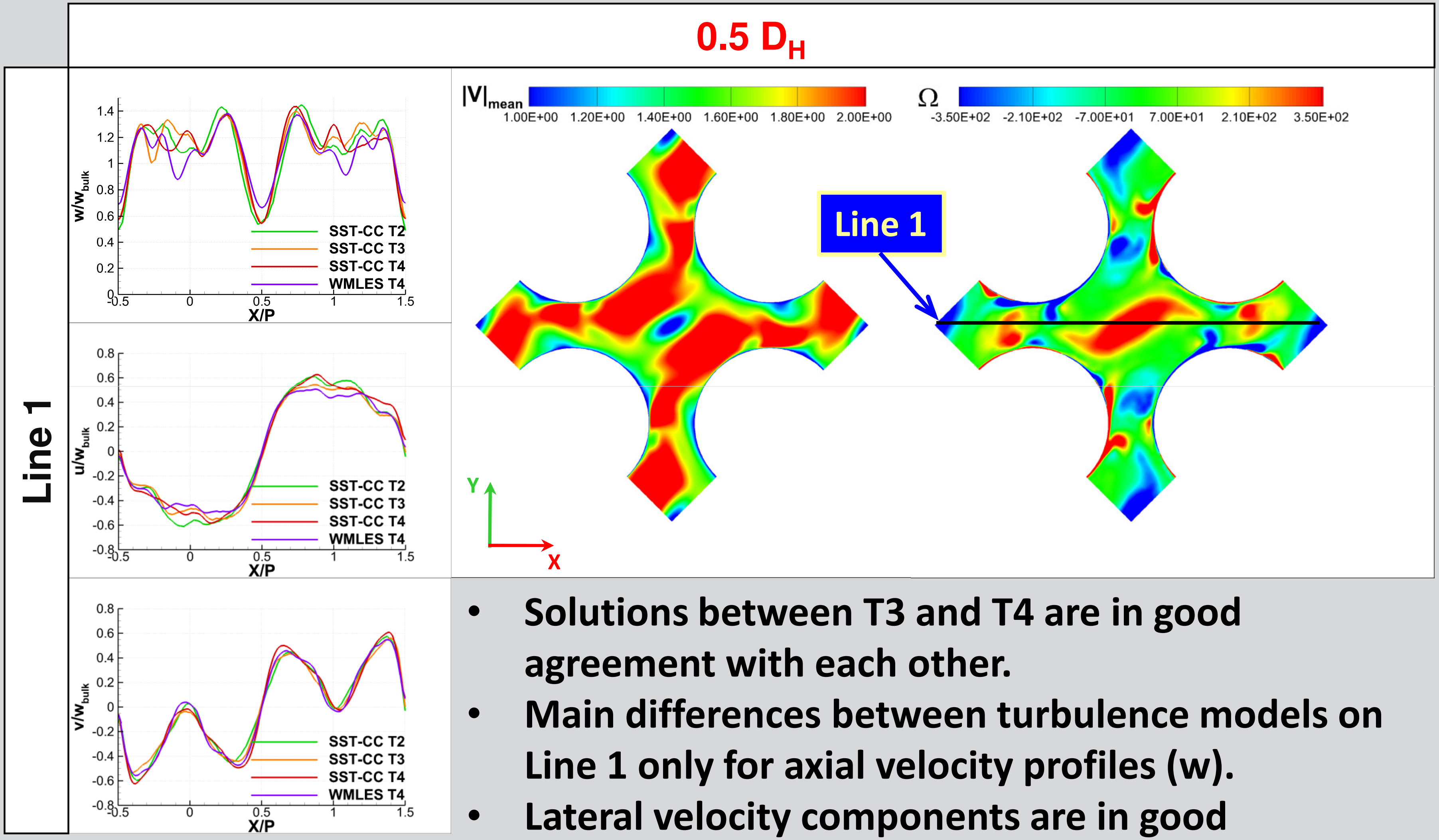
Meshes for xy-Periodic Subdomain with Representative Split Type Spacer Grid Element

- **URANS** (T1-T3)
  - Tet/prism mesh for spacer grid geometry
  - Extruded mesh (prisms/hex) for rod bundle geometry before and after the spacer
- **WMLES** (T4)
  - Hexa mesh for simple geometry before/after spacer grid
    - $\Delta z/\delta = 0.1$
    - $\Delta y_1^+ = 1.5$
  - Tet/prism mesh for spacer grid geometry



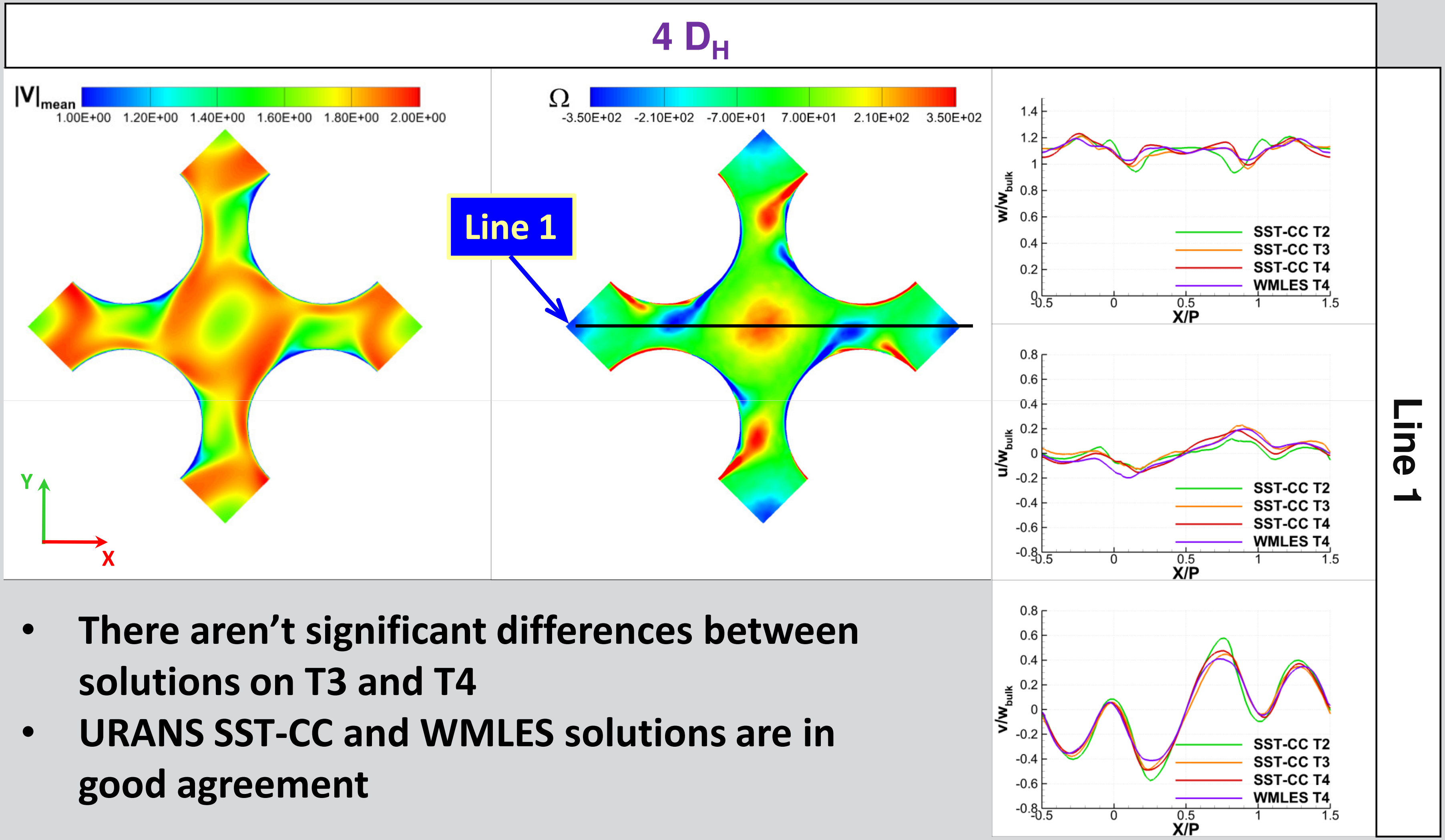
Cells	T1	T2	T3	T4
Number of cells	1.3 M	2.5 M	7.6 M	40 M
Resulting mesh count for full domain	17 M	32 M	97 M	-
Max. Δy_1^+	17	13	8	5
Mean Δy_1^+	7	5	3	1.5
Min. cell volume, mm^3	0.05	0.04	0.03	0.001
Max. cell volume, mm^3	2	2	2	1

Flow Pattern at z=0.5 D_H



- Solutions between T3 and T4 are in good agreement with each other.
- Main differences between turbulence models on Line 1 only for axial velocity profiles (w).
- Lateral velocity components are in good agreement.

Flow Pattern at z=4.0 D_H



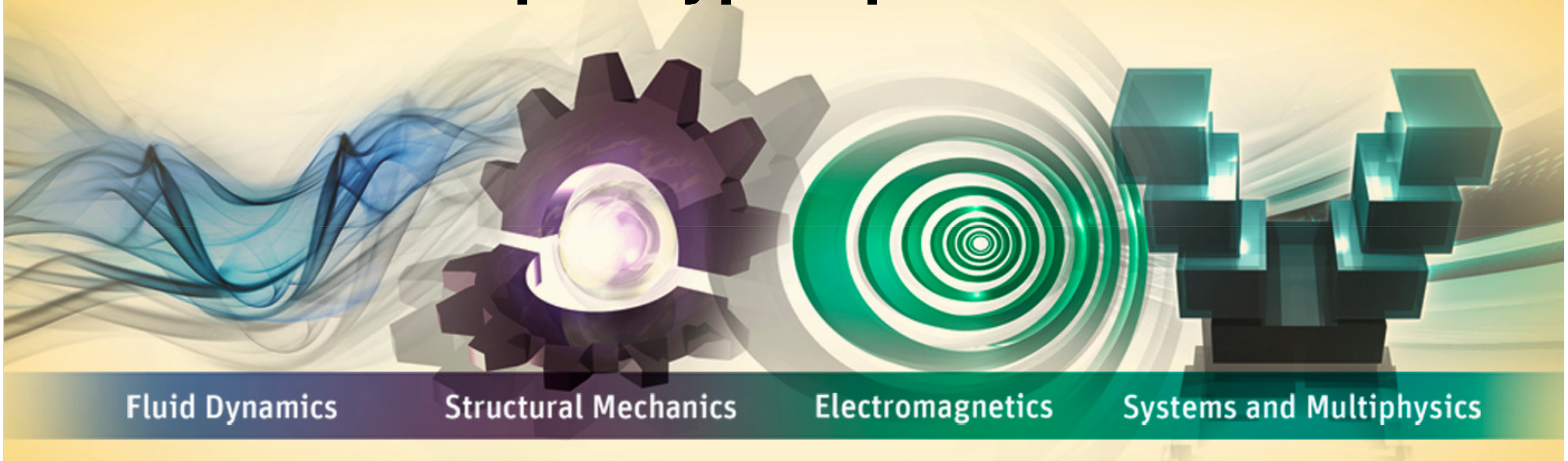
- There aren't significant differences between solutions on T3 and T4
- URANS SST-CC and WMLES solutions are in good agreement

Conclusions

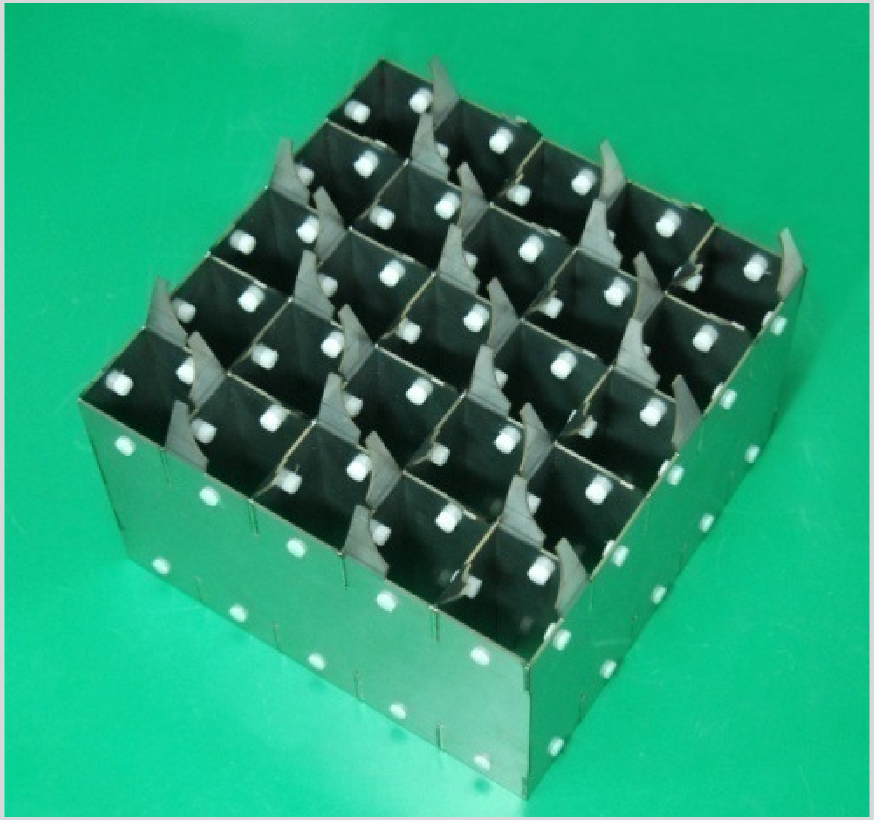
- Mesh resolution of Mesh T3 should be sufficient
- Numerics settings for reliable converged CFD solutions identified
- Flow development time and averaging time identified
- URANS SST-CC or ω-based RSM models should be sufficiently accurate
- Full scale-resolving simulation used as reference, but most likely too computational intensive in the blind phase of the MATiS-H benchmark
- Uncertainty:
  - There might be an influence from the xy-periodic boundary conditions on LES results
  - Influence of exterior channel walls not represented here

