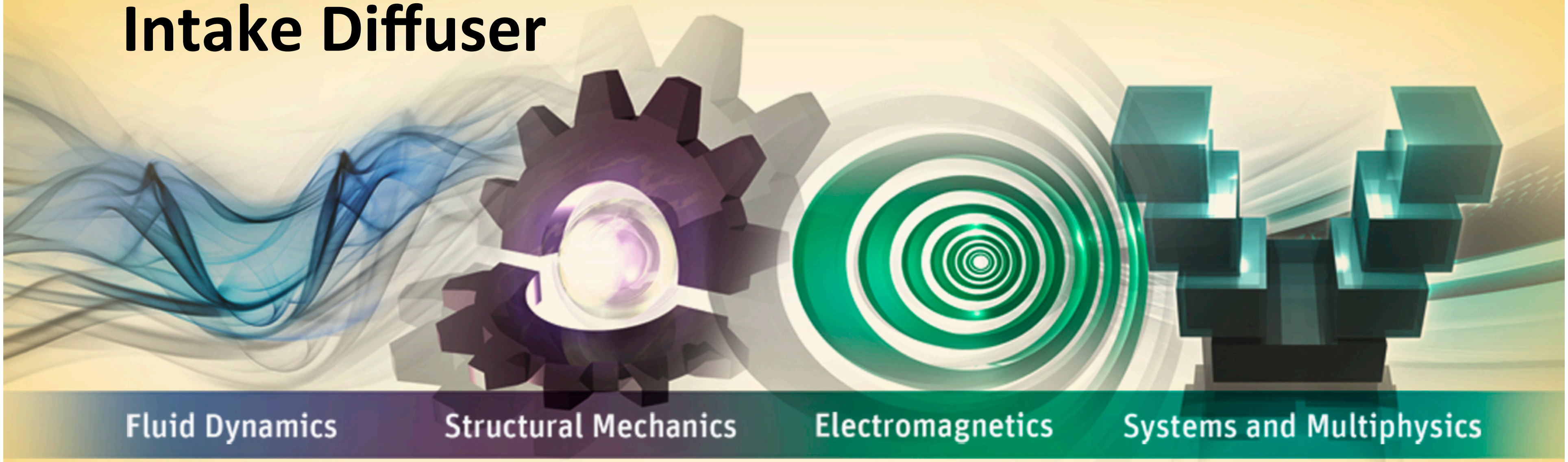
ANSYS

Realize Your Product Promise™

Pressure-based Coupled Simulation of Pressure Recovery and Distortion in S-shaped Intake Diffuser



Saravana Kumar

Bala Subramanyam

and

Konstantine A. Kourbatski, Ph.D.

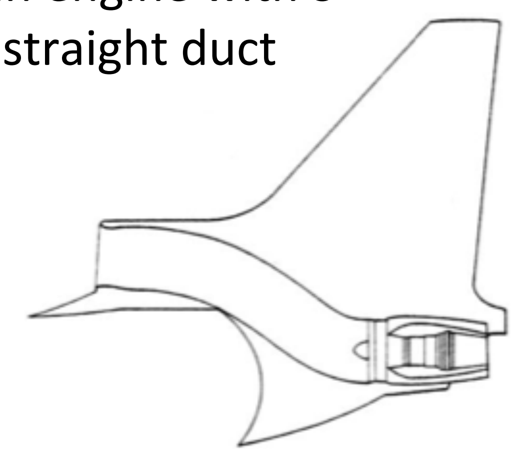
Applied Modeling & Simulation Seminar Series
NASA Ames Research Center, September 23, 2014

Outline

1. S-shaped Intake Diffuser (S-duct): applications and flow physics
2. Problem description
3. Numerical model
   - Pressure-based coupled solver
   - Physical models and boundary conditions
4. Computational meshes
5. Turbulence models
6. Results
   - Baseline model without Vortex Generator (VG) vanes
   - Effect of experimental instrumentation (baseline model with Kulite Probes)
   - Model with VGs
   - Comparison with and without VG Vanes
7. Comparison with experimental data

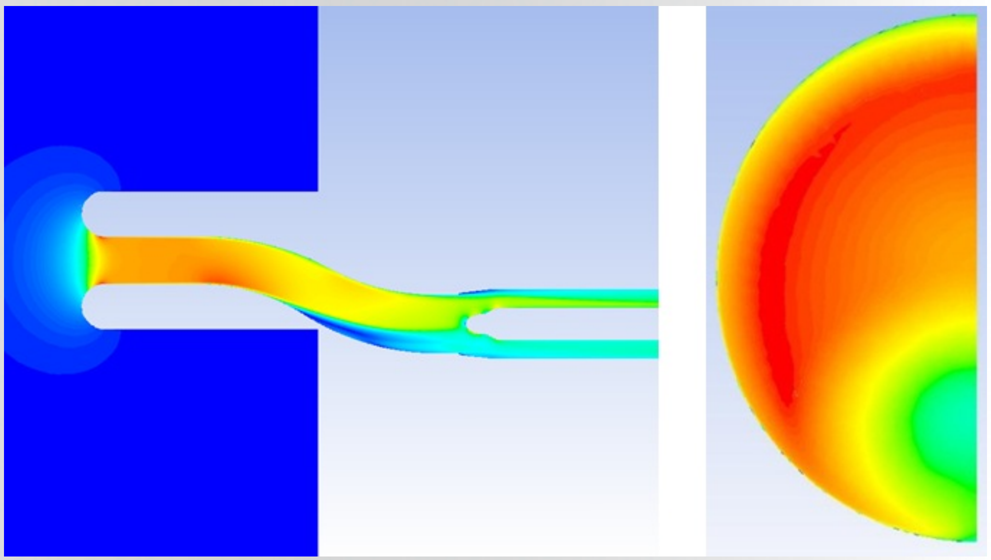
S-duct: Applications and Flow Physics

- S-shaped Intake Diffuser (S-duct) is used to direct the flow from atmosphere to aircraft engine intake
  - Most fighter jets
    - engine is buried into the fuselage
    - stealth
  - Some commercial aircrafts
    - inlet to the center engine on trijet aircraft
- Introduces losses to the system
- The flow inside the duct is complicated and diffusive because of duct's curvature
- Advances in flow control devices make it possible for an engine with S-duct inlet to have performance equivalent to that of a straight duct engine

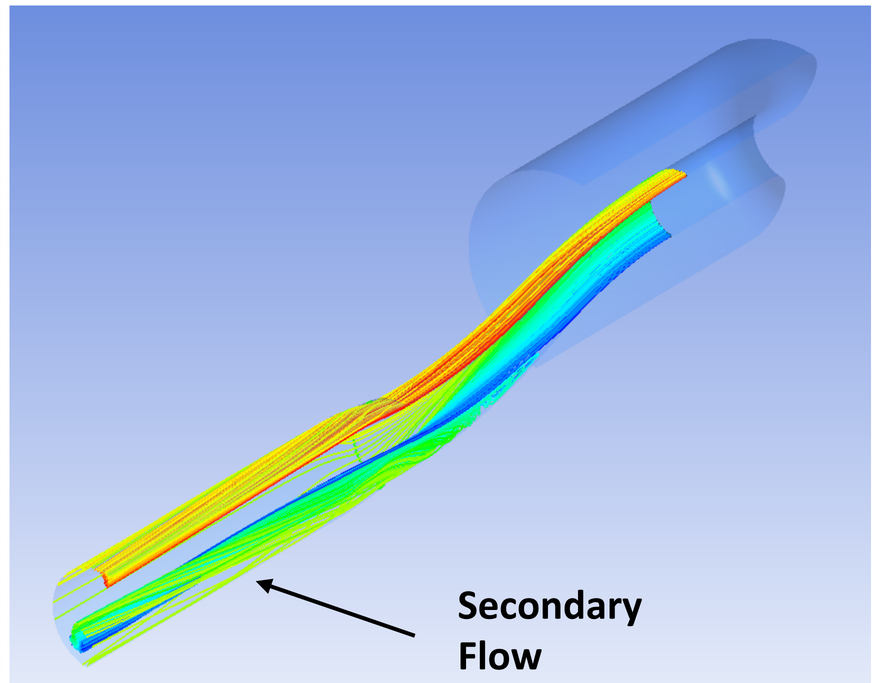


S-duct: Applications and Flow Physics

Flow Separation



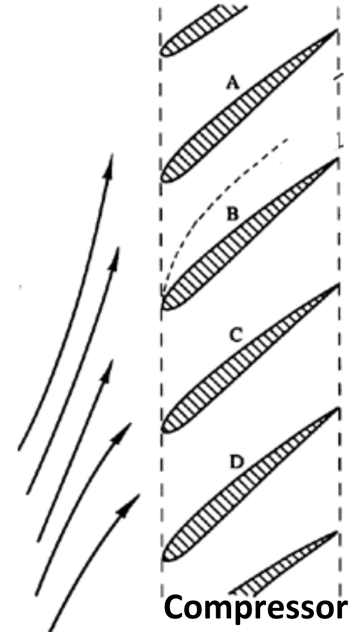
Secondary flow



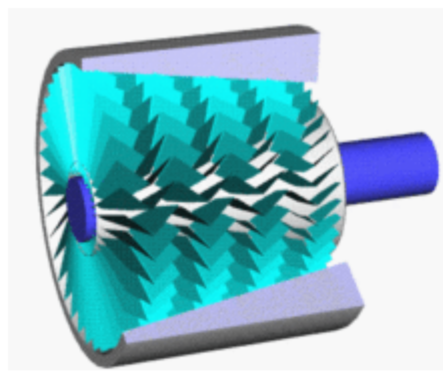
S-duct: Applications and Flow Physics

Effects of Secondary Flows and Separation

- Separation introduces total pressure losses which degrade compressor aerodynamics
- Secondary flows produce high distortion of flow at the compressor plane
  - may even lead to compressor surge
- Separation and secondary flows directly affect compressor, hence the total engine performance
- Performance of the duct is measured quantitatively using ***pressure recovery*** and ***distortion coefficients***



Compressor blades



Pressure Recovery and Distortion Coefficient

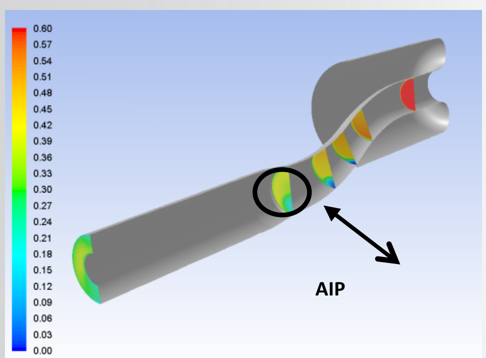
Pressure Recovery (PR)

$$PR = PT_2 / PT_1$$

PT_2 = Average total pressure at Aerodynamic Interface Plane (AIP)

PT_1 = Total pressure at inlet

Pressure recovery directly indicates the energy loss



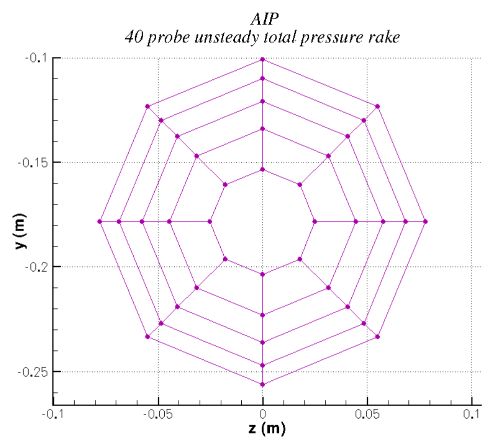
Circumferential Distortion Coefficient (DC)

$$DC = \frac{1}{5}\sum_{i=1}^{5} \frac{PAV_i - PAVLOW_i}{PAV_i}$$

PAV_i = Avg. Total pressure at ring i

$PAVLOW_i$ = Avg. Total pressure below PAV_i at ring i

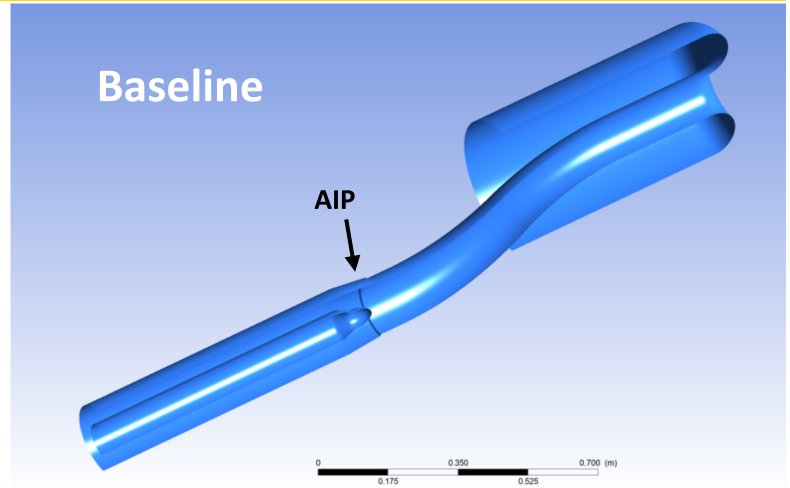
Distortion coefficient indicates the deviation of total pressure from average

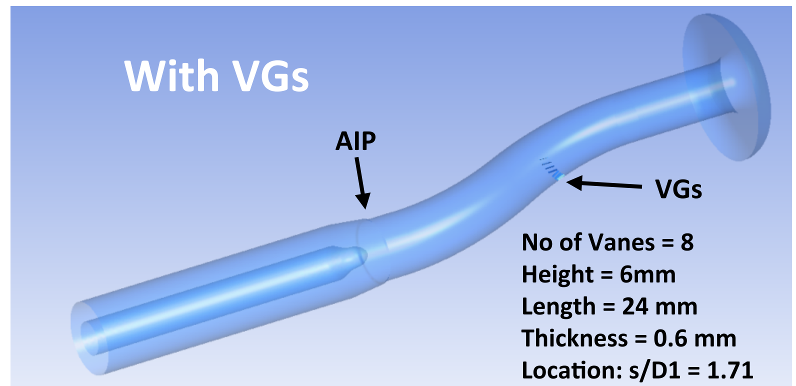


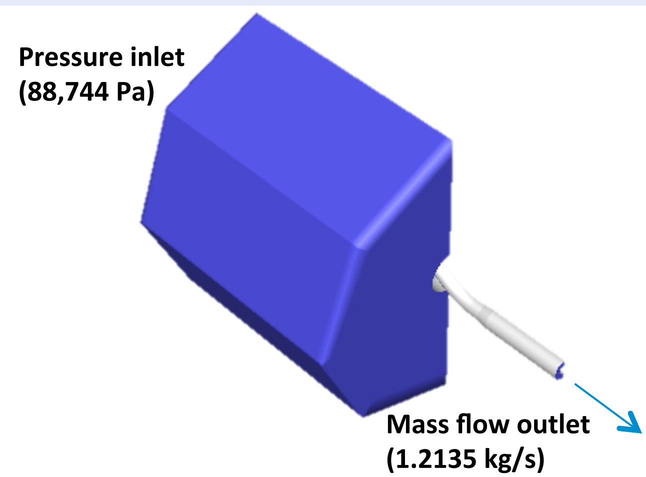
Problem Description

Benchmark problem of the ***1st and 2nd AIAA Propulsion Aerodynamics Workshops (PAW-I 2012 and PAW-II 2014):***

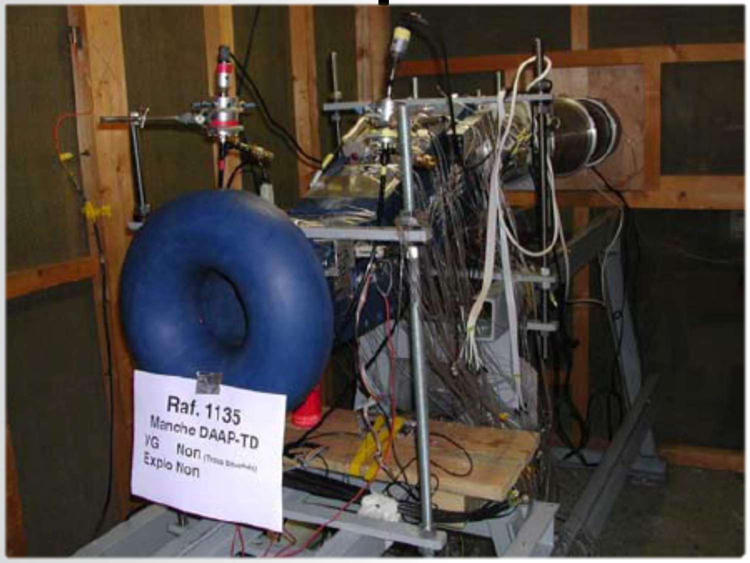
- Inlet diameter D_1 = 133.15 mm
- Outlet diameter D_2 = 164.00 mm
- Area ratio (outlet / inlet) = 1.52
- Length of S-duct = 5.23 x D_1
- Offset of the intake resulting from centerline curvature is 1.34 x D_1
- Flow Conditions:
  - Total Pressure = 88,744 Pa
  - Total Temperature = 286.2 K
  - Mass flow outlet = 2.427 kg/s
- 164 mm diameter pipe connects the outlet to the AIP equipped with 40 Kulites pressure probes