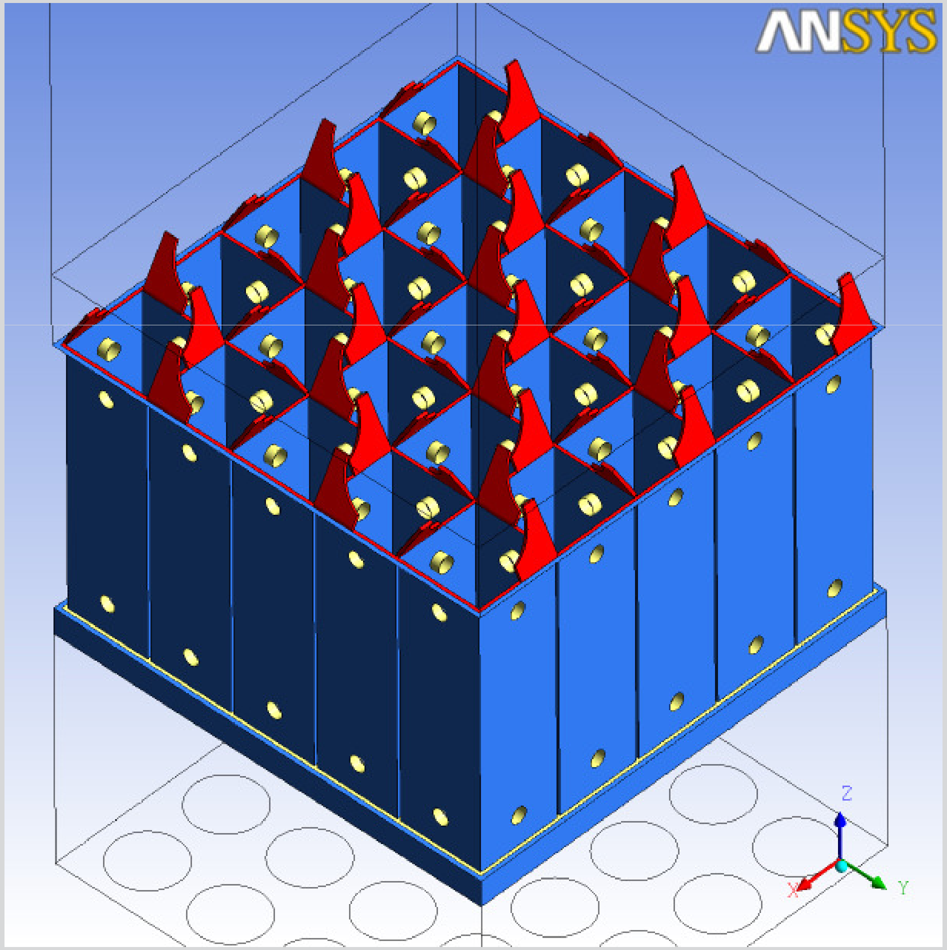
Realize Your Product Promise™

Step 3a: Final MATiS-H Benchmark Simulations
– Split Type Spacer Grid –

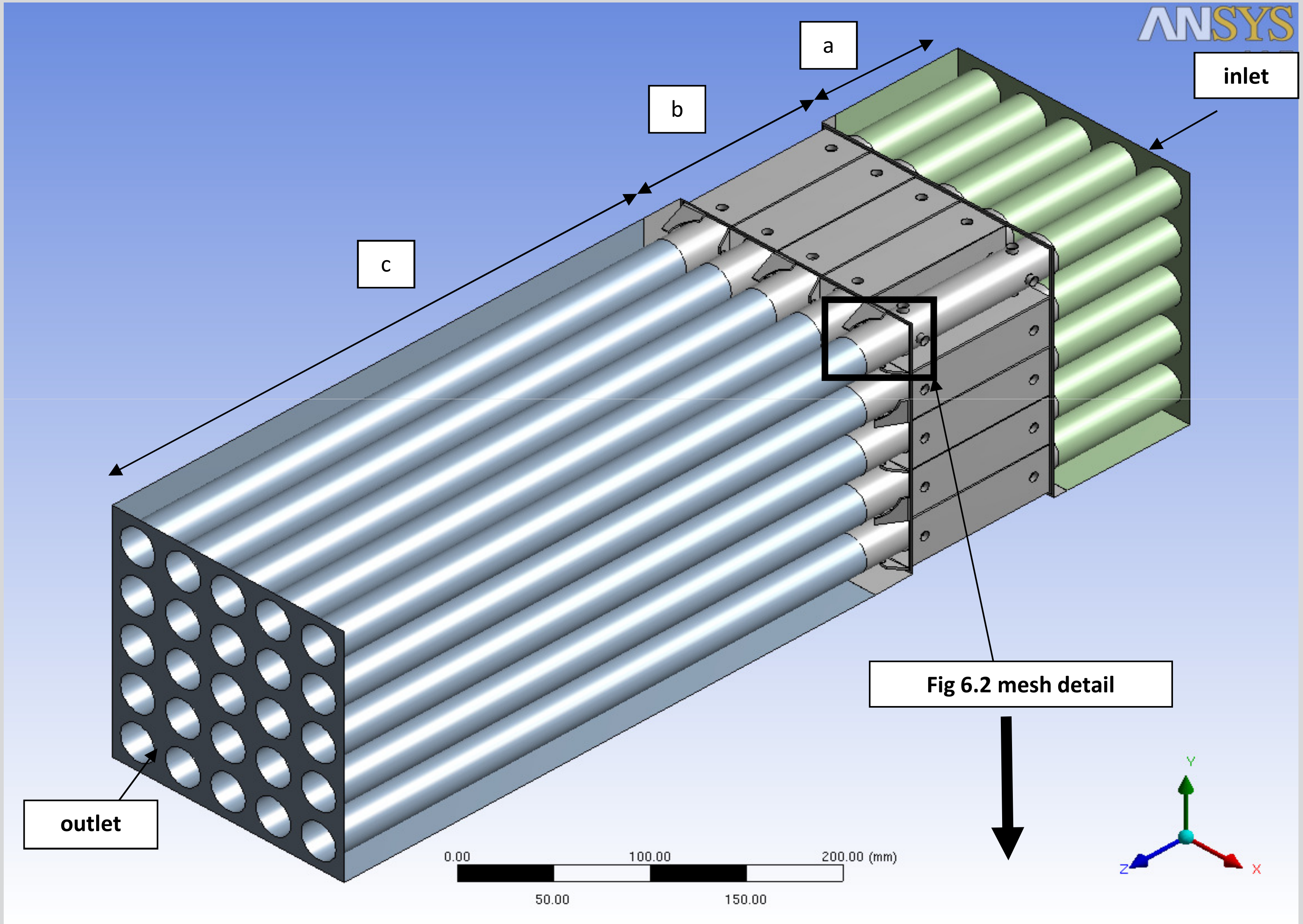


Split Type Spacer Grid with Buttons

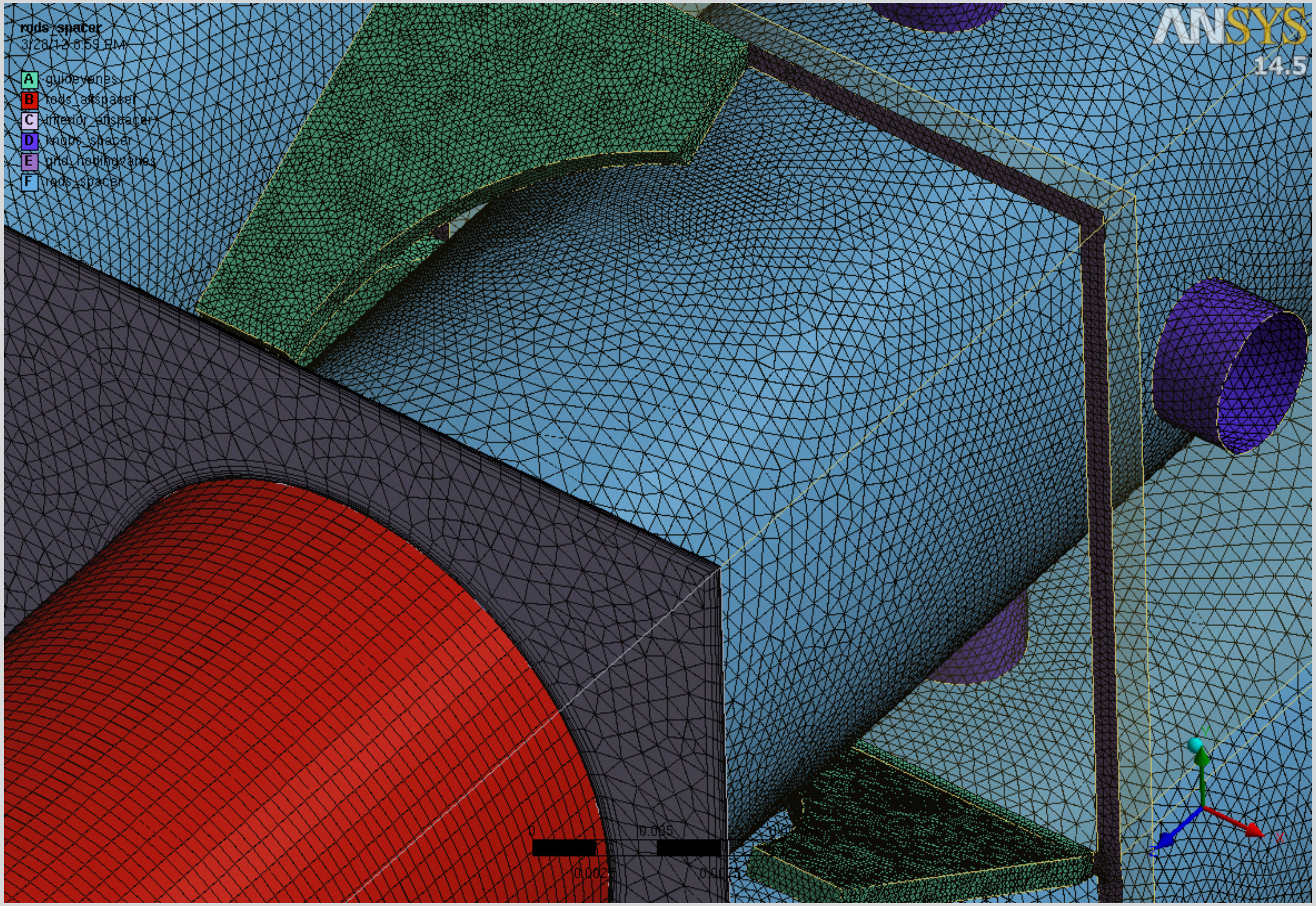




Computational Domain



Split Type Vanes - Mesh Detail



Mesh Hierarchy for Split Type Spacer

- **Hybrid mesh using ANSYS Workbench meshing**
- **Tet/Prism mesh in spacer grid, part (b)**
- **Sweeping applied to rod bundle, parts (a) and (c)**
- **All mesh interfaces are conformal mesh (no GGI)**

	Mesh 1	Mesh 2	Mesh 3
Purpose of the mesh	CFD setup derivation	RANS/URANS tests	RANS/URANS productive mesh, SAS-SST & ZLES
Number of elements	11.0M	31.5M	96.3M
Number of nodes	4.4M	15.4M	40.6M
Y^+_{max}	92.3	20.6	10.1/ (Fluent: 5.11)
Y^+_{mean}	39.6	9.5	4.2
Min cell size, mm	0.1	0.04	0.03
Min face angle [°]	6.0	6.5	9.6
Growth rate	1.2	1.1	1.05

[1] ANSYS CFX and ANSYS Fluent use node centered vs. cell centered discretization schemes, which affects the definition of Y^+ and leads to different Y^+ values for the same mesh. Here we specify the Y^+ values based on the ANSYS CFX simulation results. Due to the cell centered discretization of ANSYS Fluent corresponding Y^+ values on the same mesh are roughly by a factor of two smaller.

CFD Test Matrix for MATiS-H Split Type Spacer Grid

ANSYS CFD solver	Turbulence model	Mesh 2	Mesh 3
ANSYS CFX 14.0	SST-CC	X	X
	BSL RSM	X	X
	ZLES SAS-SST		X
ANSYS Fluent 14.0	SST-CC	X	X
	ω-RSM	X	X
	SAS-SST		X

CFD simulation methodology:

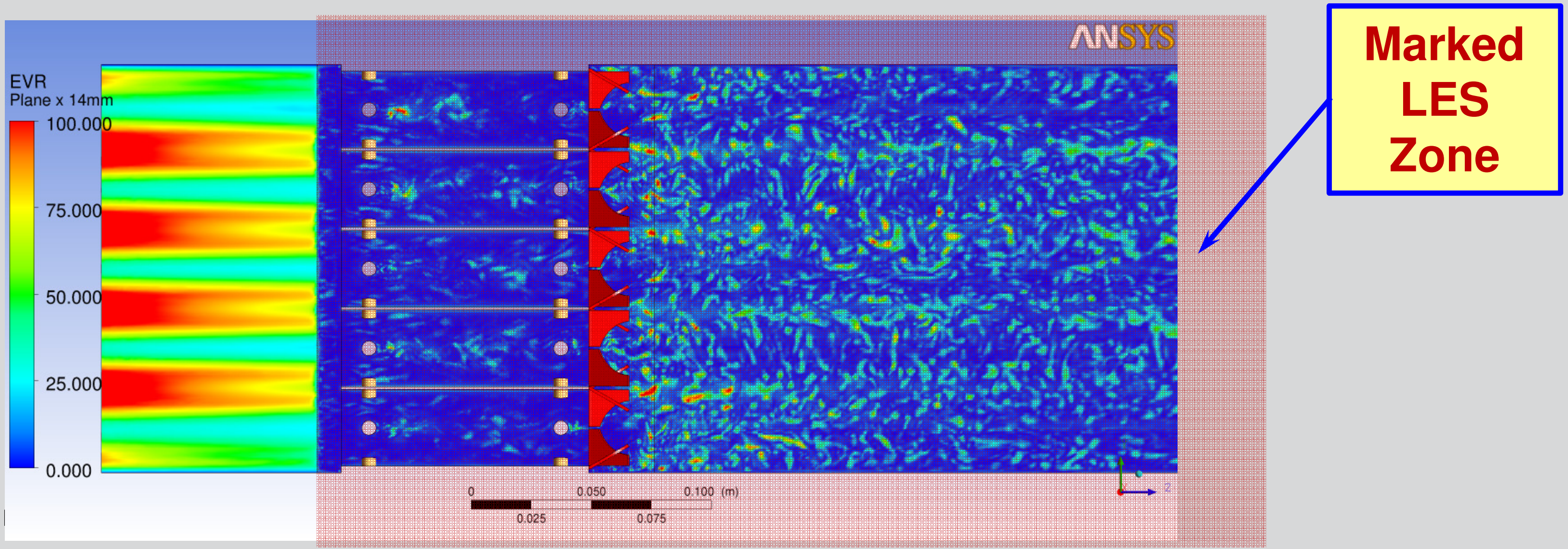
- **Start with steady-state simulation on coarsest mesh**
- **Initialization of transient flow simulation from the steady-state solution.**
  - **Time for transient flow development (T_1=0.25s).**
  - **Time for statistical time-averaging of results (T_2=0.6-1.25s).**
- **For transient flow simulations on refined meshes the CFD simulation was initialized with the final result from the transient CFD simulation on the previous coarser grid level.**
- **SRS simulations had been initialized with URANS SST-CC results on same grid level.**

SRS Setup using Zonal LES (cont.)

- One possible approach to trigger the switch from URANS SST to LES mode is Zonal LES (ZLES) in ANSYS CFX or Embedded LES (ELES) in ANSYS Fluent.
- For ZLES a bounding box is specified, where the turbulence model approach is forced to switch to LES mode. In the ANSYS CFX simulation the LES zone marker is specified by:

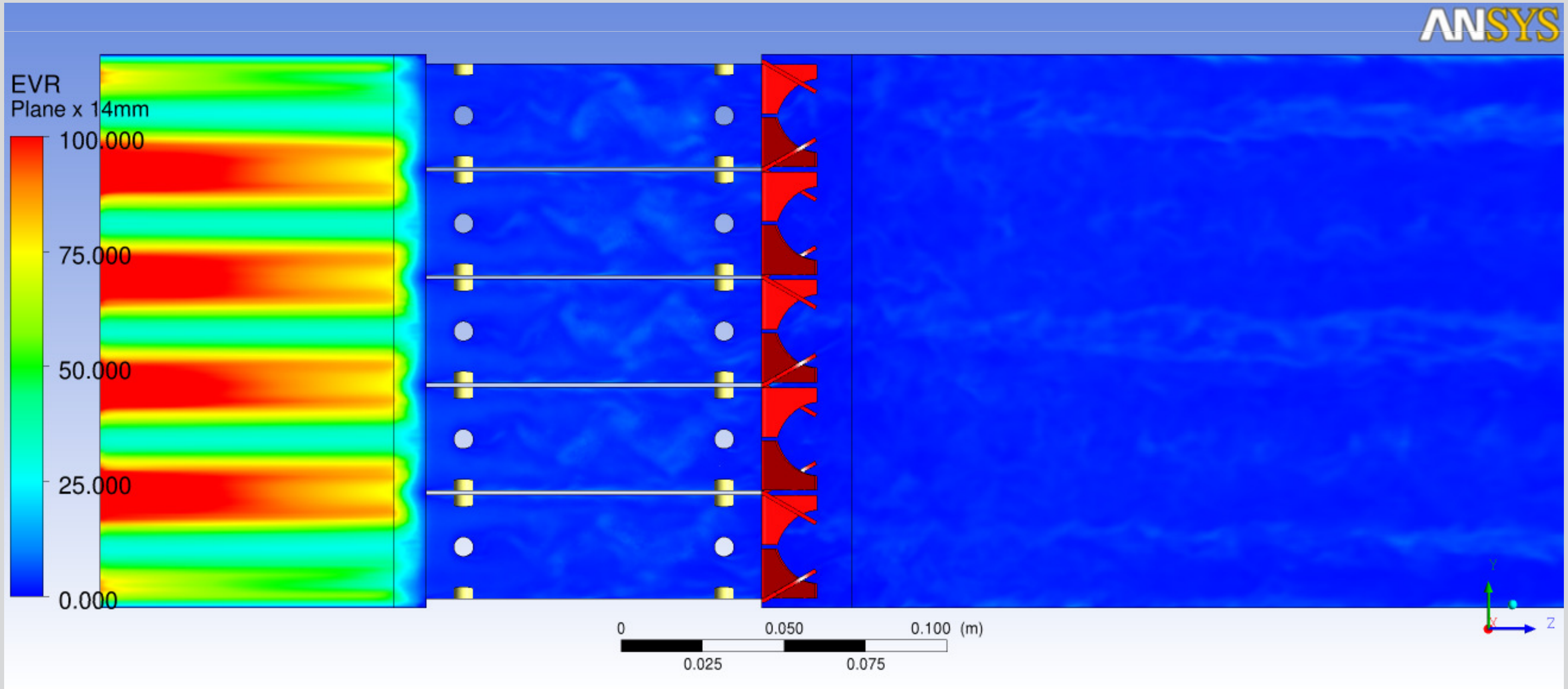
```
Zone Marker = step(z/1[m]+0.01)
```

- Plot of eddy viscosity ratio shows the switch from URANS SST to LES mode in ANSYS CFX simulation at entry to spacer grid

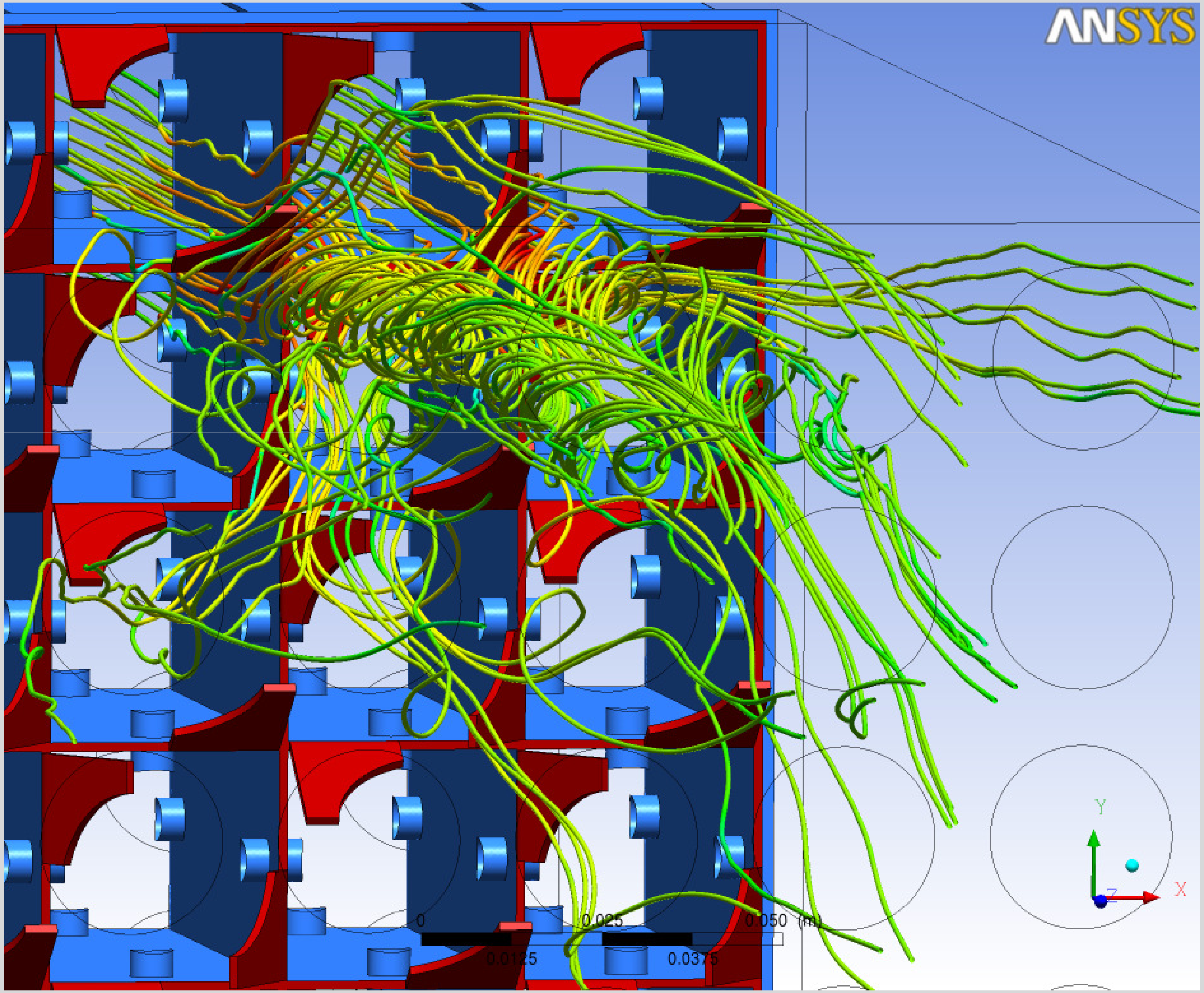


SRS Setup using SAS-SST for ANSYS Fluent

- Standard SAS-SST was used for the ANSYS Fluent simulation, i.e. without ELES approach using the automatic switching to LES mode in the SAS-SST model
- Image shows, that SAS-SST is automatically switching from URANS to LES mode with flow acceleration at the inlet cross section to the split type spacer grid