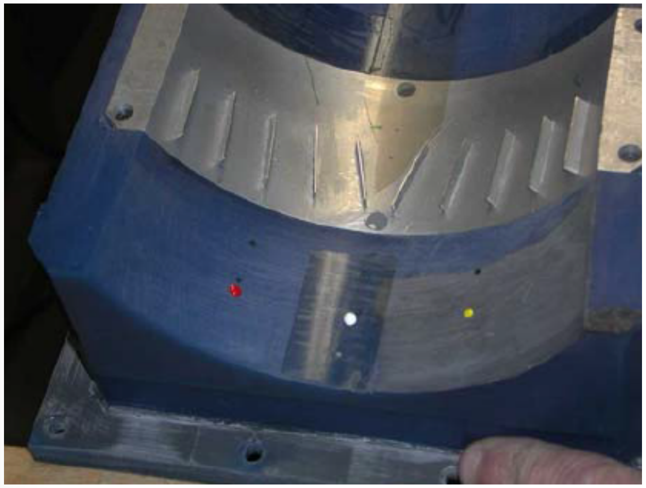






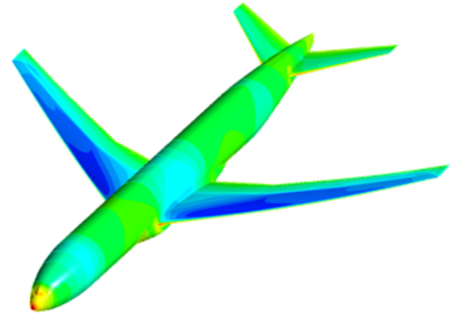
Experimental Setup – Model with VGs[1]





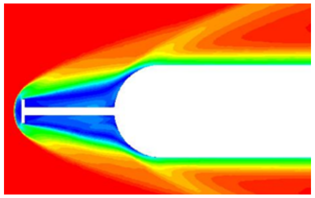
[1]Anne-Laure Delaut, Eric Garnier, Didier Pagan, "Flow Control in High-Offset Subsonic Air Intake," AIAA Paper 2011-5569. AIAA Joint Propulsion Conference & Exhibit.

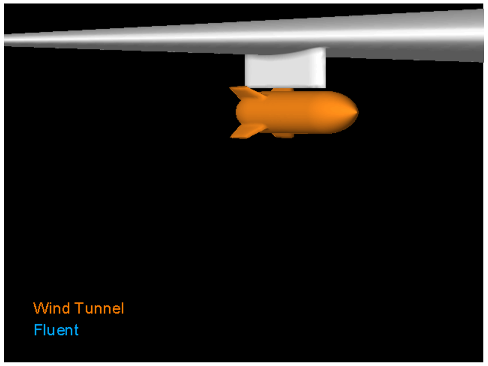
ANSYS Numerical Model

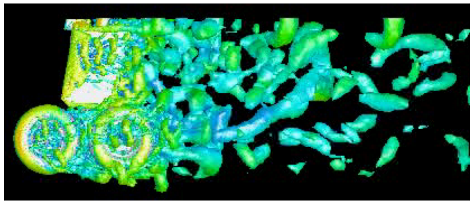


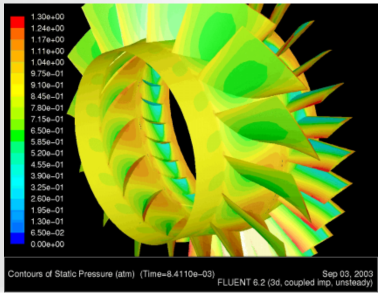
ANSYS Fluent R15.0

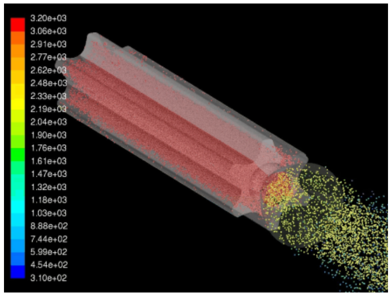
- General-purpose CFD solver
- 2D and 3D, incompressible, compressible, steady, transient
- Inviscid, laminar, turbulent (*RANS, RSM, LES, DDES, transition, …*)
- Acoustics
- Heat transfer, phase change and radiation
- Reacting flows: comprehensive chemical reaction modeling
- Multiphase: discrete phase, Eulerian, free-surface flows …
- Rotating machinery
- Moving and deforming meshes
- Optimization tools, adjoint solver
- Customizations tools
- Highly scalable parallel processing

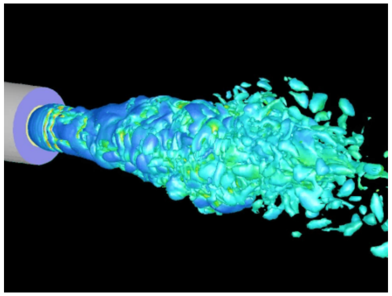












Numerical Model

- Steady-state simulation
- Control-volume-based technique
- Pressure-based coupled double-precision solver
  - pressure field is extracted by solving a pressure correction equation obtained by manipulating continuity and momentum equations
  - density is obtained from the equation of state
  - full implicit coupling between momentum and continuity equations
  - coupled system solved using coupled algebraic multigrid (AMG) scheme
  - Incomplete Lower Upper (ILU) smoother to smooth residuals between levels of AMG
- 2^{nd} order upwind scheme for interpolating face values from cell centers in the continuity, momentum and energy equations
- Least Squares cell-based gradient evaluation
  - 2^{nd} order spatial accuracy preserved

Turbulence Models – *Spalart-Allmaras*

- One-equation model proposed by *P. Spalart and S. Allmaras*[1]
- Solves a modeled transport equation for the kinematic eddy viscosity
- Designed for aerodynamic applications involving wall-bounded flows
- Effectively a low Re number model requiring the viscosity-affected near-wall region of the boundary layer to be properly resolved ($y^+ \sim 1$)
- In ANSYS Fluent, the Spalart-Allmaras model has been extended with a y^+ insensitive wall treatment
  - The formulation blends automatically from a viscous sublayer formulation to a logarithmic formulation based on y^+

[1] *P. Spalart and S. Allmaras. "A one-equation turbulence model for aerodynamic flows". Technical Report AIAA-92-0439. American Institute of Aeronautics and Astronautics. 1992.*

Turbulence Models - *Realizable k-ε*

- Proposed by *Shih et al*[1]
- Intended to address deficiencies of traditional k-ε models
  - flows with strong streamline curvature, vortices, and rotation, round jet anomaly
- The model adopts,
  - new eddy-viscosity formula involving a variable C_μ
  - ε equation based on dynamic equation of the mean-square vorticity fluctuation
- Two-layer enhanced wall treatment for y+ ~ 1 meshes in ANSYS Fluent,
  - *k*-ε formulation is employed in the fully turbulent region
  - one-equations model[2] is solved in the viscosity-affected near-wall region
  - eddy viscosity in smoothly blended between the near-wall and outer regions

[1]*Shih, T.-H., Liou, W. W., Shabbir, A., Yang, Z., and Zhu, J., "A New k-ε Eddy-Viscosity Model for High Reynolds Number Turbulent Flows - Model Development and Validation," Computers Fluids, Vol. 24, No.3, 1995, pp. 227-238.*

[2]*M. Wolfshtein. "The Velocity and Temperature Distribution of One-Dimensional Flow with Turbulence Augmentation and Pressure Gradient". Int. J. Heat Mass Transfer. 12. 301–318. 1969.*

Turbulence Models - *SST k-ω*

- Proposed by *Menter*[1]
- Blends the robust and accurate formulation of the k-ω model in the near-wall region with the free-stream independence of the k-ε model in far field
- Effectively a low Re number model requiring the viscosity-affected near-wall region of the boundary layer to be properly resolved ($y^+ \sim 1$)
- In ANSYS Fluent, the SST model has been extended with a y^+ insensitive wall treatment
  - The formulation blends automatically from a viscous sublayer formulation to a logarithmic formulation based on y^+

[1] Menter, F. R., "Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications," AIAA Journal, Vol. 32, No. 8, 1994, pp. 1598-1605.

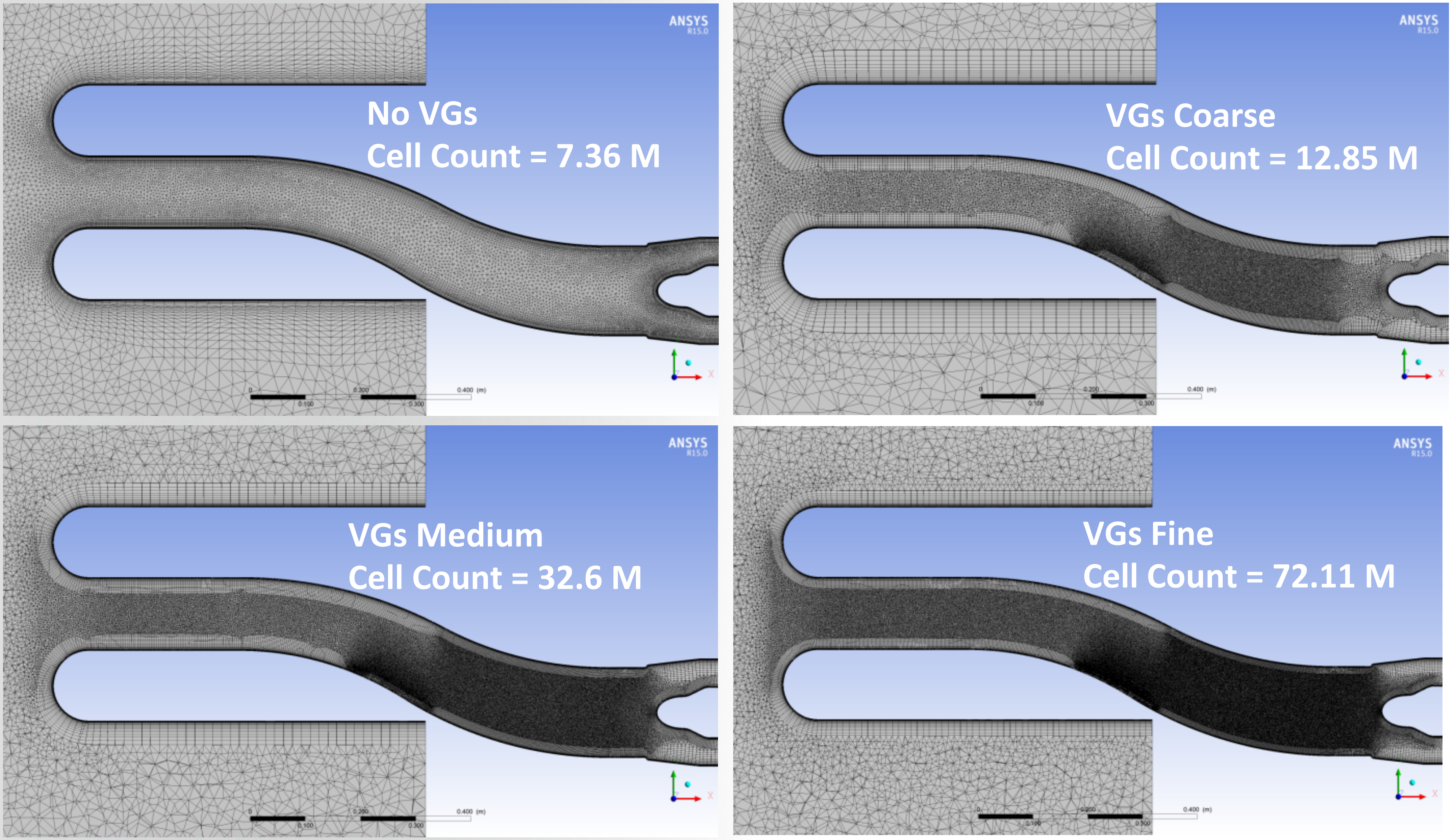
Turbulence Models – *Reynolds Stress Model*

- Reynolds Stress Model – Stress Omega (RSM)[1, 2]
- Solves equations for each Reynolds stresses, together with an equation for the dissipation rate
- More accurate for flows with anisotropic turbulence, e. g. cyclone flows, highly swirling flows, stress-induced secondary flows in ducts
- Computationally more expensive compared to 2-equation models
- Requires modeling of the pressure-strain and dissipation-rate terms
- Relies on a scale equation (ε- or ω-)
- Modeling of the pressure-strain term in Low-Re Stress Omega formulation of RSM:
  - Based on the omega equations and the Launder-Reece-Rodi (LPR) model[2]

[1] *E. Launder, G. J. Reece, and W. Rodi. "Progress in the Development of a Reynolds-Stress Turbulence Closure". J. Fluid Mech.. 68(3). 537–566. April 1975*

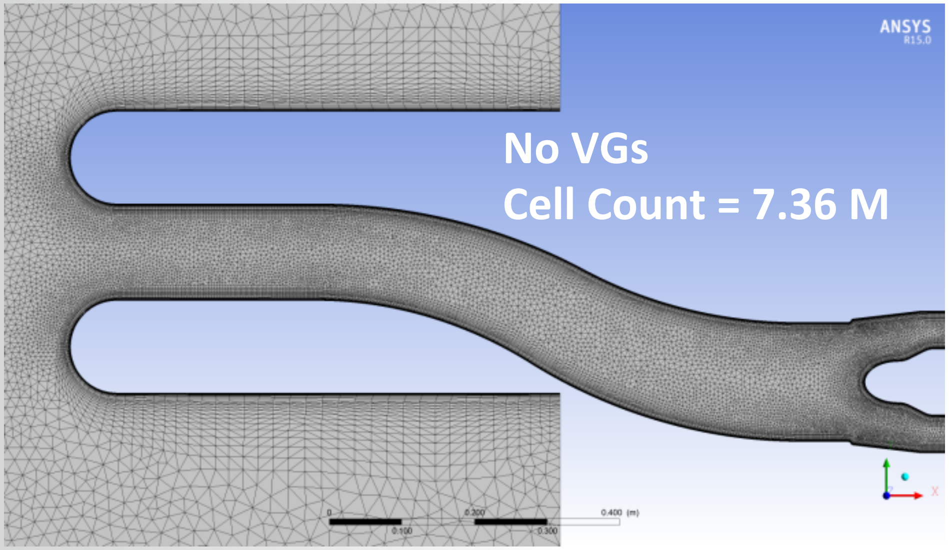
[2] *D. C. Wilcox. Turbulence Modeling for CFD. DCW Industries, Inc. La Canada, California. 1998.*