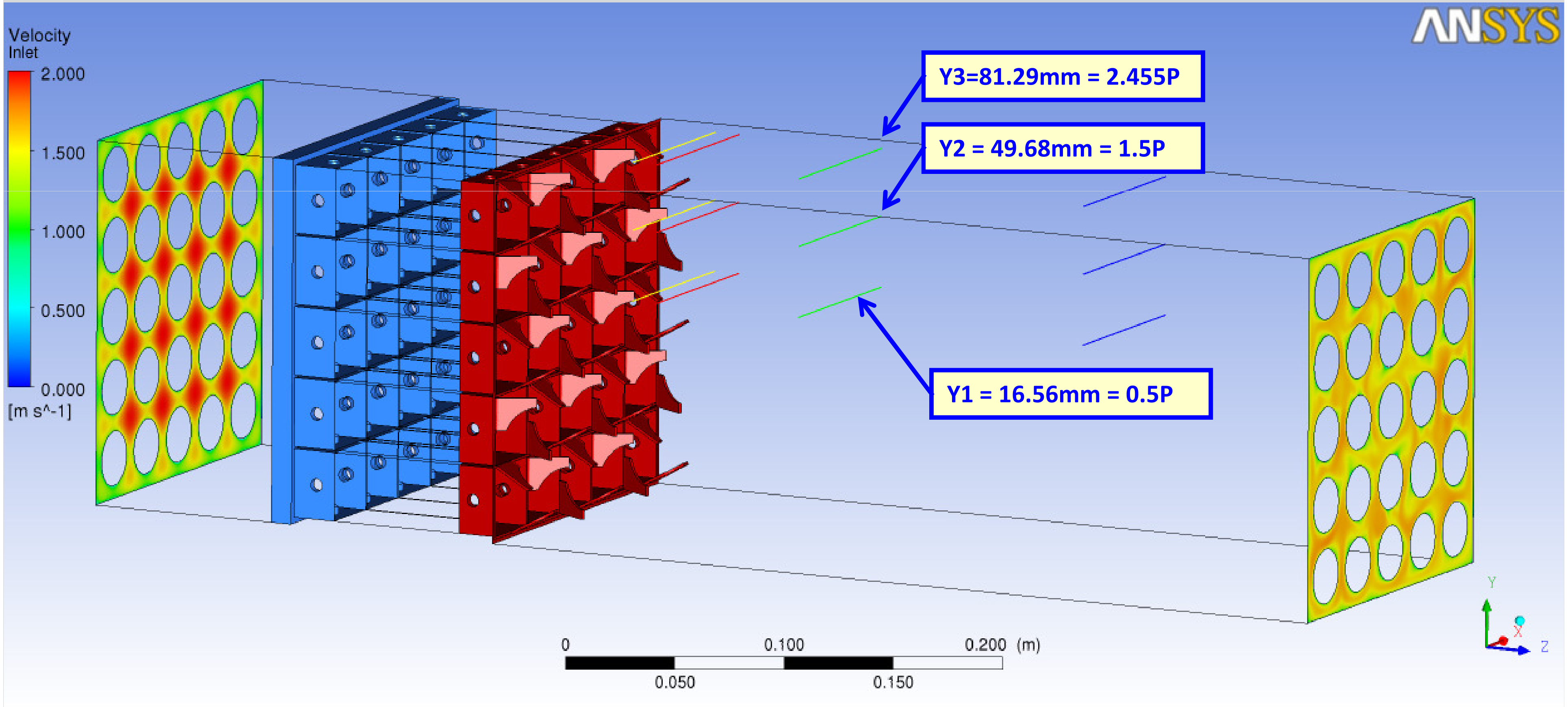


Streamlines for Split Type Spacer Grid



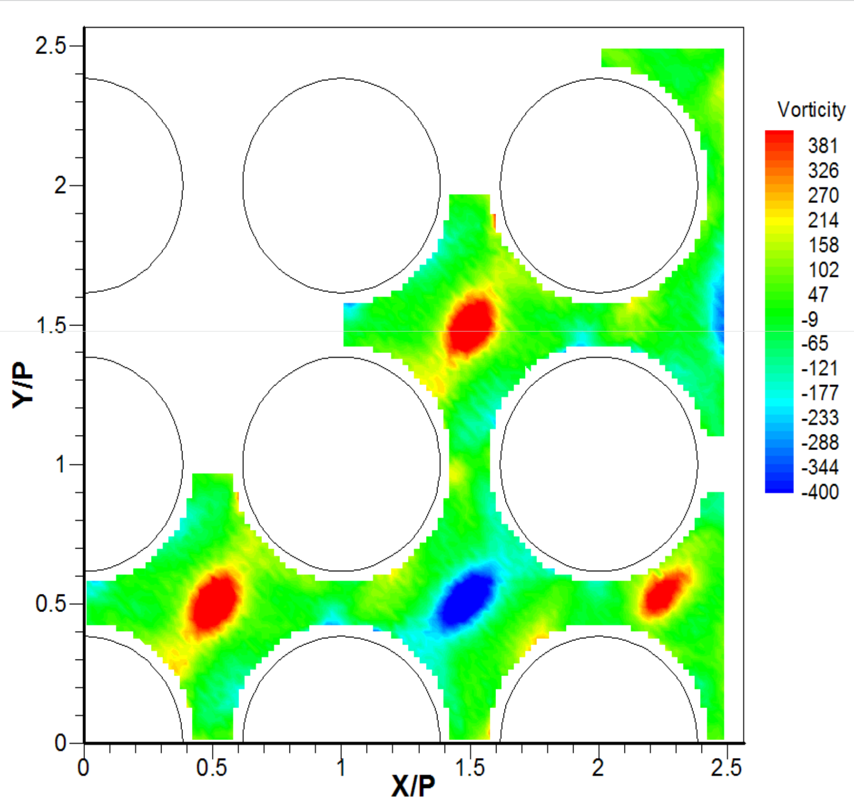
Location of Comparison Line Profiles

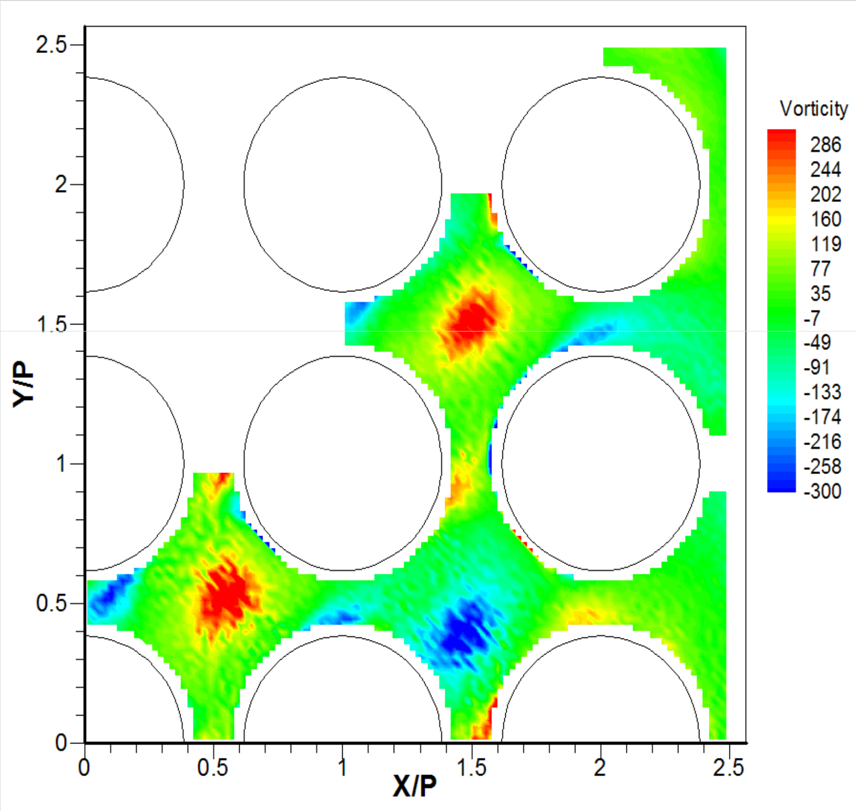
- **KAERI measurements at 4 axial distances to the tip of spacer grid guide vanes:**
  $z=0.5\cdot D_H$, $z=1.0\cdot D_H$, $z=4.0\cdot D_H$ and $z=10.0\cdot D_H$



KAERI Measurements
Vorticity Map – Split Type

$$\omega_z = \frac{\partial V}{\partial x} - \frac{\partial U}{\partial y}$$





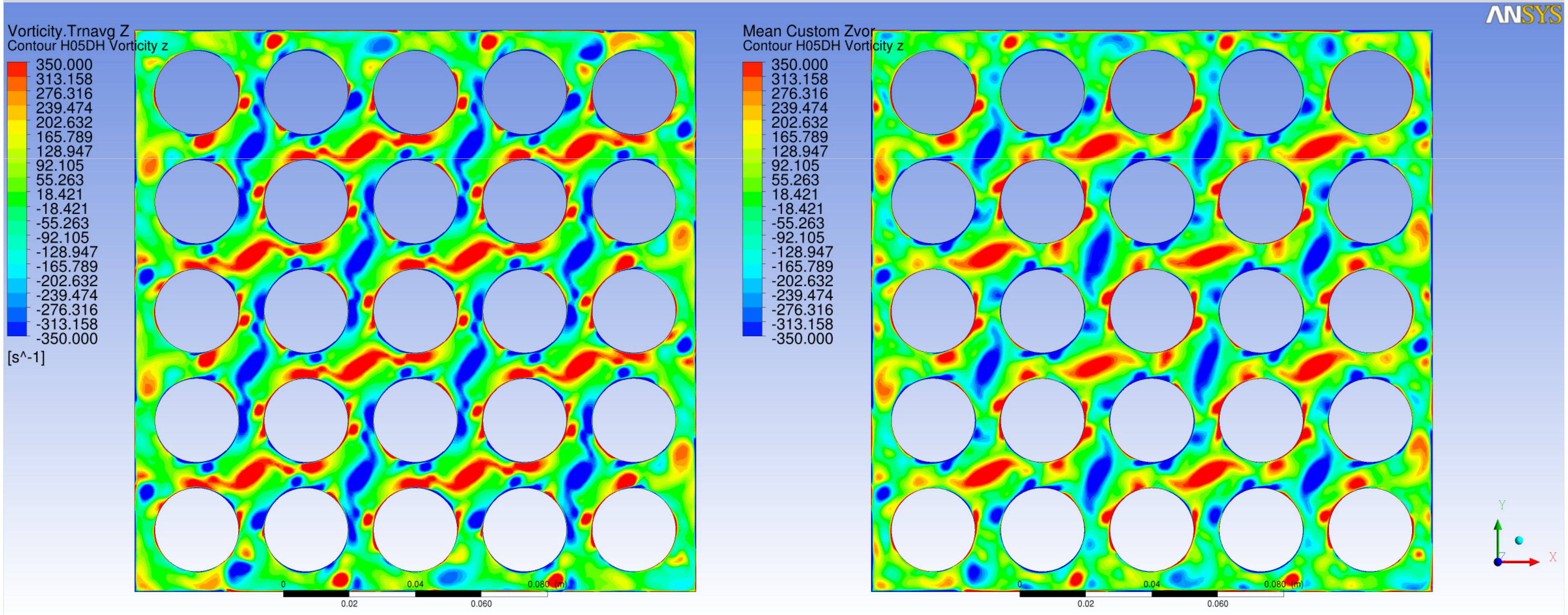
Z = 0.5 DH

Z = 4.0 DH

Mean Z Vorticity ω_z, $z=0.5\cdot D_H$

ANSYS CFX, SST-CC, Mesh_3

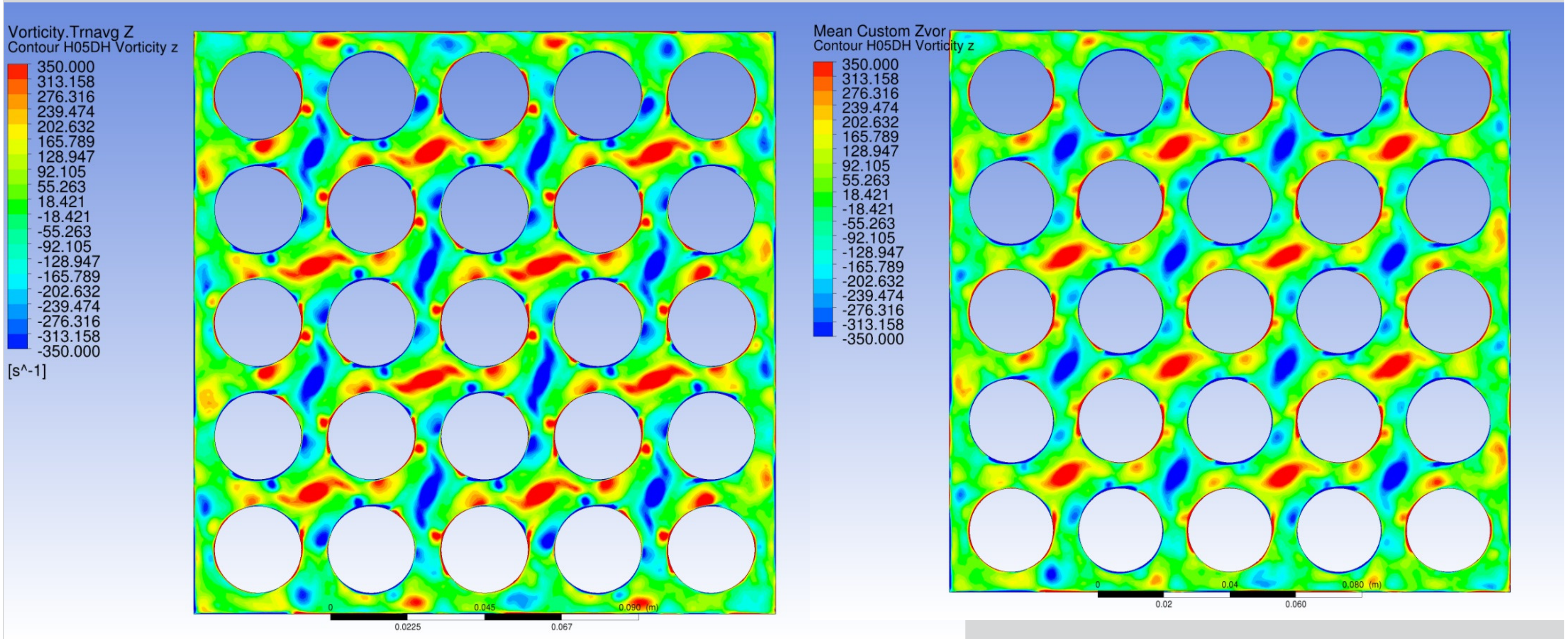
ANSYS Fluent, SST-CC, Mesh_3



Mean Z Vorticity ω_z, $z=0.5\cdot D_H$

ANSYS CFX, ZLES SAS-SST, Mesh_3

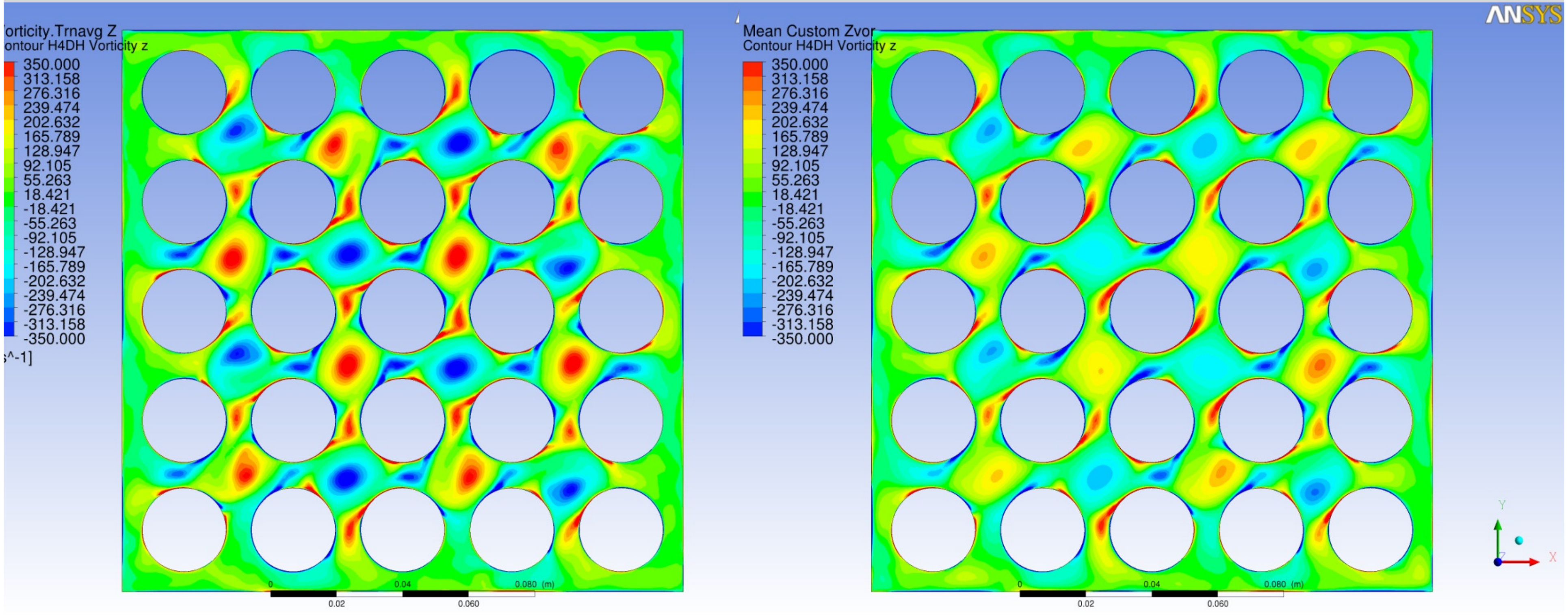
ANSYS Fluent, SAS-SST, Mesh_3



Mean Z Vorticity ω_z, $z=4.0 \cdot D_H$

ANSYS CFX, SST-CC, Mesh_3

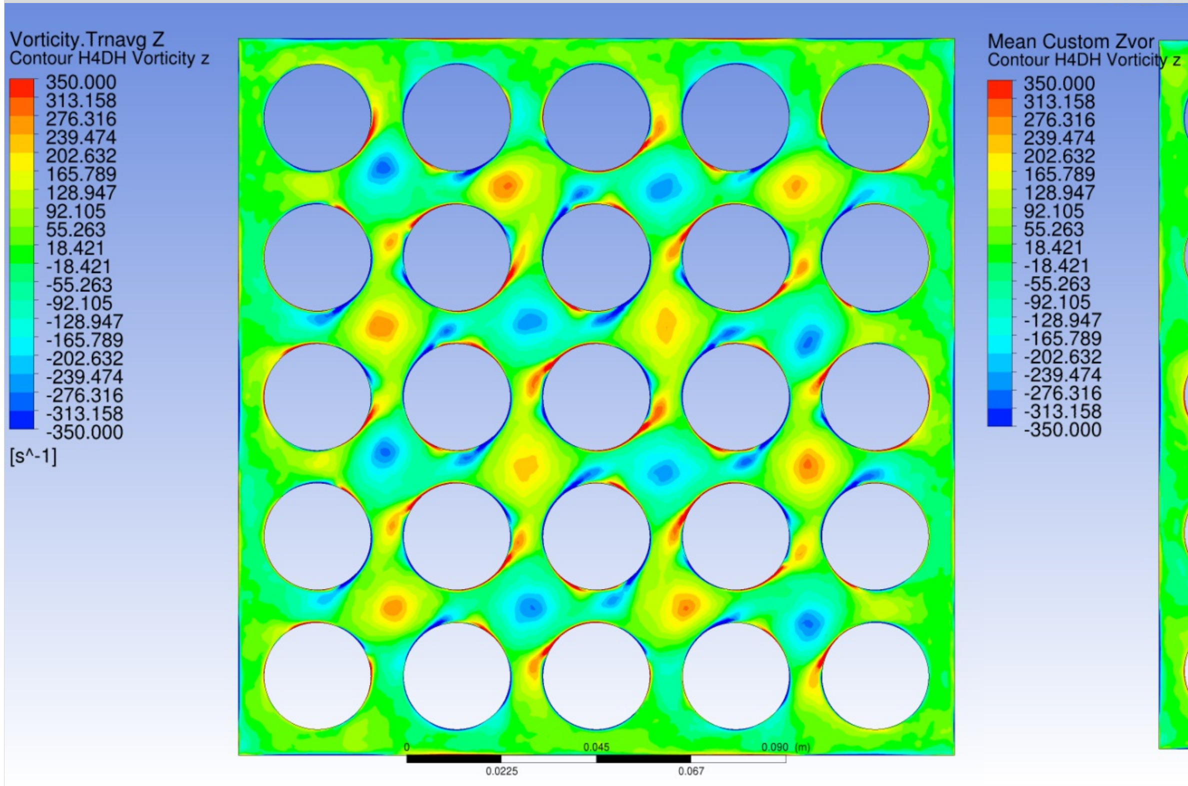
ANSYS Fluent, SST-CC, Mesh_3

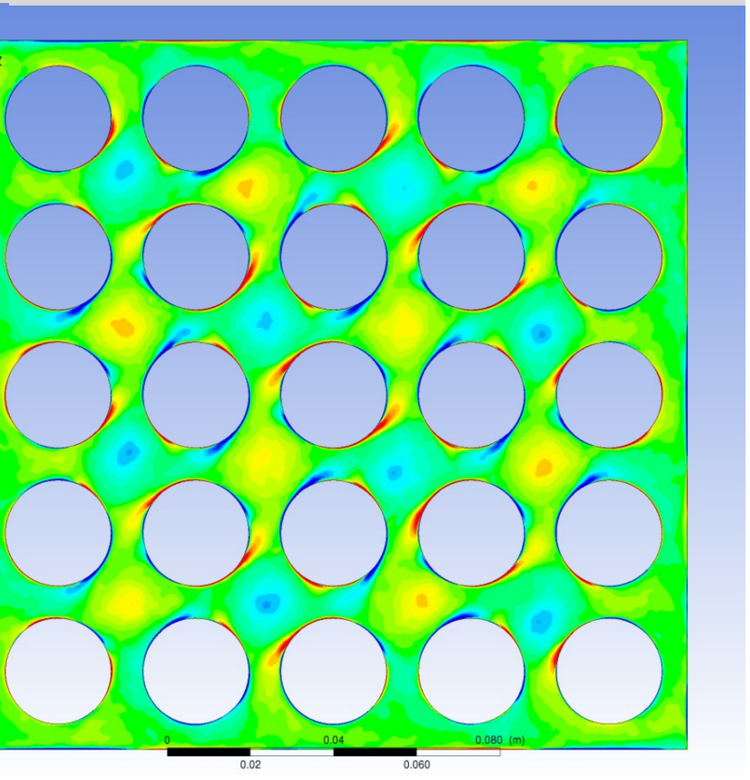


Mean Z Vorticity ω_z, $z=4.0\cdot D_H$

ANSYS CFX, ZLES SAS-SST, Mesh_3

ANSYS Fluent, SAS-SST, Mesh_3

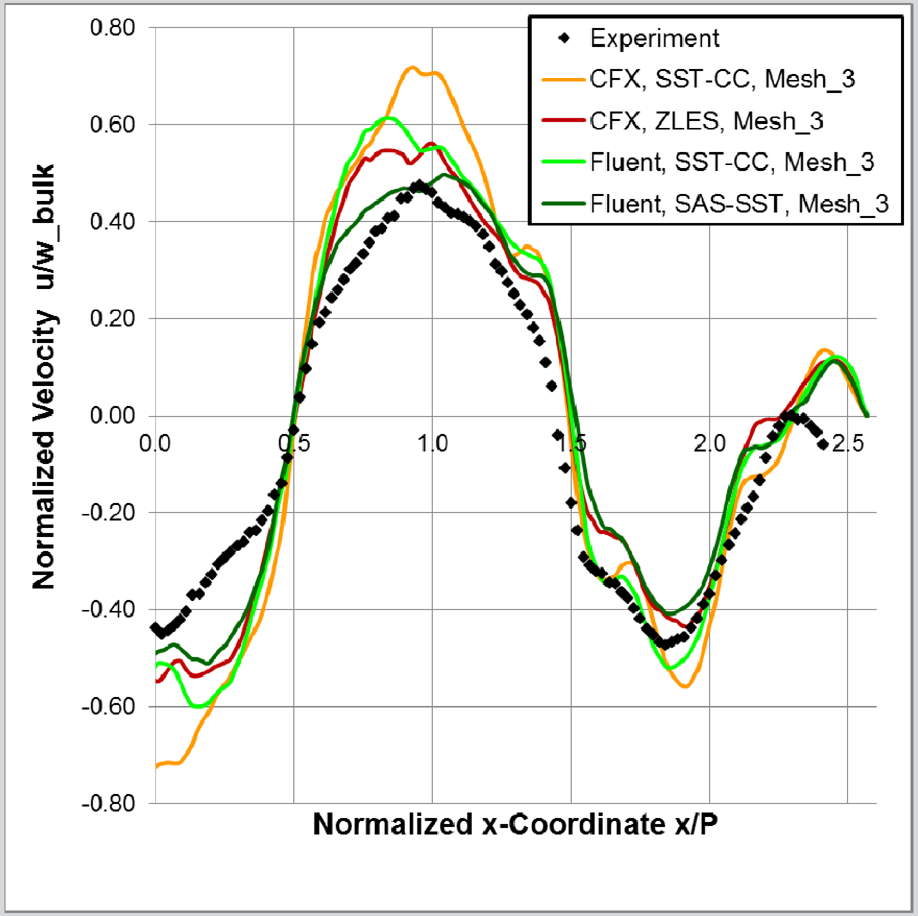


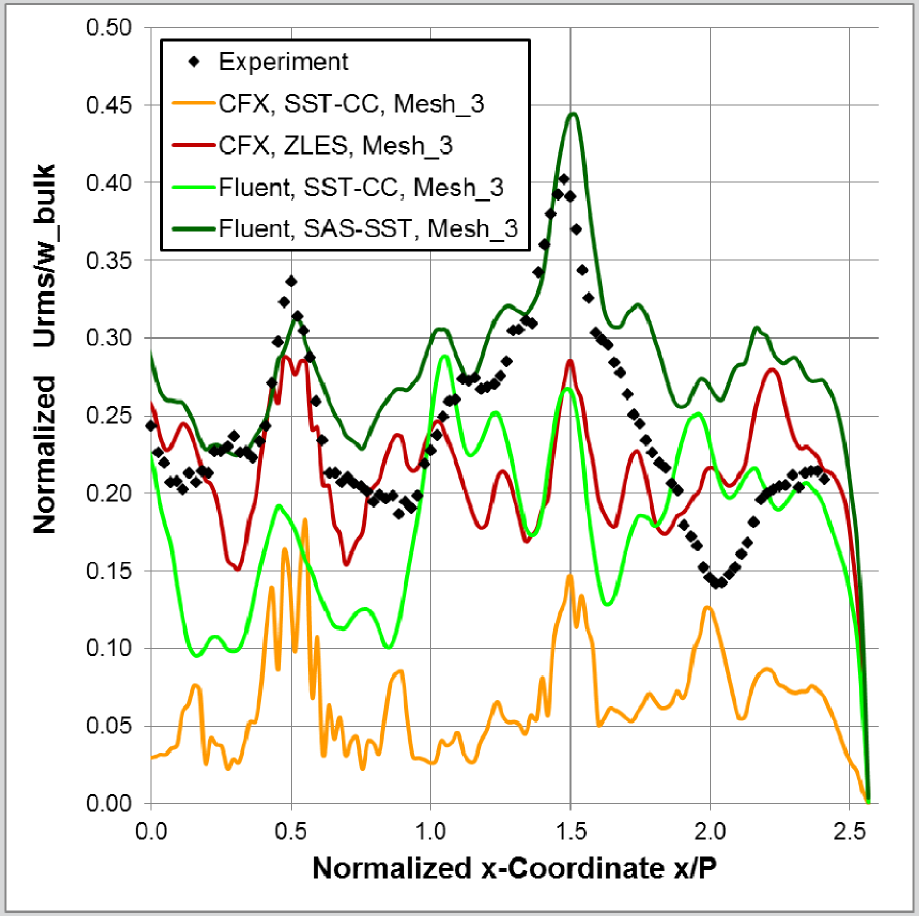


Comparison to KAERI LDV Data

SRS vs. SST-CC, Profile y1=0.5P @ z=0.5·D_H

Mean transversal velocity U and U_{RMS}

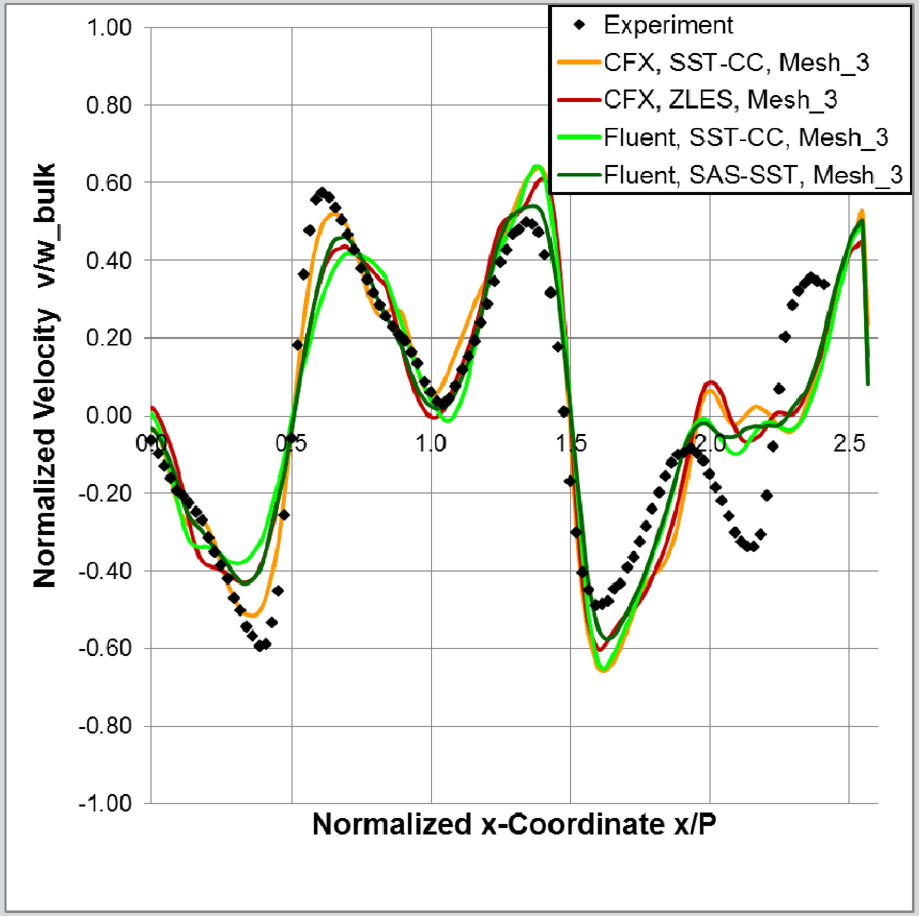


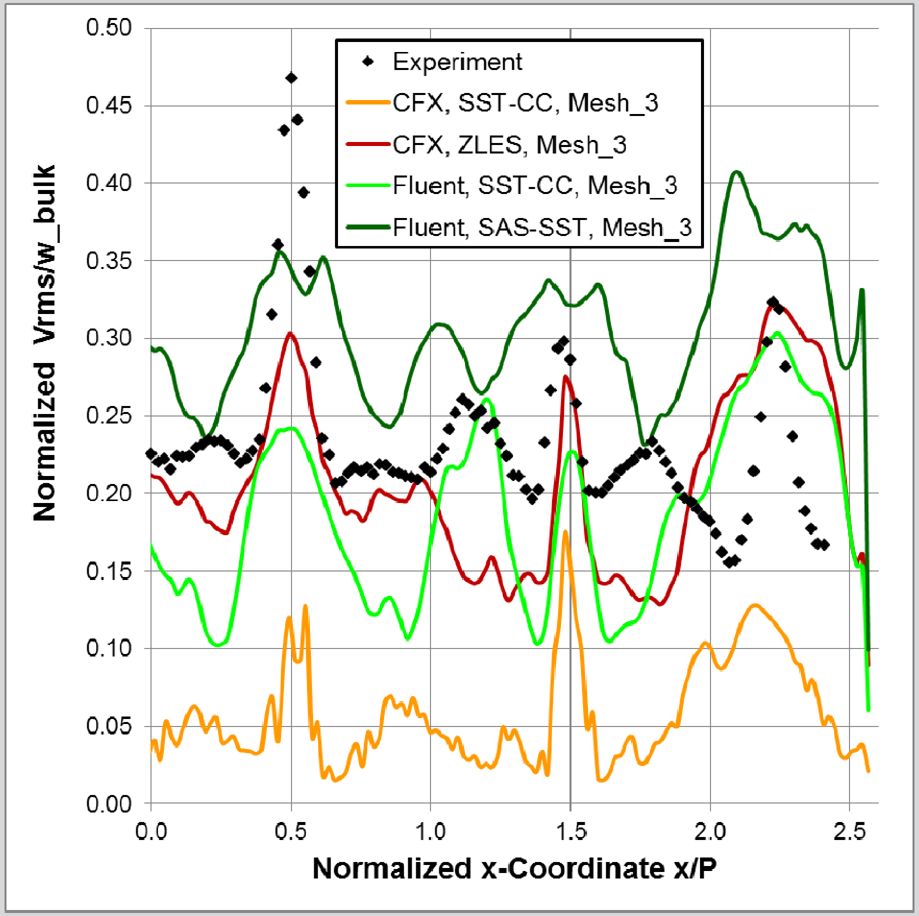


Comparison to KAERI LDV Data

SRS vs. SST-CC, Profile y1=0.5P @ $z=0.5 \cdot D_H$

Mean transversal velocity V and V_{RMS}

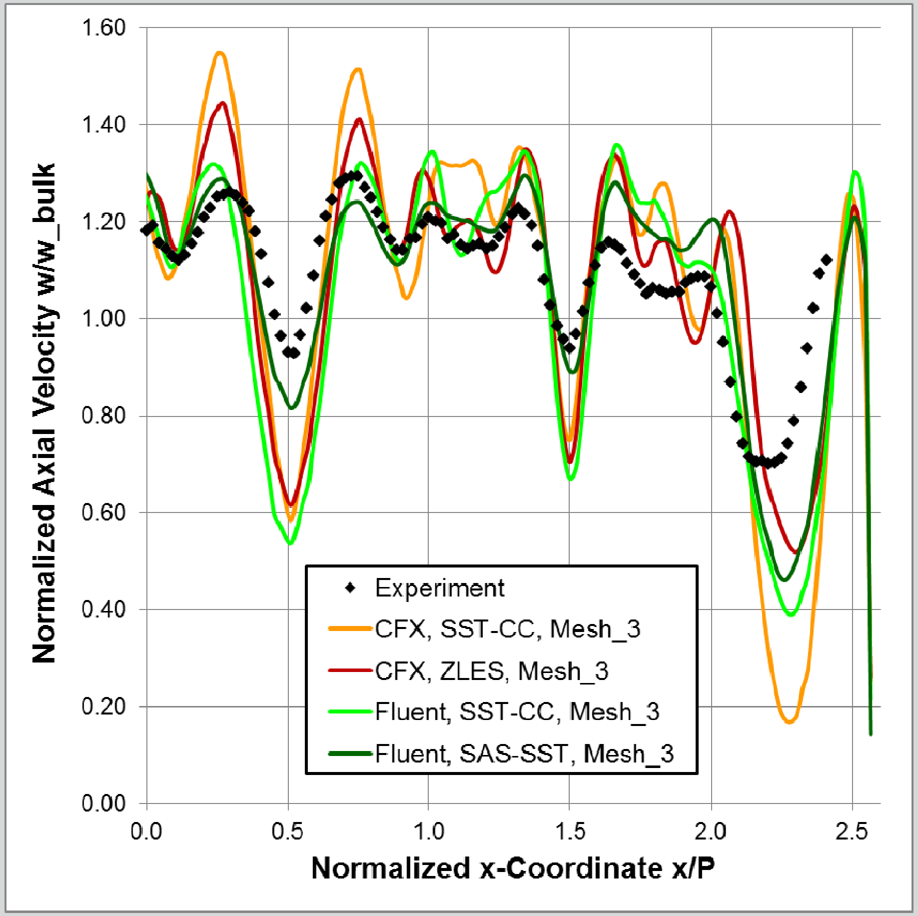


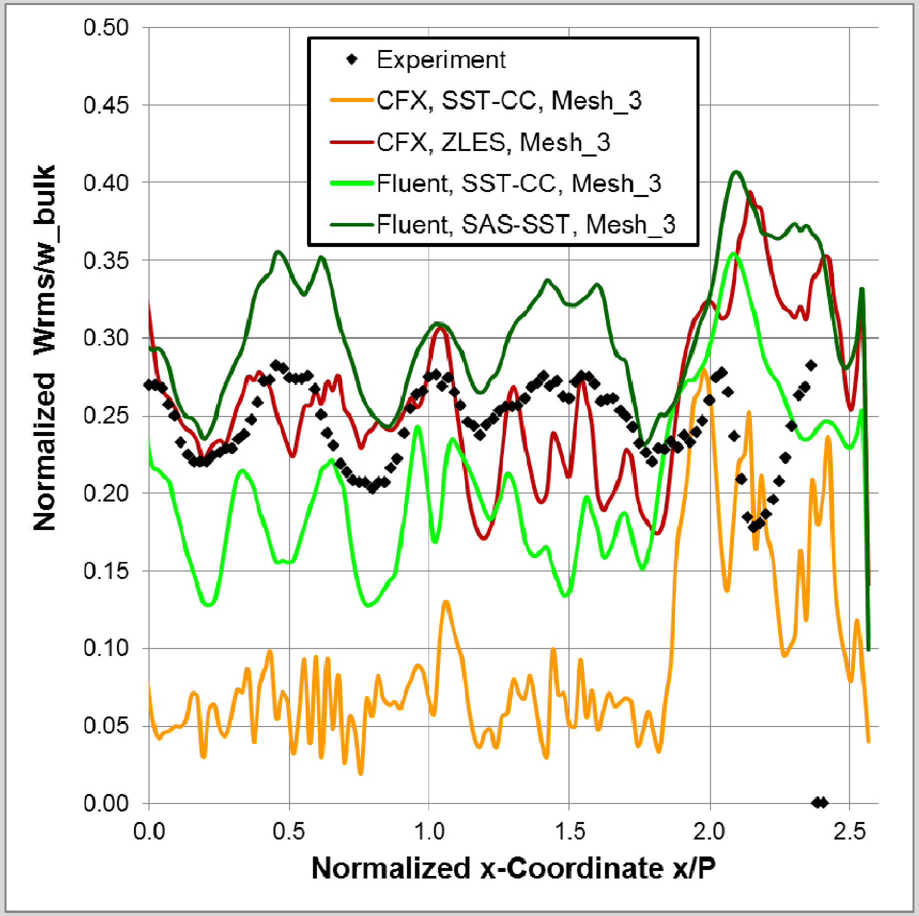


Comparison to KAERI LDV Data

SRS vs. SST-CC, Profile y1=0.5P @ $z=0.5 \cdot D_H$

Mean axial velocity W and W_{RMS}

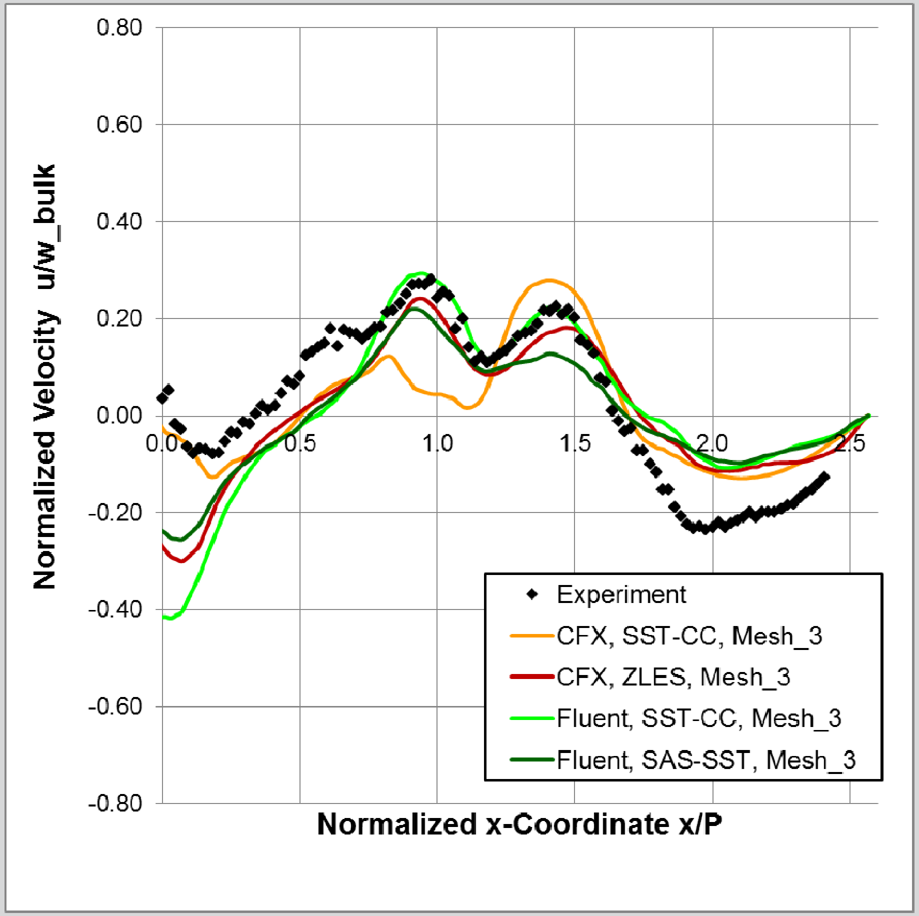


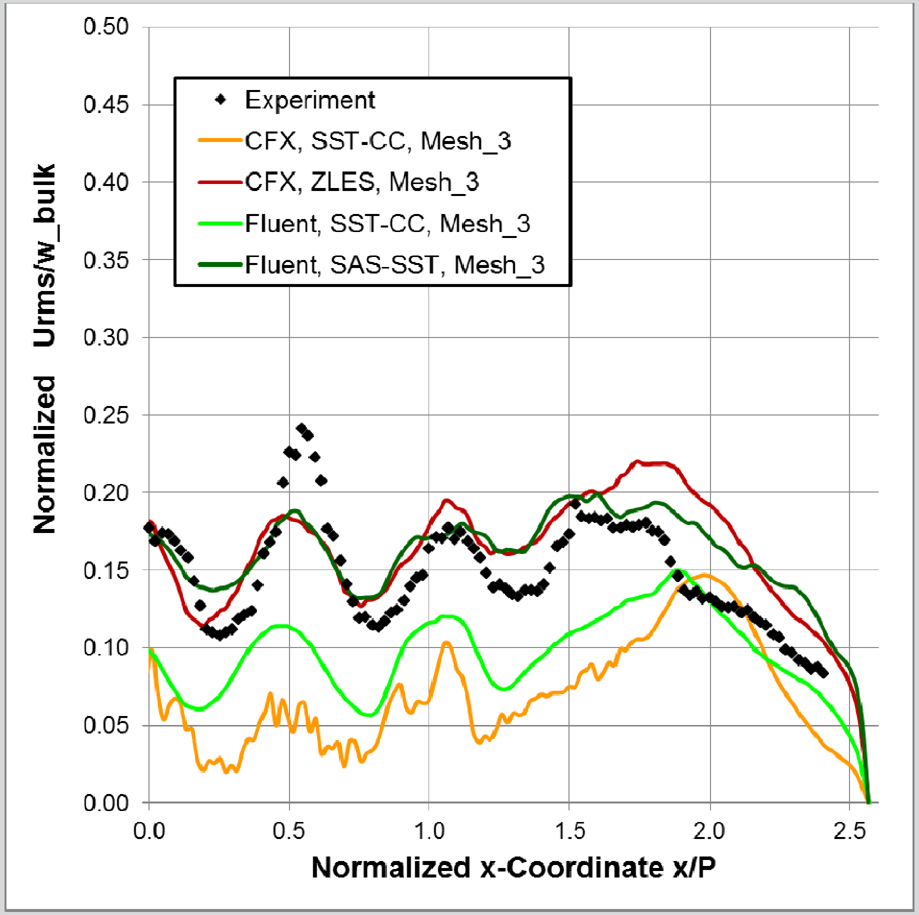


Comparison to KAERI LDV Data

SRS vs. SST-CC, Profile y1=0.5P @ z=4.0·D_H

Mean transversal velocity U and U_{RMS}