Computational Meshes

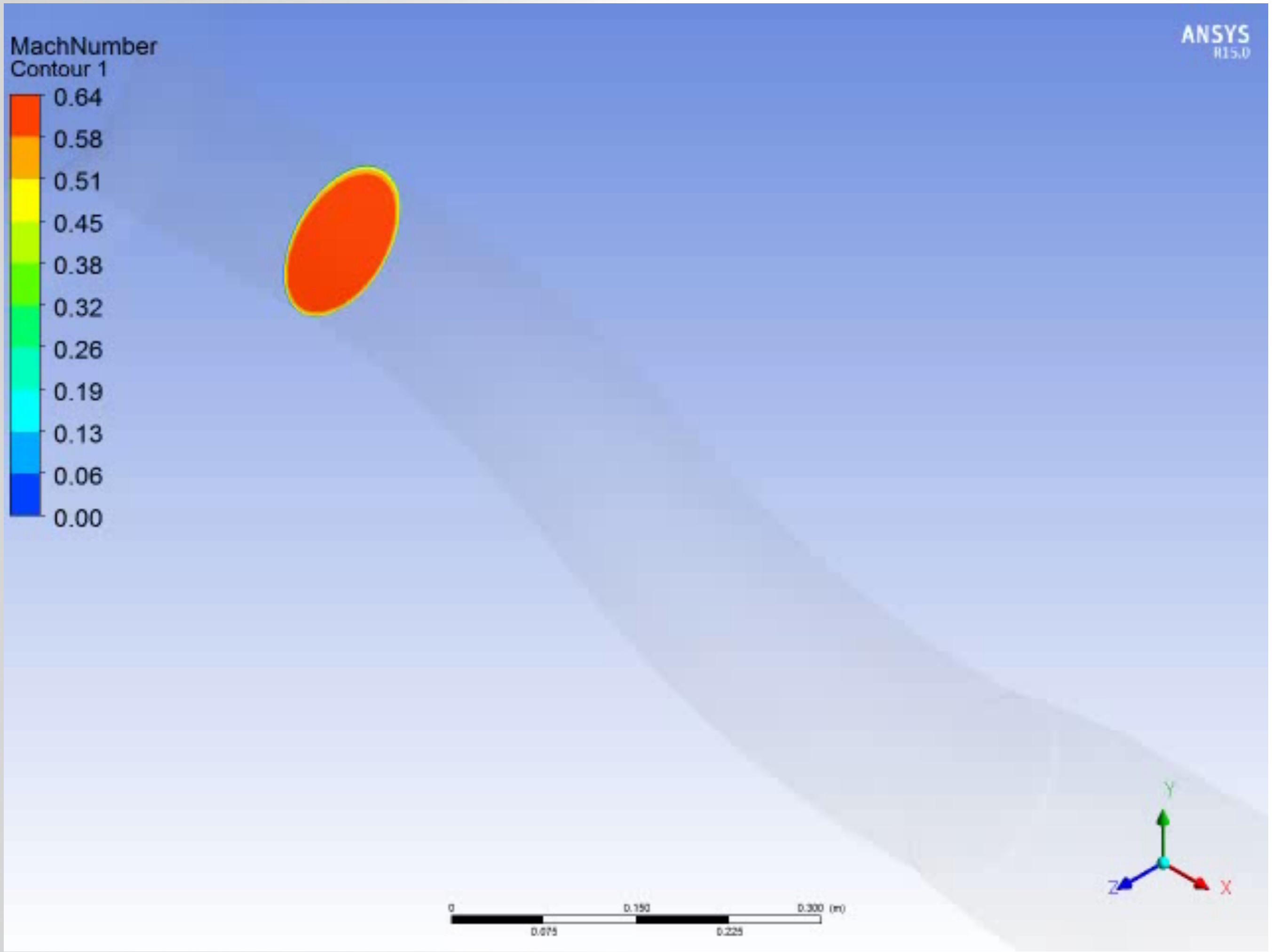


Wall Y+ ~ 1

Results – *Baseline without VGs*

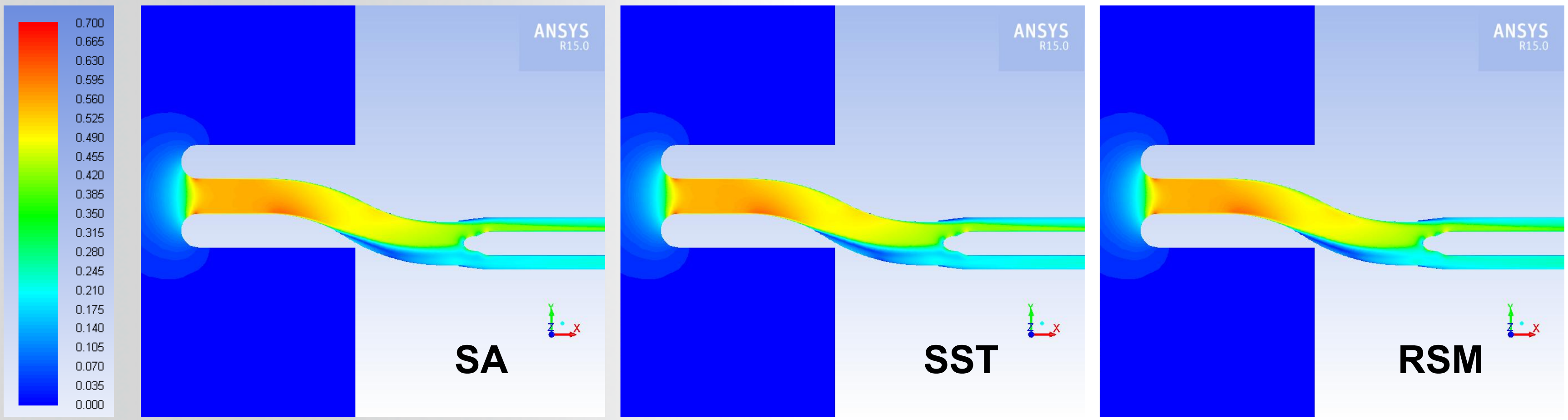


Results – *Baseline without VGs*

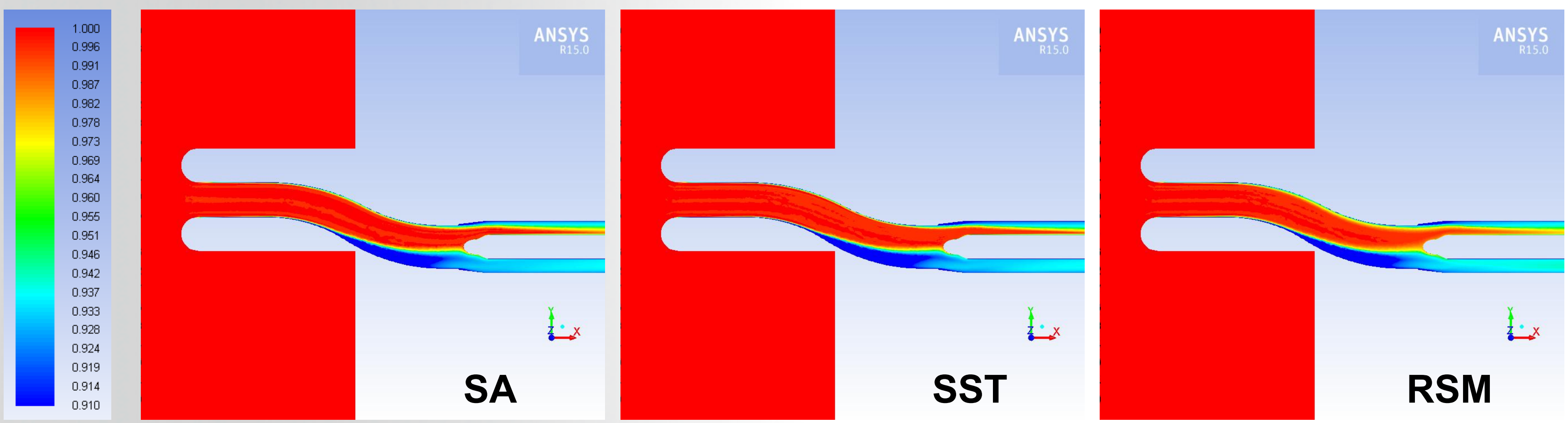


Turbulence Model Comparison

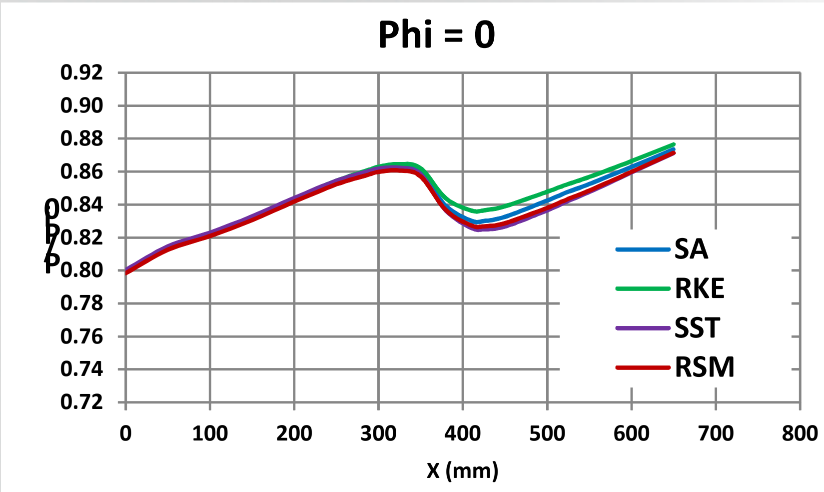
Mach Number

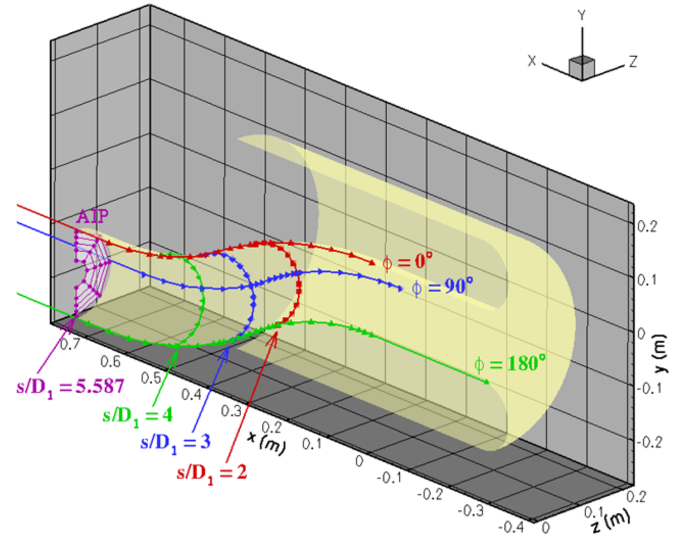


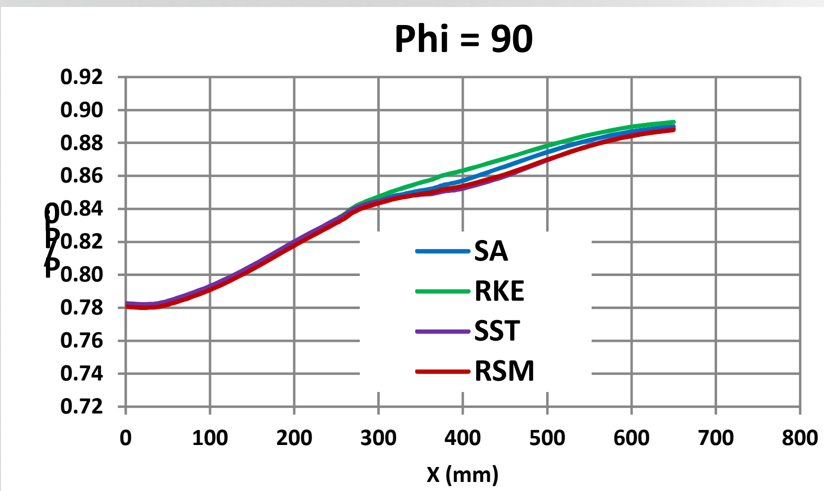
Total Pressure Ratio

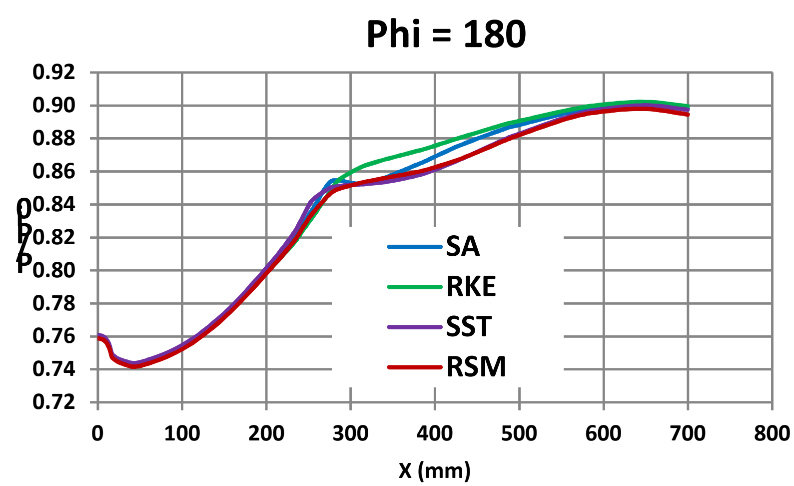


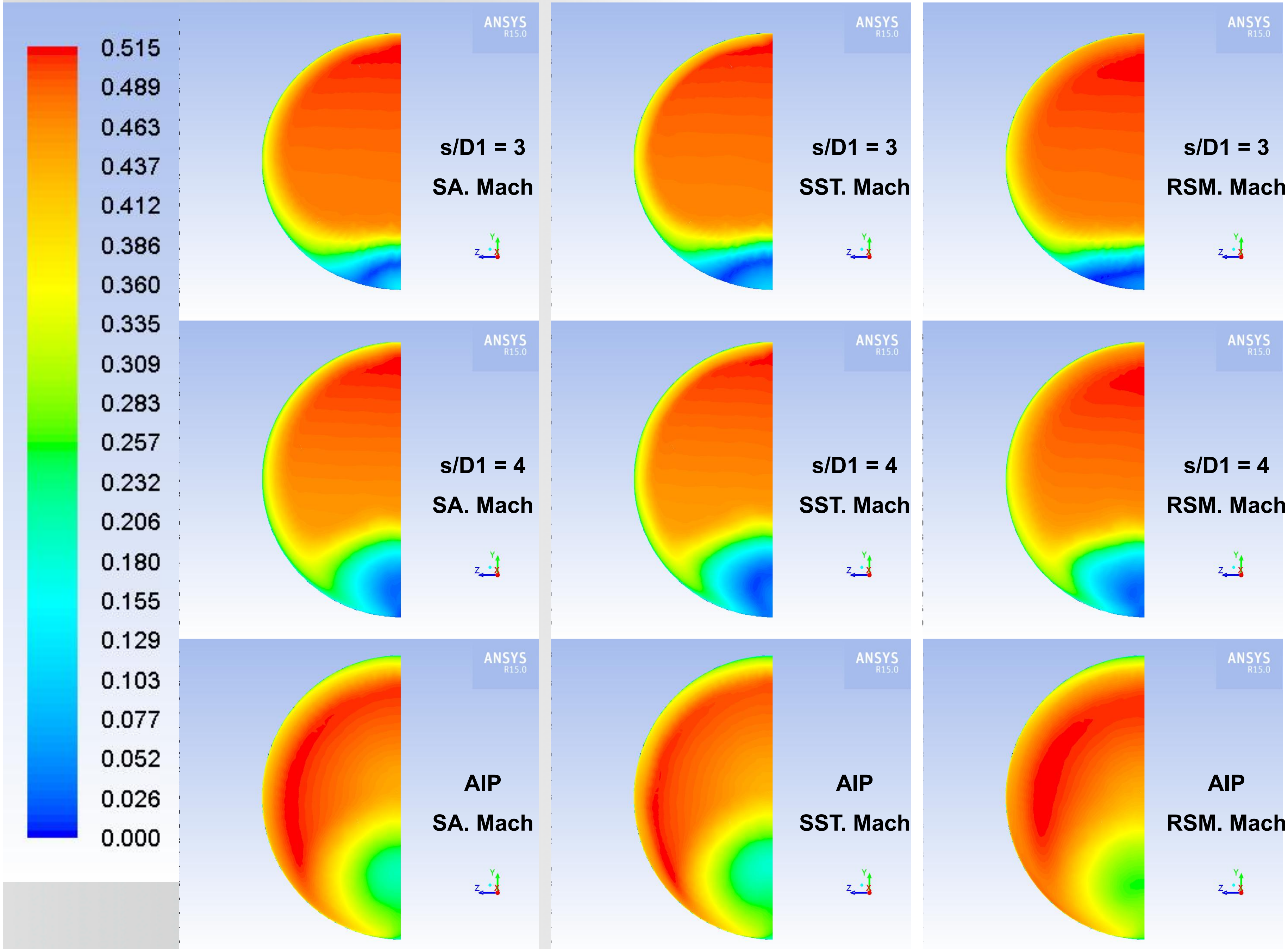
Turbulence Model Comparison





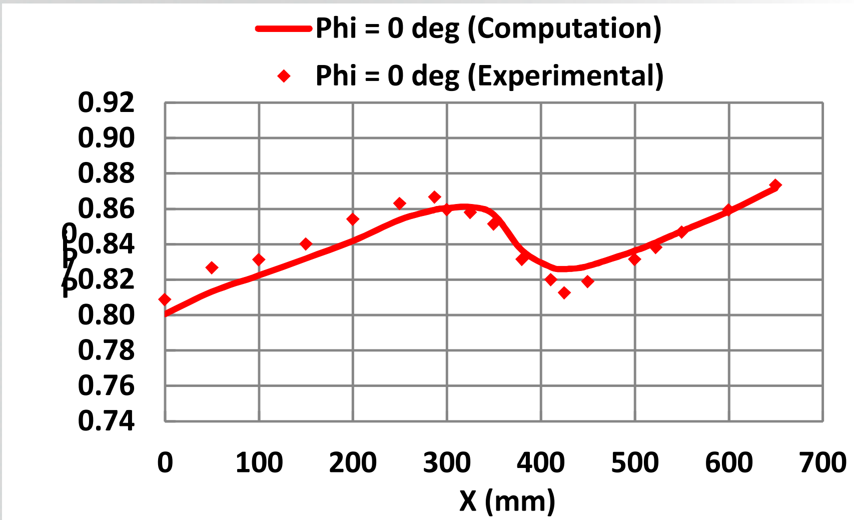


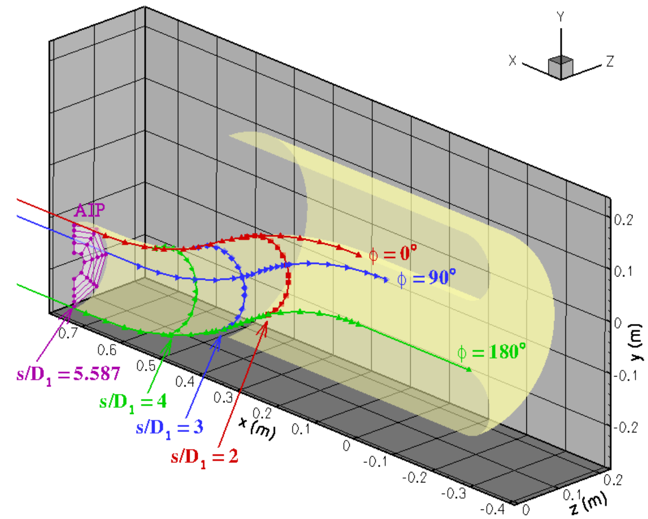


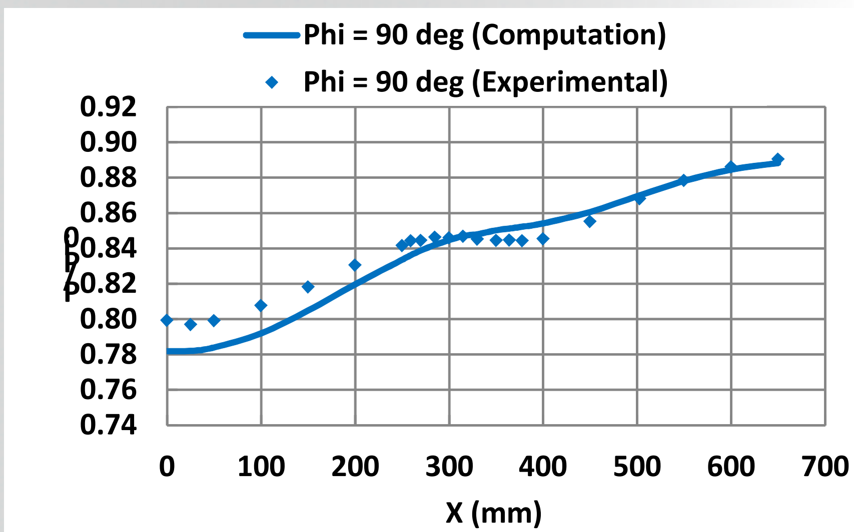

0.515
0.489
0.463
0.437
0.412
0.386
0.360
0.335
0.309
0.283
0.257
0.232
0.206
0.180
0.155
0.129
0.103
0.077
0.052
0.026
0.000
s/D1 = 3
SA. Mach
s/D1 = 3
SST. Mach
s/D1 = 3
RSM. Mach
s/D1 = 4
SA. Mach
s/D1 = 4
SST. Mach
s/D1 = 4
RSM. Mach
AIP
SA. Mach
AIP
SST. Mach
AIP
RSM. Mach

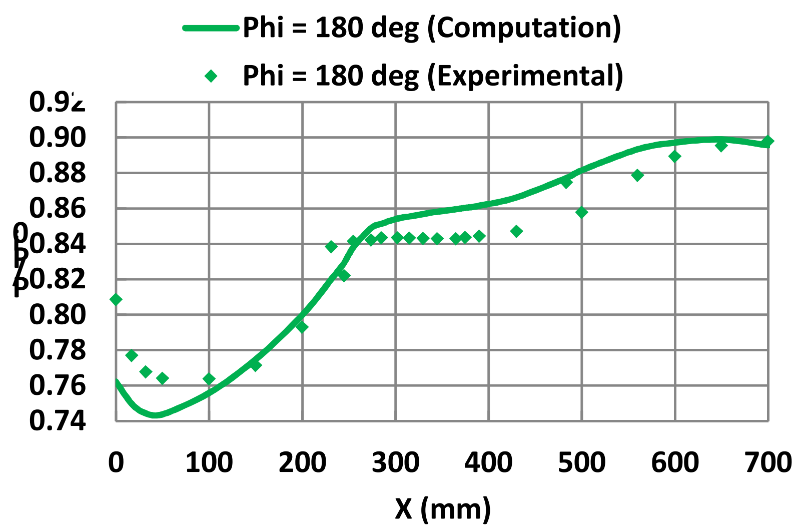
ANSYS

Static Pressure Distribution (RSM)
Comparison with Experiment









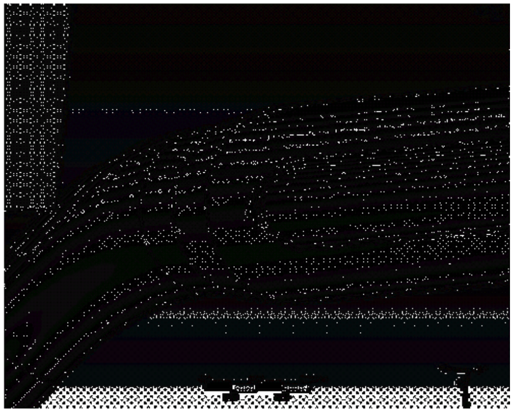
Observations – *Baseline Model*

Models	SA	RKE	SST	RSM	Exp. Value
PR	0.9728	0.9747	0.9724	0.9695	0.9700
DC	0.0276	0.0311	0.0317	0.0252	0.0200

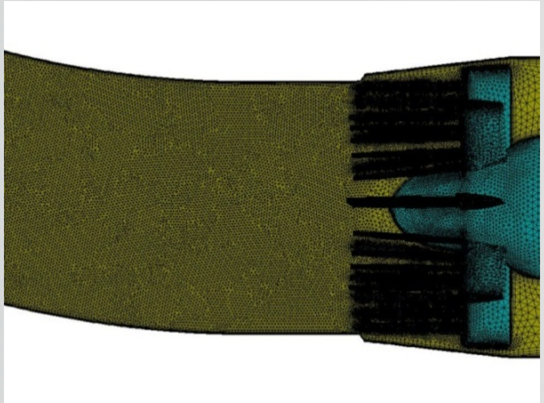
- All models predicted similar surface static pressure trends
- Static pressure on the wall in the streamwise direction calculated by SA and RKE deviates from SST and RSM curves after the separation point