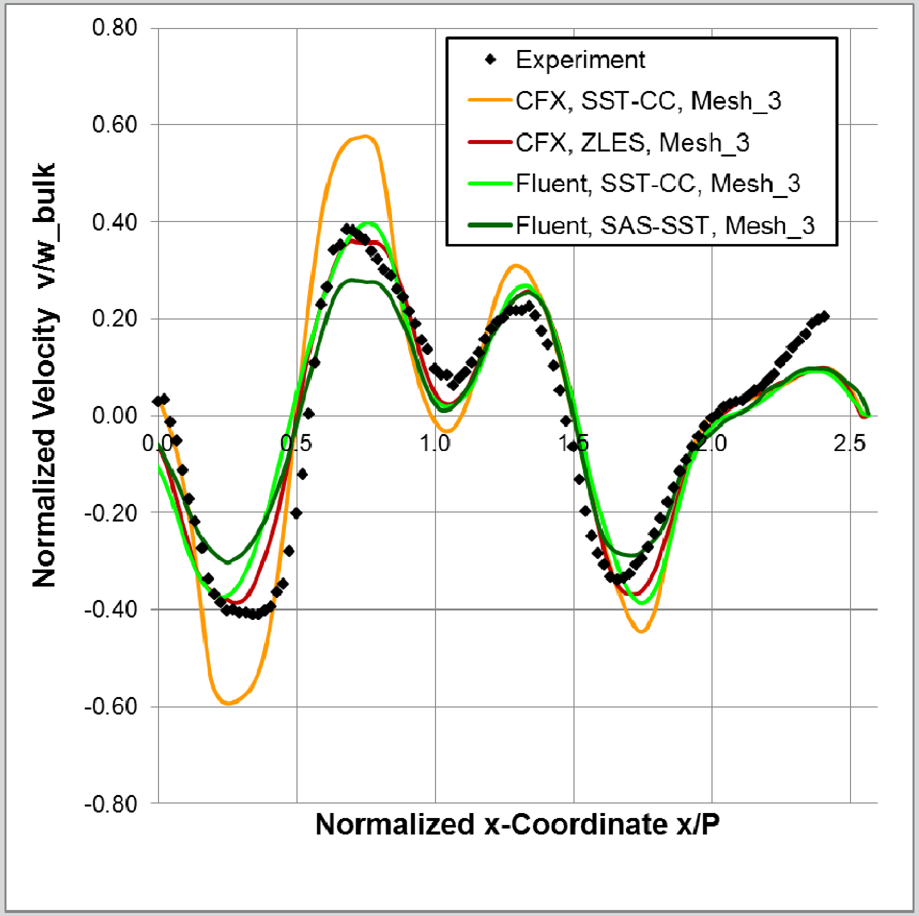


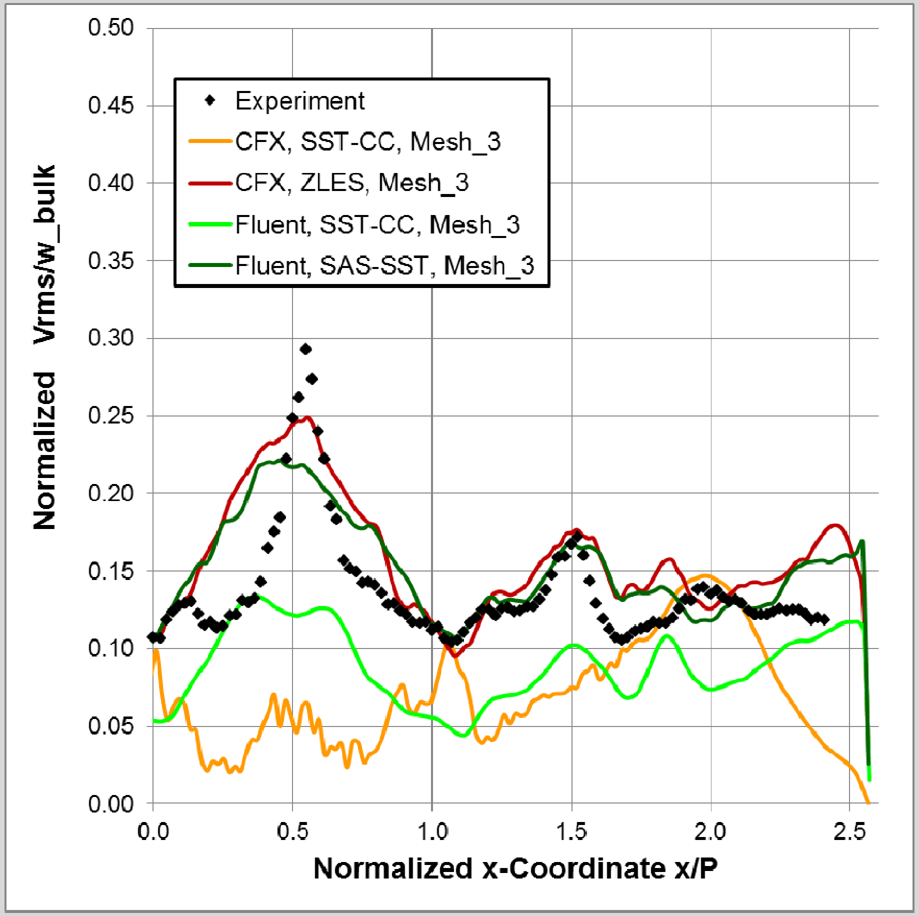


Comparison to KAERI LDV Data

SRS vs. SST-CC, Profile y1=0.5P @ z=4.0·D_H

Mean transversal velocity V and V_{RMS}

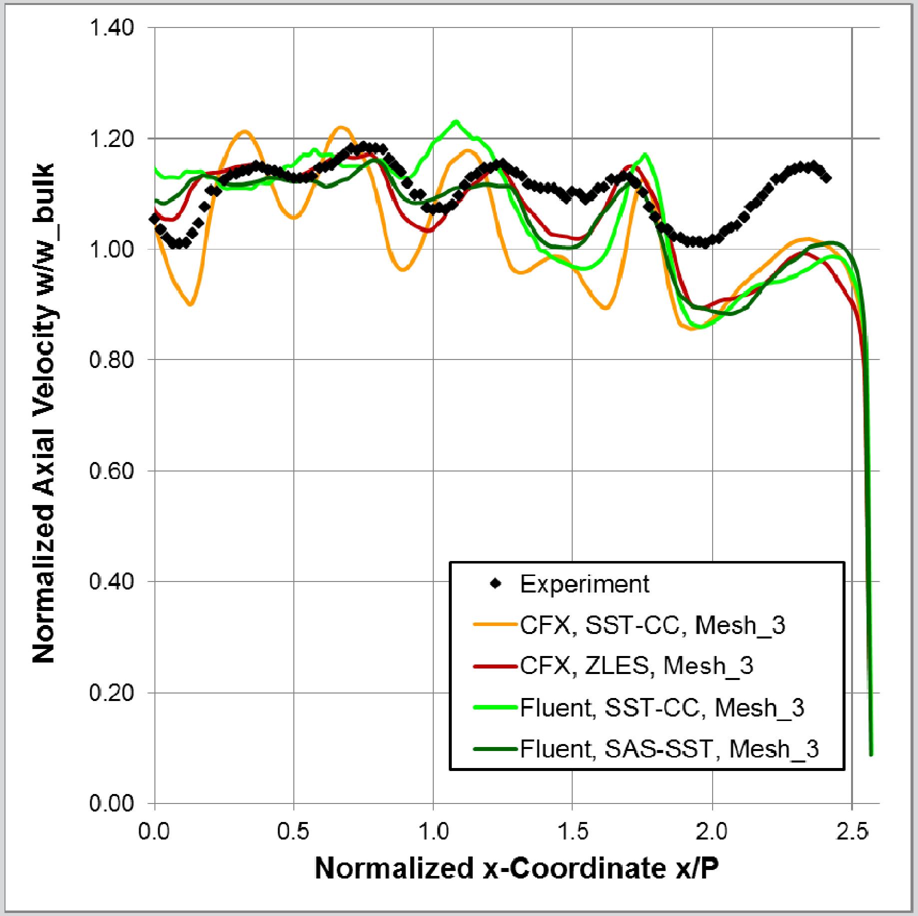


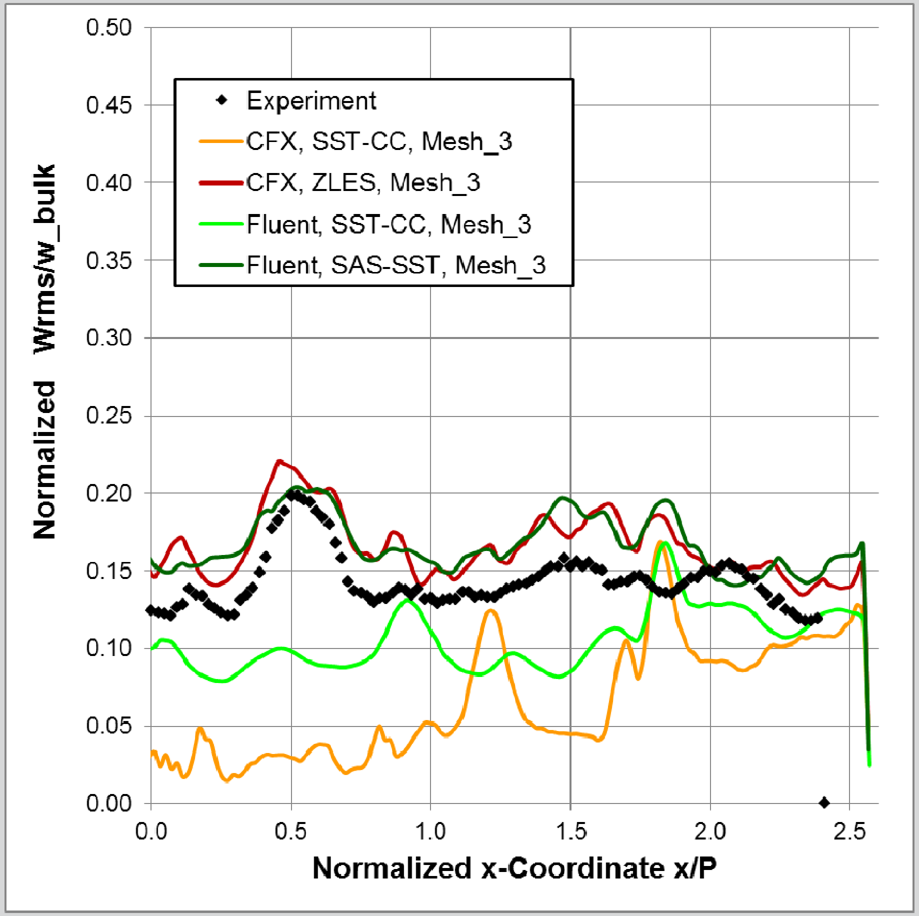


Comparison to KAERI LDV Data

SRS vs. SST-CC, Profile y1=0.5P @ z=4.0·D_H

Mean axial velocity W and W_{RMS}





Conclusion from Comparison to KAERI LDV Data

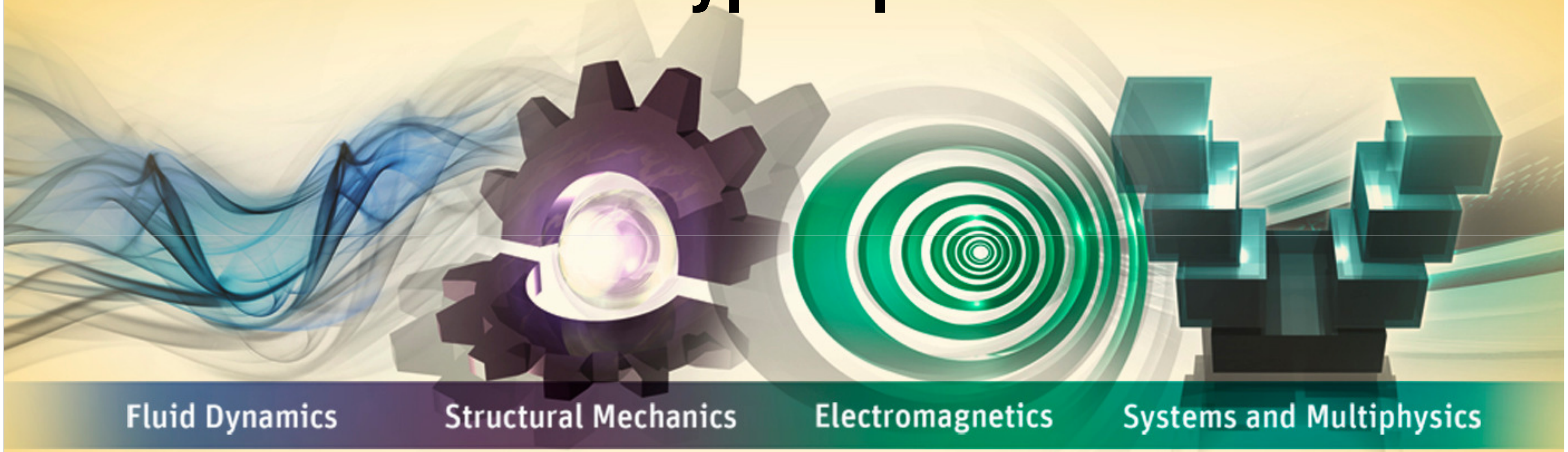
- URANS solutions in both ANSYS CFD solvers tend to overpredict the mean axial velocities in the vortices spawned from the spacer guide vanes. Lateral u and v velocity components are predicted in better agreement to data.
- ZLES and SAS-SST approaches are better agreement with KAERI LDV measurements.
- While some overprediction of axial velocity can still be observed at $z=0.5 \cdot D_H$ and $z=1.0 \cdot D_H$, the agreement for the cross sections further downstream is quite satisfactory.
- For ZLES and SAS-SST the results for velocity RMS values are in good agreement to measured velocity RMS values, which underlines the fact that the SRS simulations resolved the major part of turbulent fluctuations in the flow under investigation.

Conclusion from Comparison to KAERI LDV Data (cont.)

- Finally the ZLES and SAS-SST approaches are able to predict mean velocity profiles in good agreement with KAERI LDV measurements.
- While some overprediction of axial velocity can still be observed at $z=0.5 \cdot D_H$ and $z=1.0 \cdot D_H$, the agreement for the cross sections further downstream is quite satisfactory.