- SST model predictions are closer to the RSM predictions than SA and RKE
- RSM predicts DC values closer to the experimental measurement
- Compared to pressure recovery, distortion coefficient is slightly overpredicted
  - DC predictions are within the standard deviation interval of test data
  - Test data were not corrected to the effect of Kulite probes at the AIP
  - Addition simulation was done to evaluated the effect of experimental probes

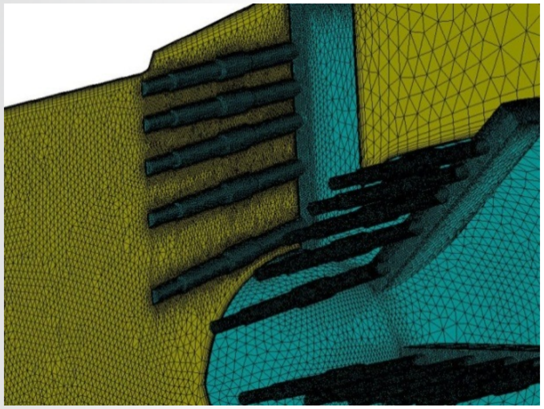
Model with Kulite Pressure Probes

- Added the probes and racks
- Created the mesh based on the mesh refinement study using ANSYS Meshing
- Similar meshes with and without probes were created to remove mesh effect
  - Cell Count ~ 19 Million
  - Y^+ ~ 0.88