- As for the URANS simulations the agreement of predicted x- and y-velocity components is in better agreement to data for the SRS simulations as well.
- Results for velocity RMS values are in good agreement to measured velocity RMS values, which underlines the fact that the SRS simulations resolved the major part of turbulent fluctuations in the flow under investigation.

Realize Your Product Promise™

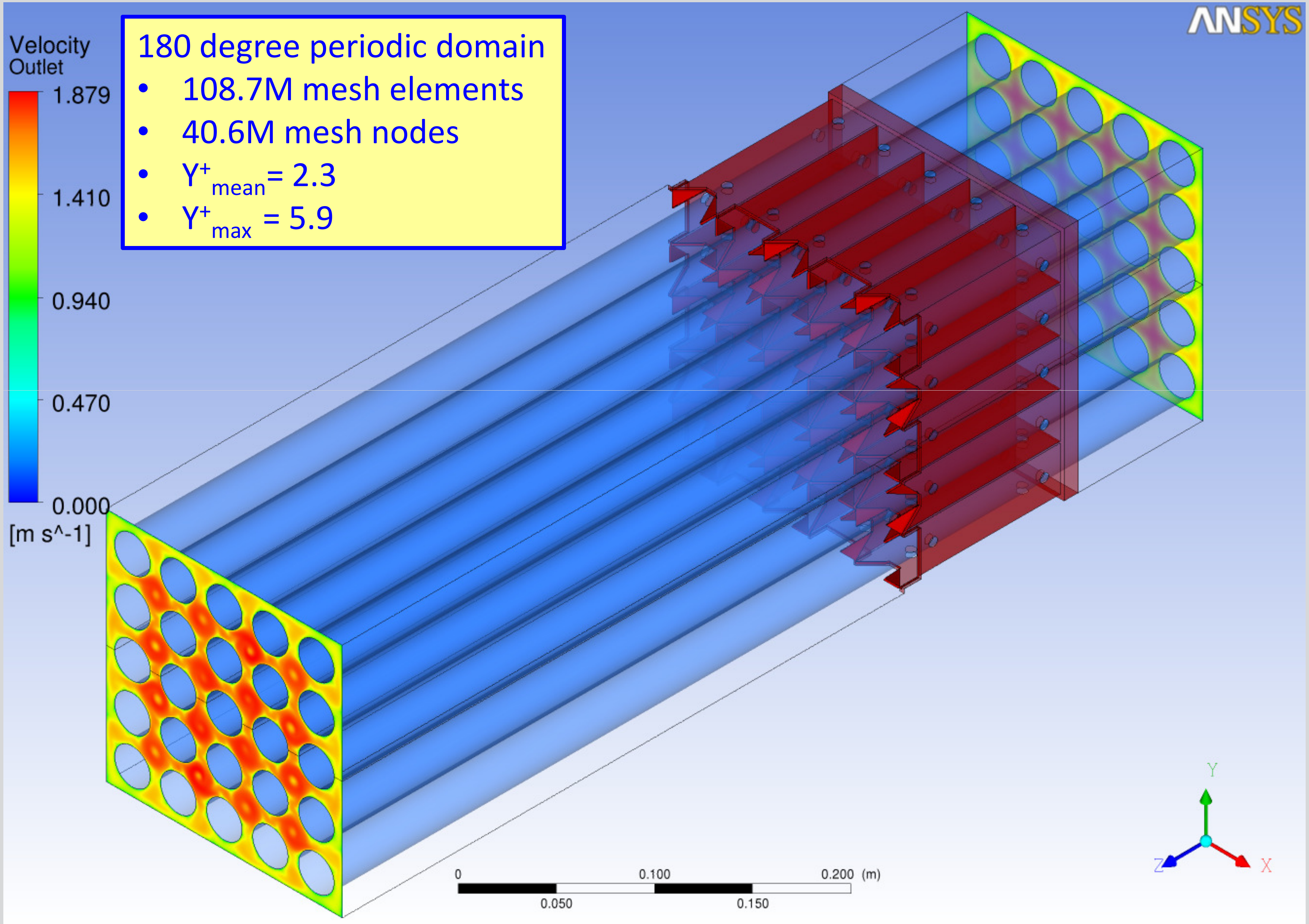
Step 3b: Final MATiS-H Benchmark Simulations – Swirl Type Spacer Grid –



Geometry of Swirl Type Spacer Grid

180 degree periodic domain

- 108.7M mesh elements
- 40.6M mesh nodes
- $Y^{+}_{mean} = 2.3$
- $Y^{+}_{max} = 5.9$



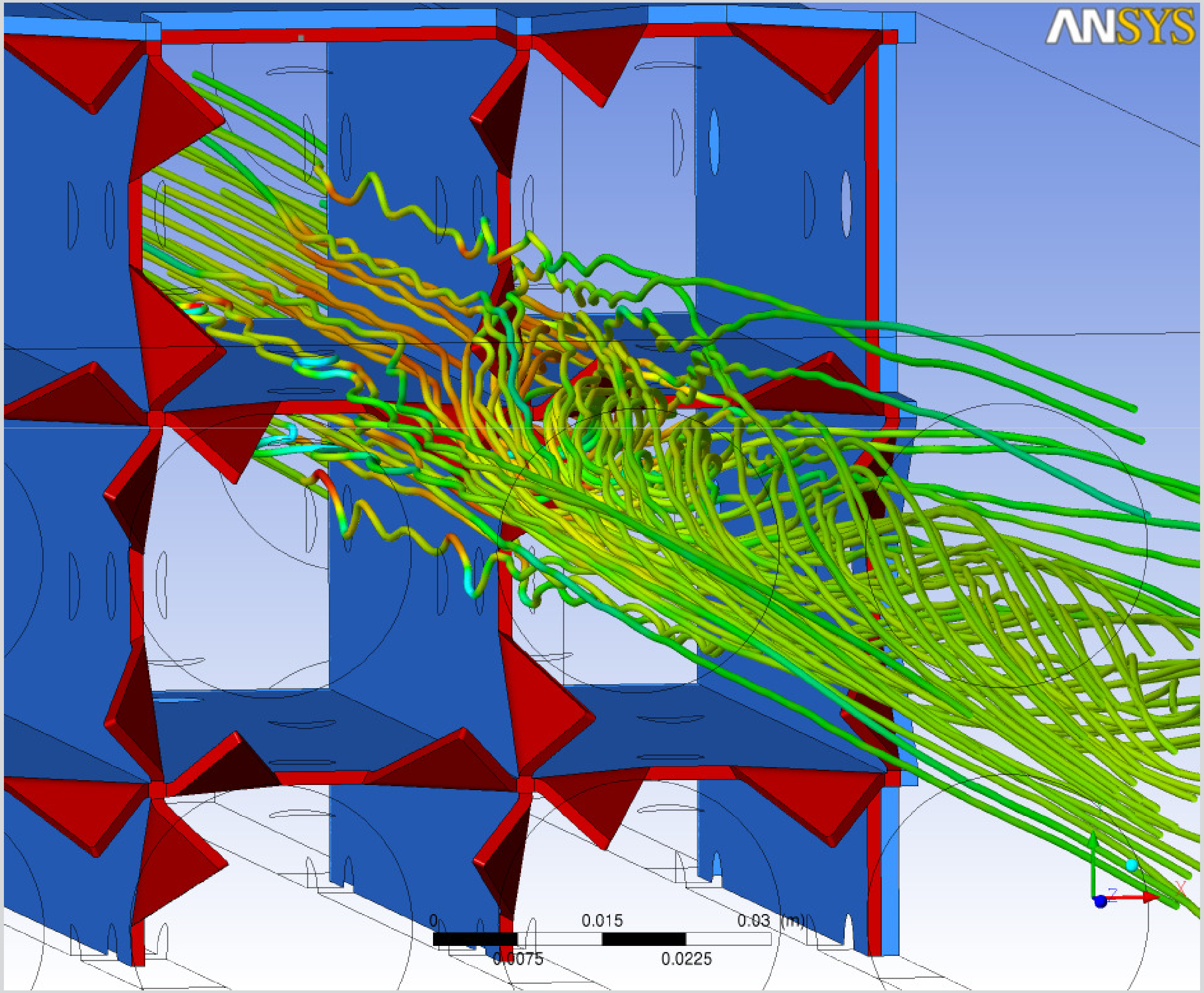
Resulting CFD Test Matrix for Swirl Type Spacer Grid

ANSYS CFD solver	Turbulence model	Mesh 1	Mesh 2
ANSYS CFX 14.0	BSL RSM	X	X
	ZLES SAS-SST		X
ANSYS Fluent 14.0	SST-CC	X	X
	SAS-SST		X

CFD simulation methodology:

- **Start with steady-state simulation on coarsest mesh**
- **Initialization of the transient flow simulation from the steady-state solution.**
  - **Time for transient flow development (T_1=0.25s).**
  - **Time for statistical time-averaging of results (T_2=0.6-1.25s).**
- **For transient flow simulations on refined meshes the CFD simulation was initialized with the final result from the transient CFD simulation on the previous coarser grid level.**
- **ANSYS Fluent, SAS-SST simulation initialized with URANS SST-CC result on Mesh2. ANSYS CFX, ZLES SAS-SST simulation initialized with previously obtained BSLRSM result on Mesh2.**

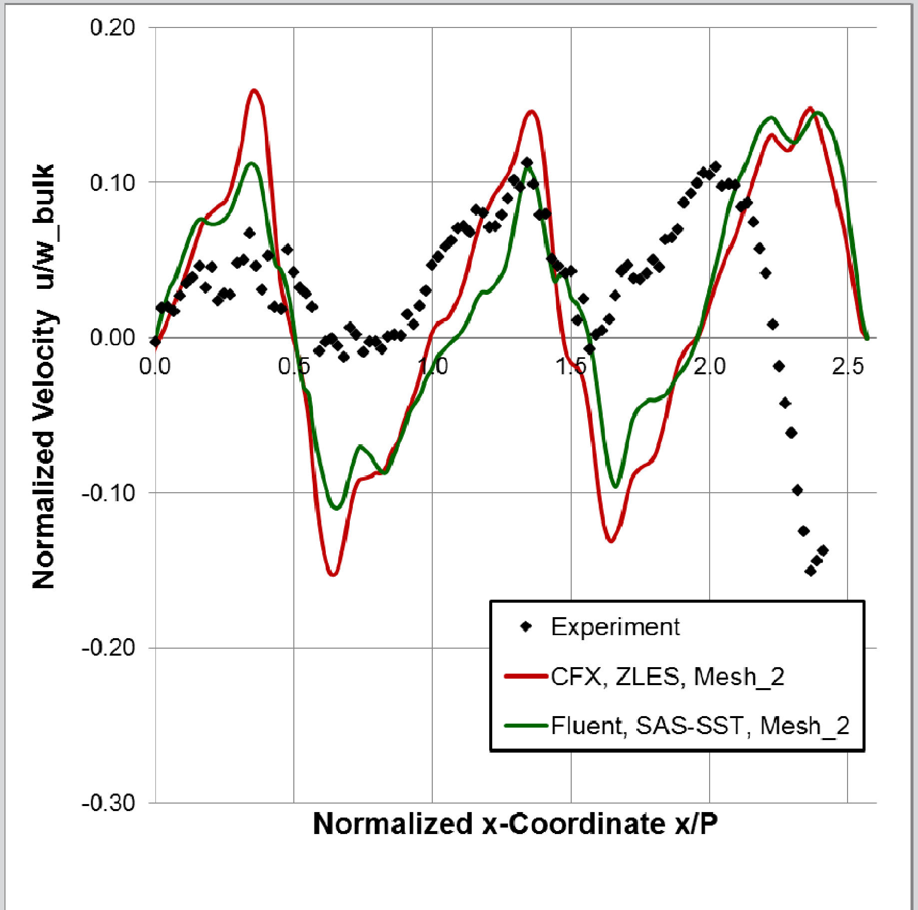
Streamlines for Swirl Type Spacer Grid

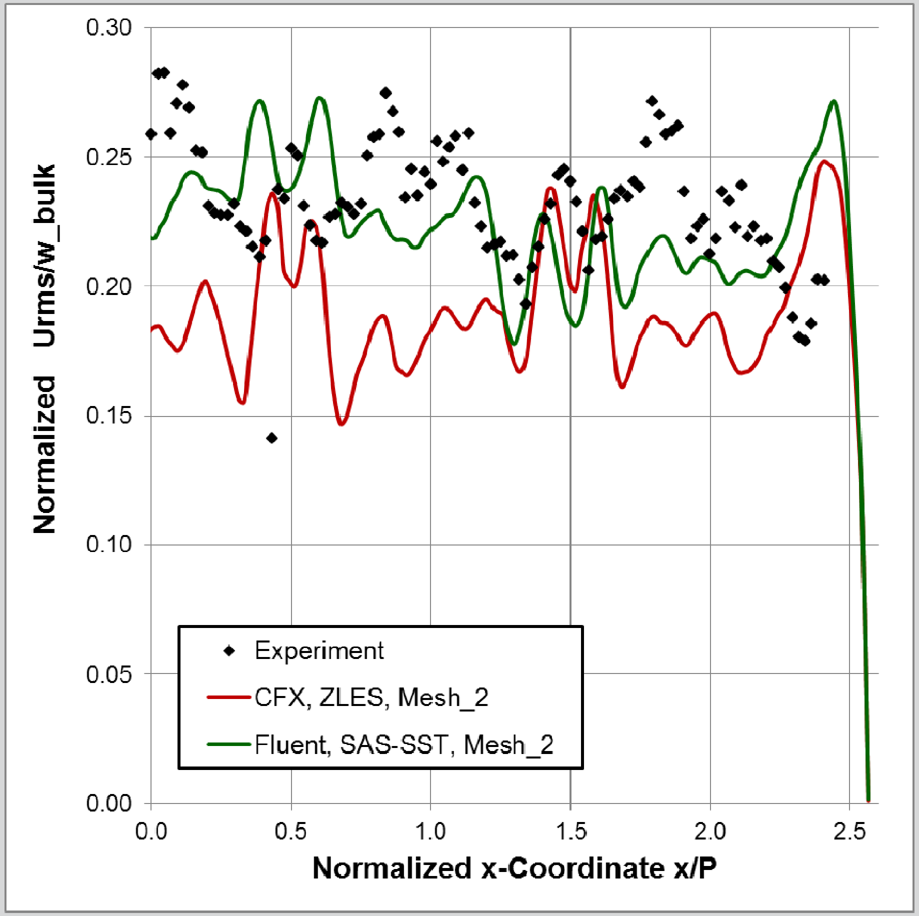


Comparison to KAERI LDV Data

SRS results, Profile y1=0.5P @ $z=0.5 \cdot D_H$

Mean transversal velocity U and U_{RMS}

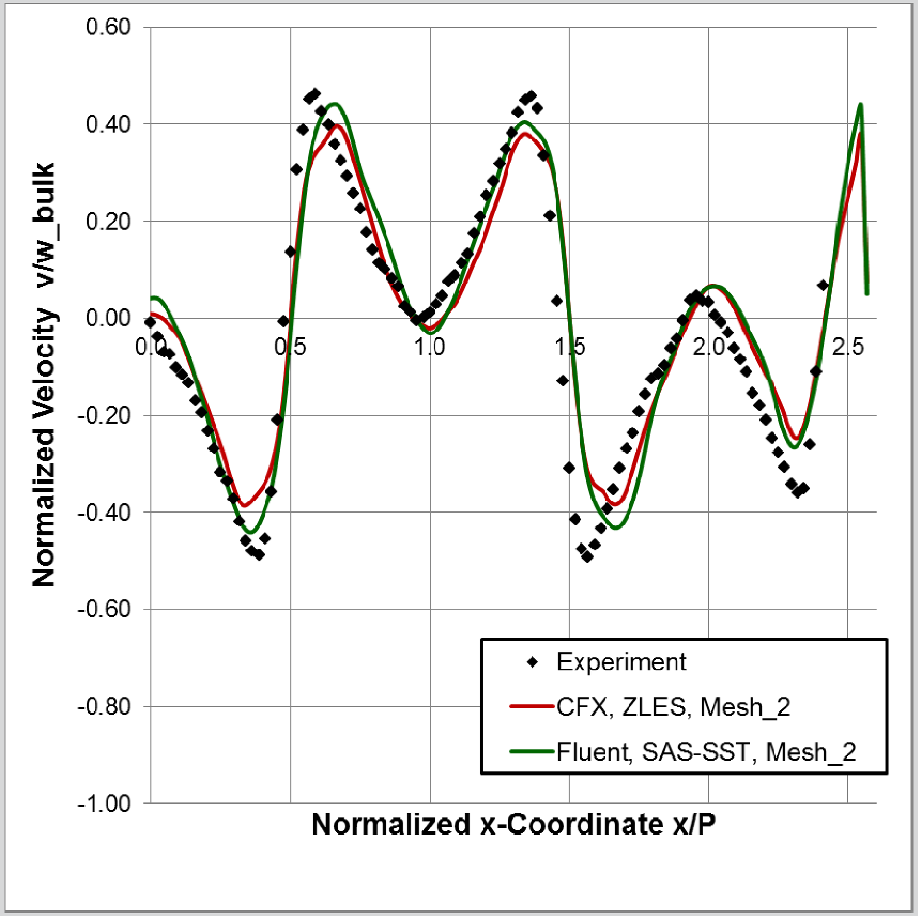


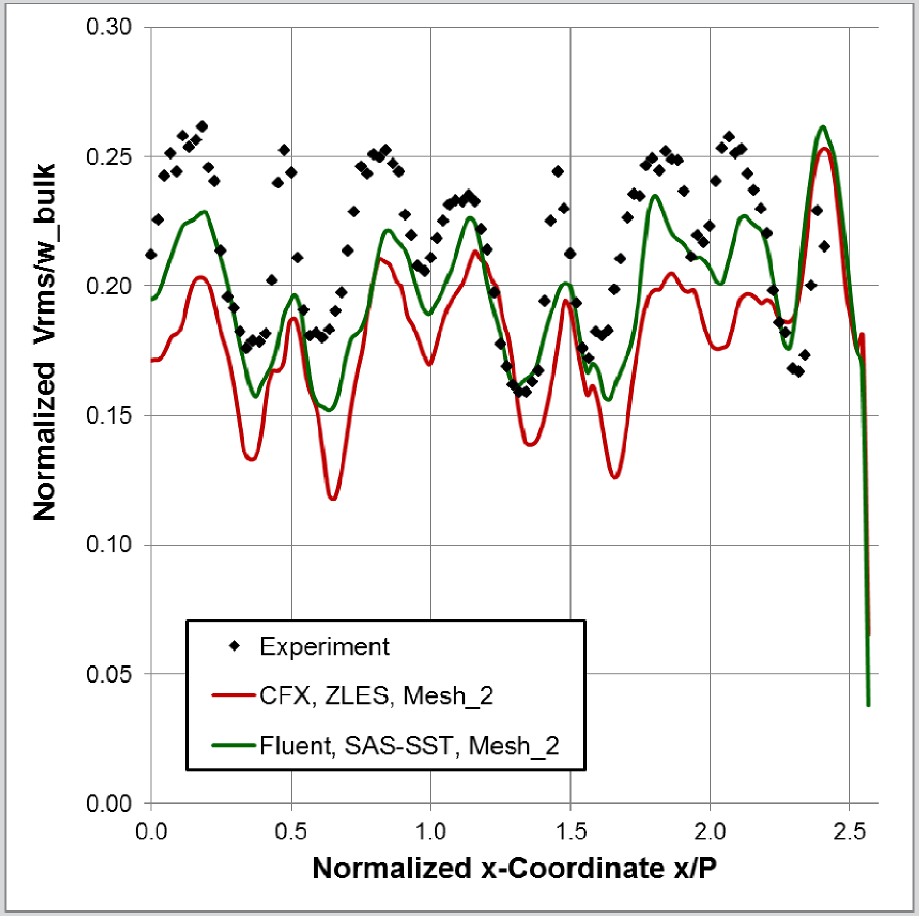


Comparison to KAERI LDV Data

SRS results, Profile y1=0.5P @ $z=0.5 \cdot D_H$

Mean transversal velocity V and V_{RMS}

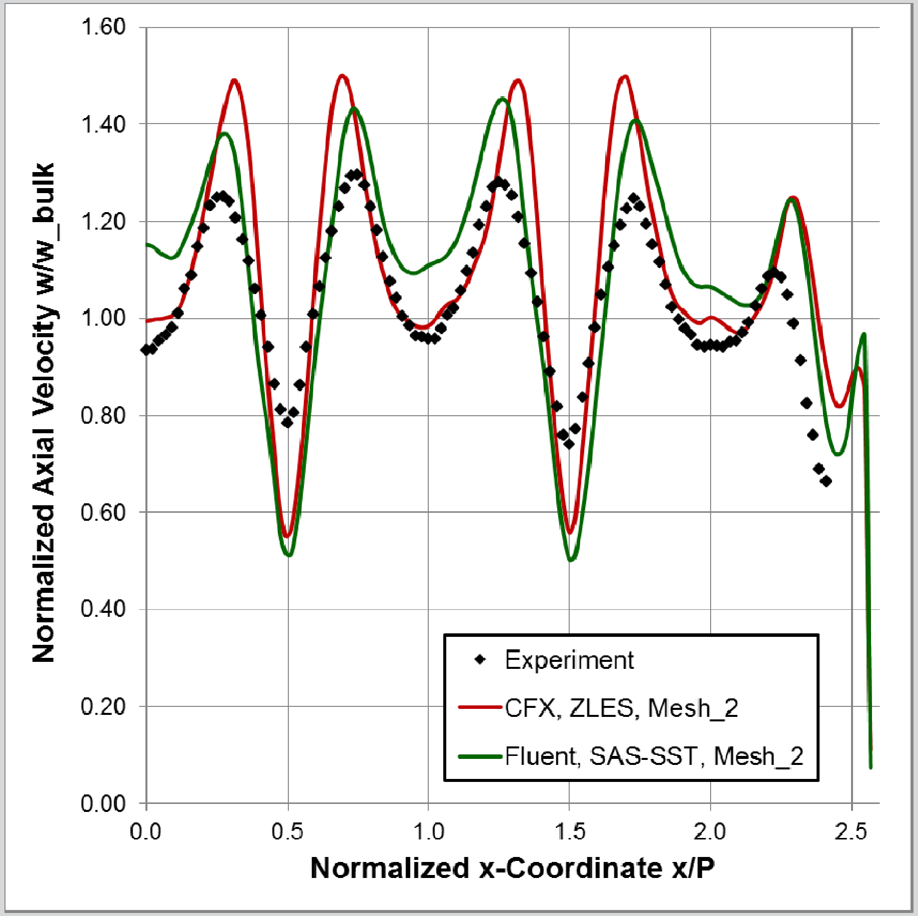


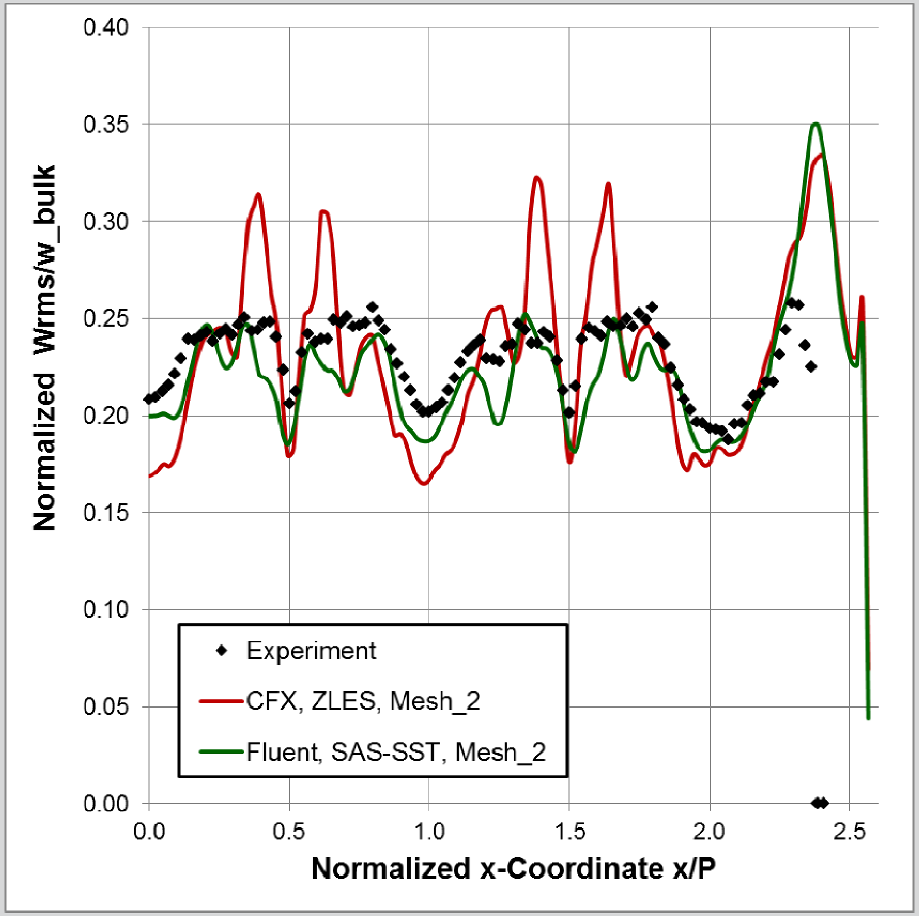


Comparison to KAERI LDV Data

SRS results, Profile y1=0.5P @ $z=0.5 \cdot D_H$

Mean axial velocity W and W_{RMS}

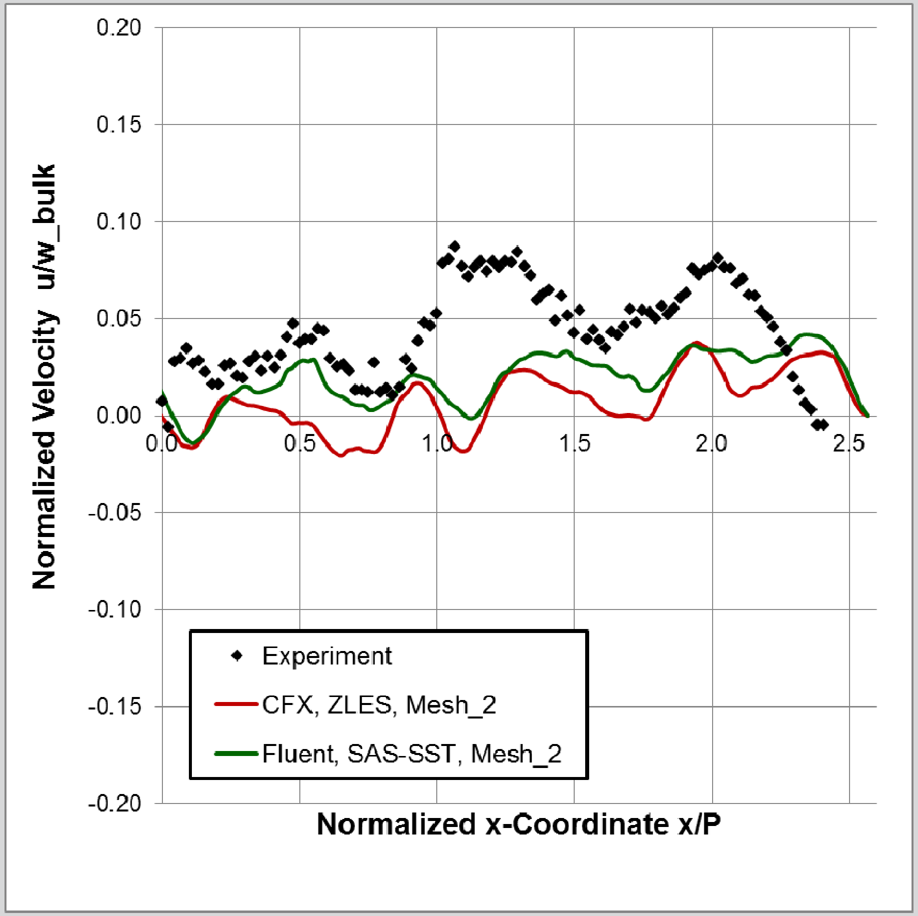


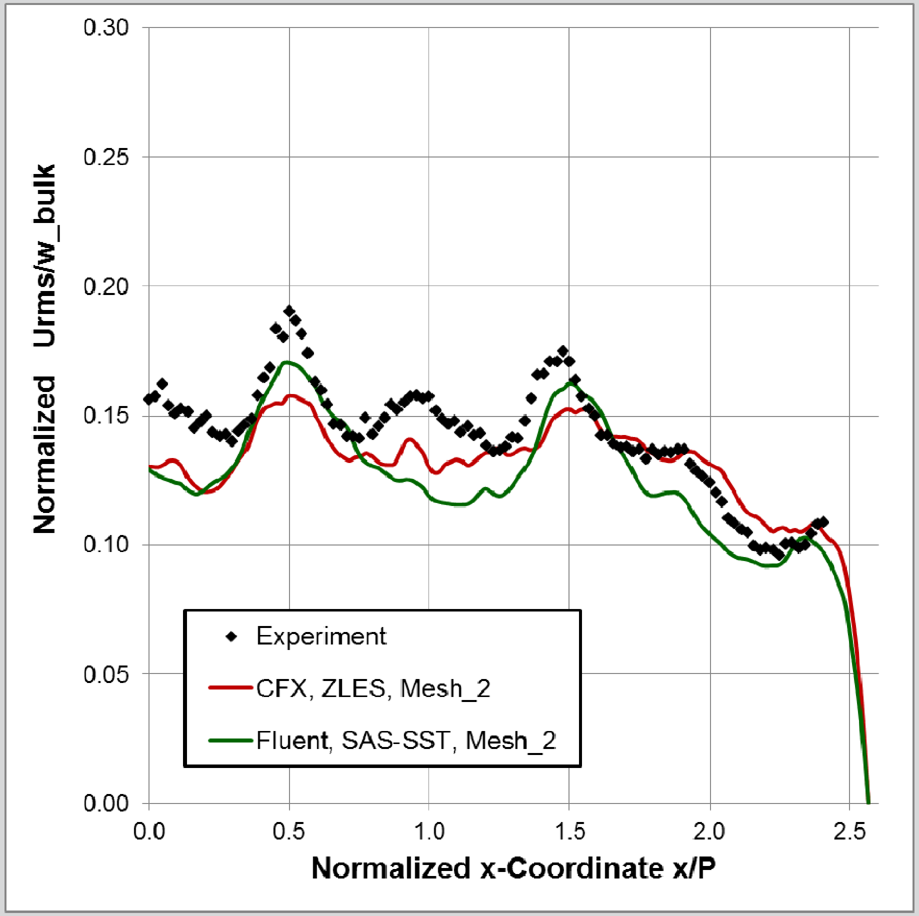


Comparison to KAERI LDV Data

SRS results, Profile y1=0.5P @ z=4.0·D_H

Mean transversal velocity U and U_{RMS}



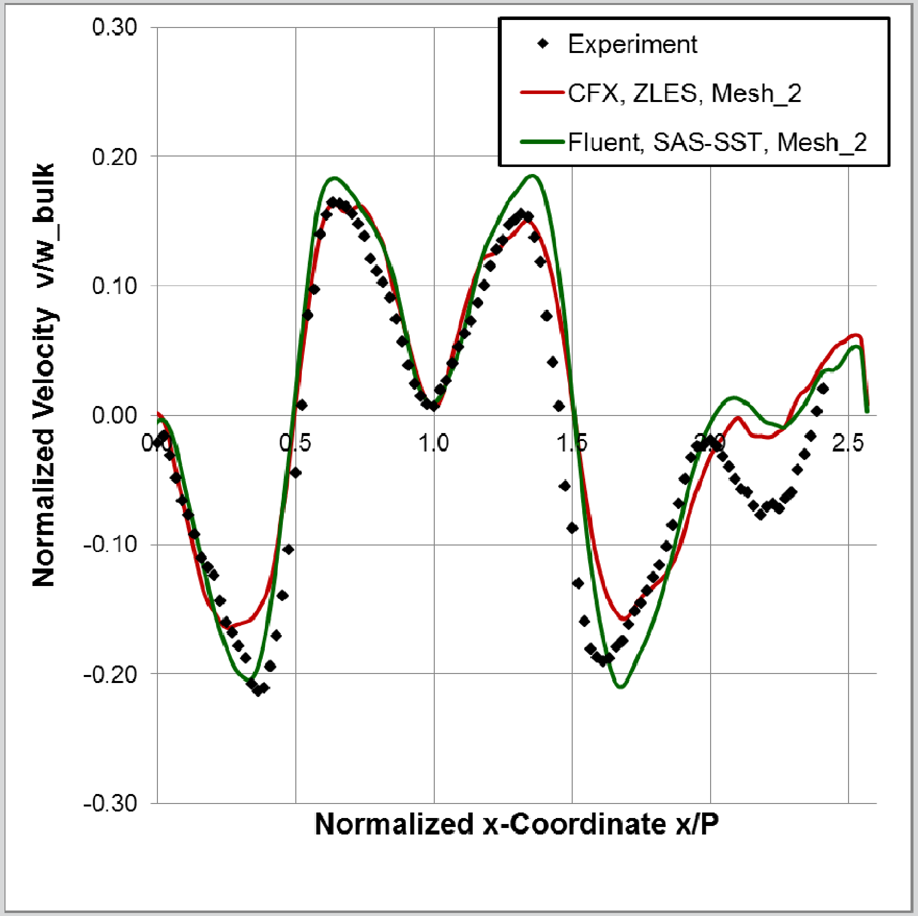


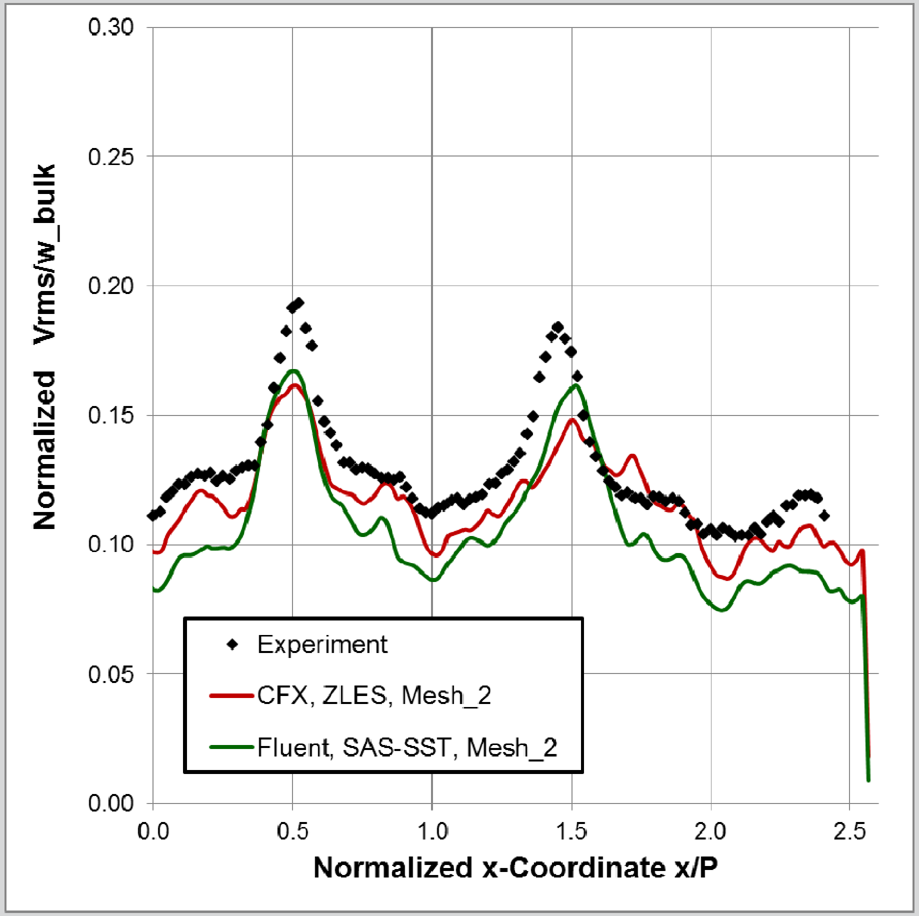
  August 9, 2013

Comparison to KAERI LDV Data

SRS results, Profile y1=0.5P @ z=4.0·D_H

Mean transversal velocity V and V_{RMS}

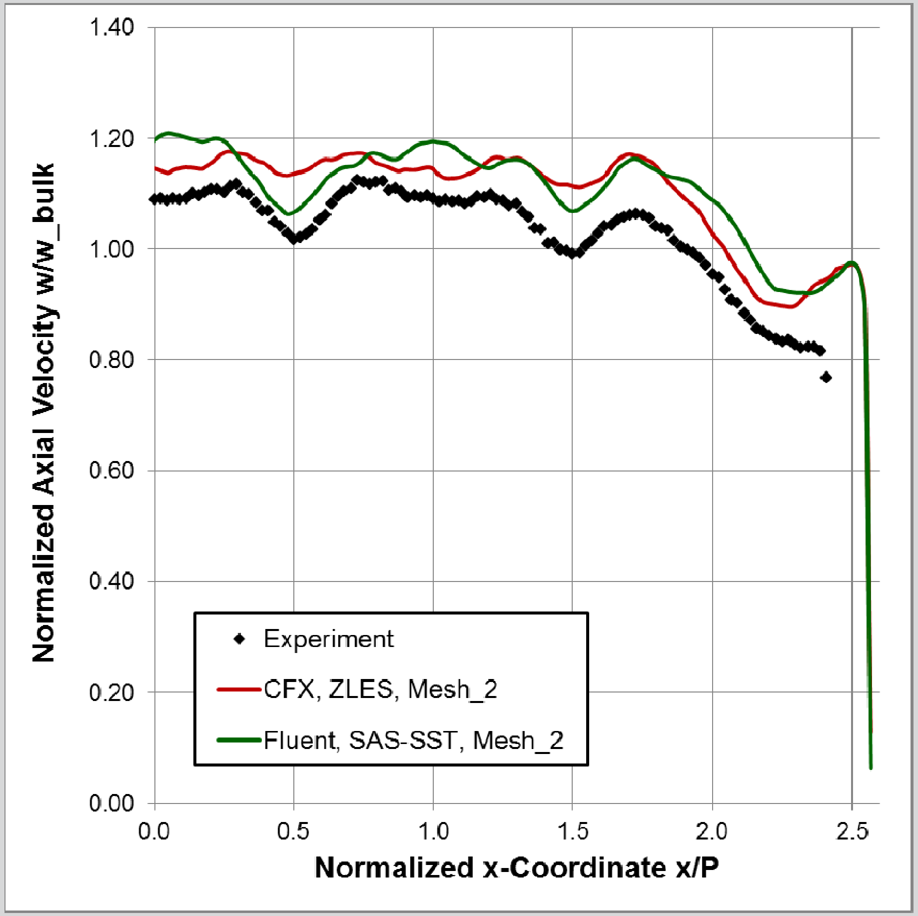


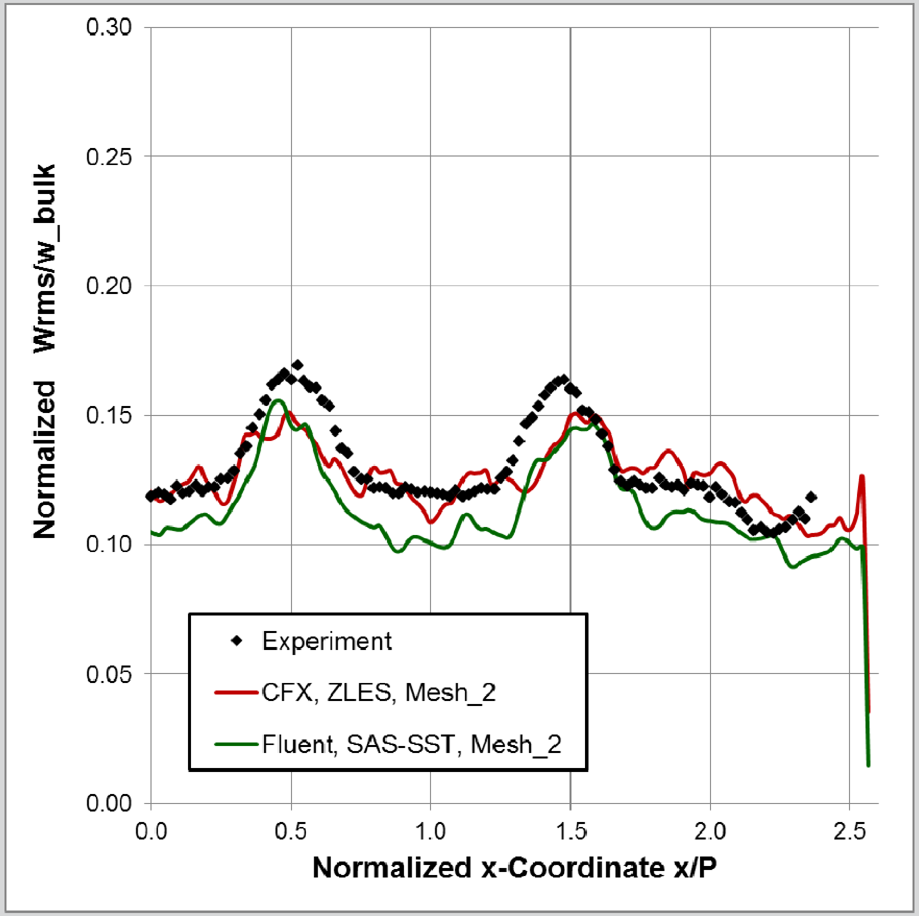


Comparison to KAERI LDV Data

SRS results, Profile y1=0.5P @ $z=4.0 \cdot D_H$

Mean axial velocity W and W_{RMS}



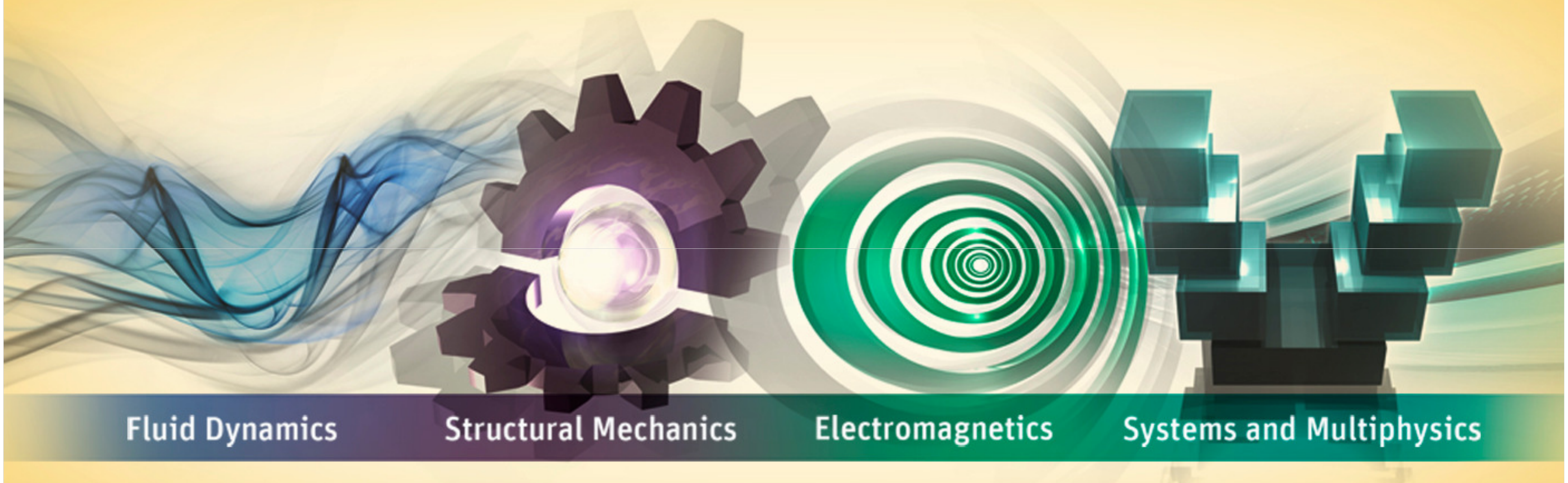


Conclusion from Comparison to KAERI LDV Data

- Conclusions drawn from the investigation of the split type spacer can be applied to the swirl type spacer grid geometry as well.
- ZLES and SAS-SST simulation results are in general good agreement with the KAERI LDV data.
- Mean y- and z-velocity components (v and w) are in very good agreement to data for all axial distances to the swirl type spacer, while the larger differences are observable for the mean x-velocity component (u).
- RMS velocities from SRS simulations show for all axial distances to the spacer the correct magnitude. Some scatter can be observed in very close distance to the guide vane tips of the spacer, while for larger axial distance $z \geq 4.0 \cdot D_H$ the predicted and measured RMS velocity profiles are in very good agreement.

Realize Your Product Promise™

Concluding Remarks



Concluding Remarks

- ANSYS: successful OECD/ NEA MATiS-H benchmark participation
- CFD investigation methodology has been developed and applied, which had allowed to investigate the challenging blind benchmark case in the given timeframe and by applying required elements of CFD Best Practice Guidelines.
- ANSYS DM and ANSYS WB Meshing successfully applied to generate meshes with about 100 M mesh elements
- Massively parallel simulation runs on up to 180 CPU cores
- Compared ANSYS CFX and ANSYS Fluent:
  - URANS SST CC & RSM models
  - Scale-resolving turbulence model approaches ZLES & SAS-SST
  - Good agreement between ANSYS Fluent and ANSYS CFX for comparable turbulence model approaches on identical meshes.
- Finally the scale-resolving turbulence model approaches in both ANSYS CFD solvers predicted flow solutions for both MATiS-H benchmark geometries in good agreement with the KAERI LDV data.

ANSYS

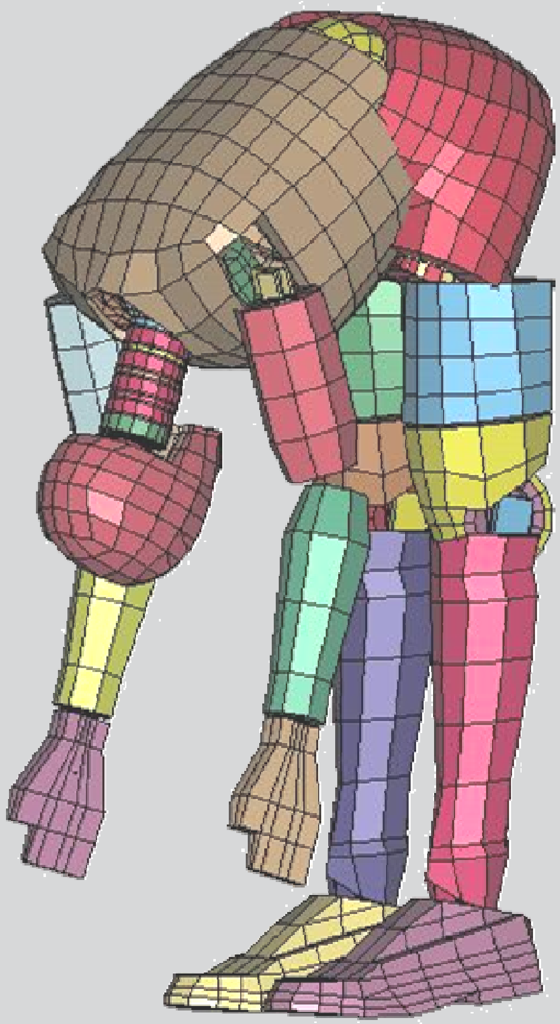
Acknowledgements

SPONSORED BY THE

Federal Ministry of Education and Research

This research has been supported by the German Ministry of Education and Research (BMBF, Grant No. 02NUK010G) in the framework of the R&D funding concept of BMBF "Basic Research Energy 2020+", the German CFD Network on Nuclear Reactor Safety Research and the Alliance for Competence in Nuclear Technology, Germany.

Questions ?



Concluding Remarks

- ANSYS successfully participated in the international OECD/NEA MATiS-H blind benchmark on turbulent flow through 5×5 rod bundle with two types of spacers.
- A CFD investigation methodology has been developed and applied, which had allowed to investigate the challenging blind benchmark case in the given timeframe and by applying required elements of CFD Best Practice Guidelines.
- Results from this precursor simulations on a simplified geometry had then be applied to the two spacer grid configurations.
- ANSYS Design Modeler and ANSYS Workbench Meshing have been successfully applied to the geometry preparation and generation of meshes with about 100 M mesh elements for both complex MATiS-H rod bundle configurations.

Concluding Remarks (cont.)

- Massively parallel simulation runs on up to 180 CPU cores have been carried out to investigate the benchmark in the given timeframe.
- ANSYS CFX and ANSYS Fluent with URANS SST & RSM models as well as scale-resolving turbulence model approaches (ZLES, SAS-SST) have been compared to each other. Comparable turbulence model approaches on identical meshes deliver CFD solutions which are in good agreement between ANSYS Fluent and ANSYS CFX.
- Finally the scale-resolving turbulence model approaches in both ANSYS CFD solvers have predicted flow solutions for both MATiS-H benchmark geometries in good agreement with the KAERI LDV data.