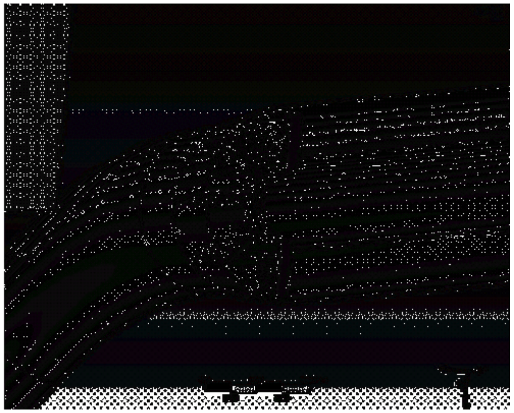


Baseline model



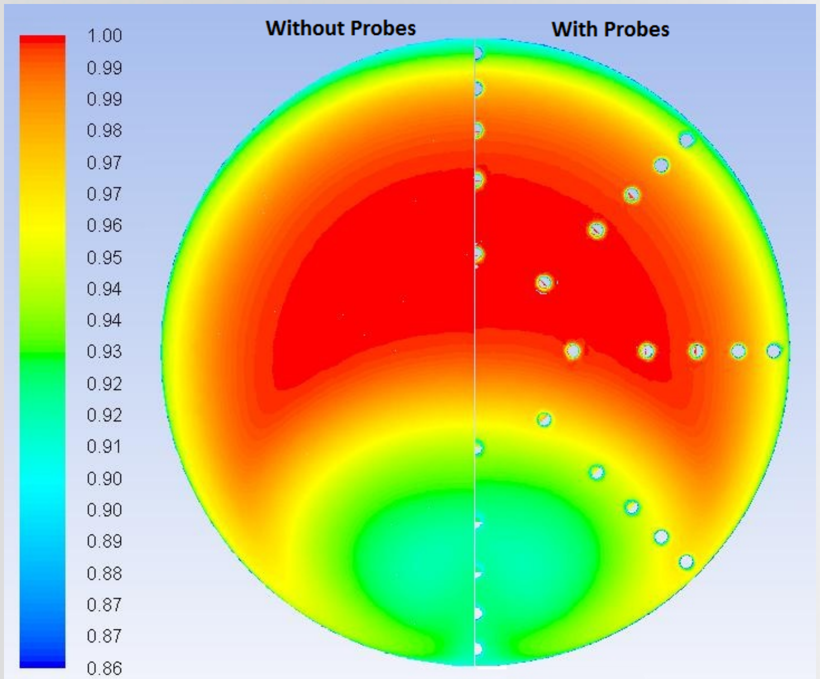


Two views of computational mesh with Kulite probes



Model with Kulite probes

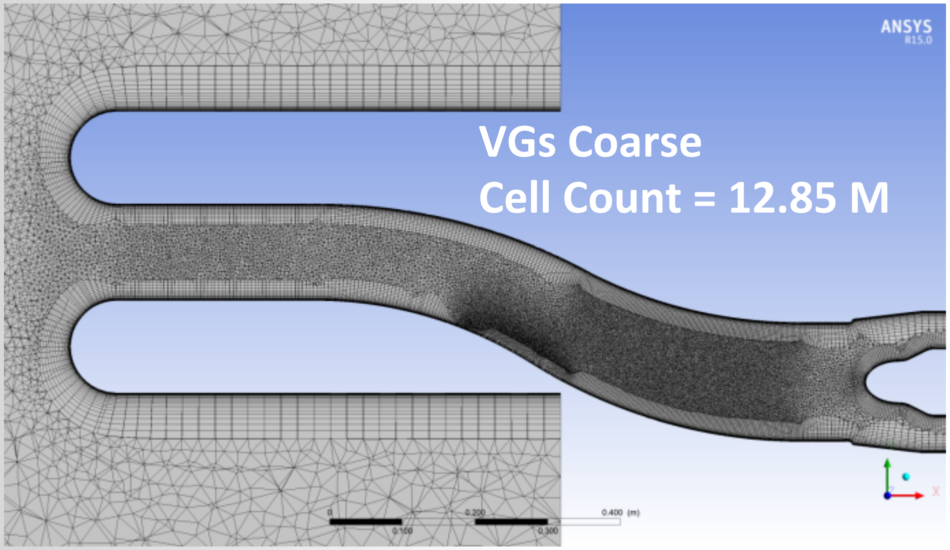
Comparison of Results with and without Probes

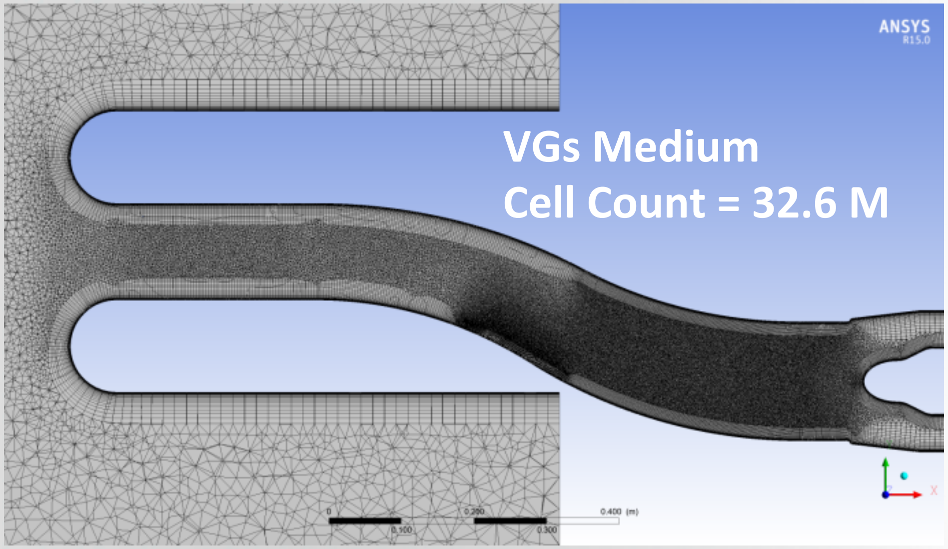


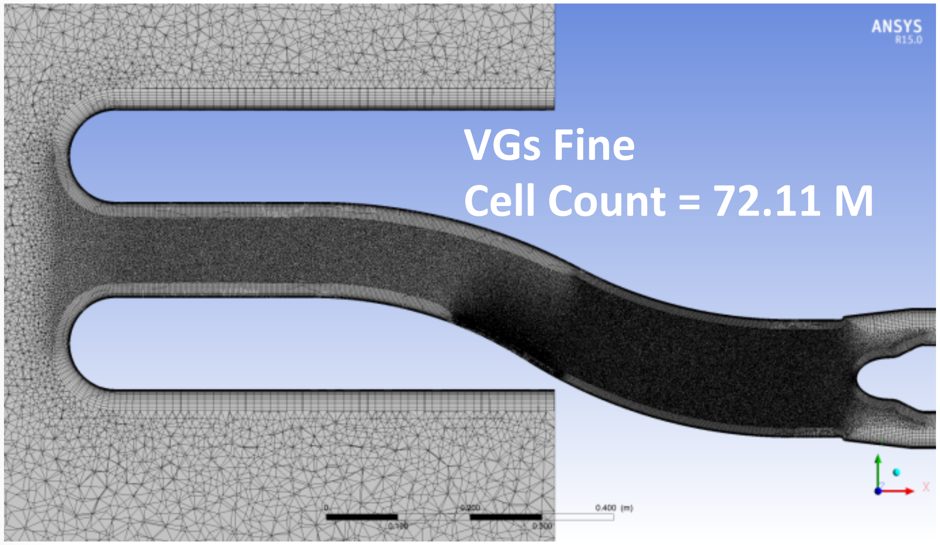
Mesh	Baseline	With Probes	Experimental Values
Models	RSM	RSM	
PR	0.9695	0.9702	0.9700
DC	0.0252	0.0231	0.0200

- DC prediction has improved after including pressure probes and racks
  - **now within 95% confidence interval**
- No impact is seen on the duct wall pressure profiles upstream of the AIP
- Difference in the results can be attributed to blockage effects introduced by probes

Results – *Model with VGs*

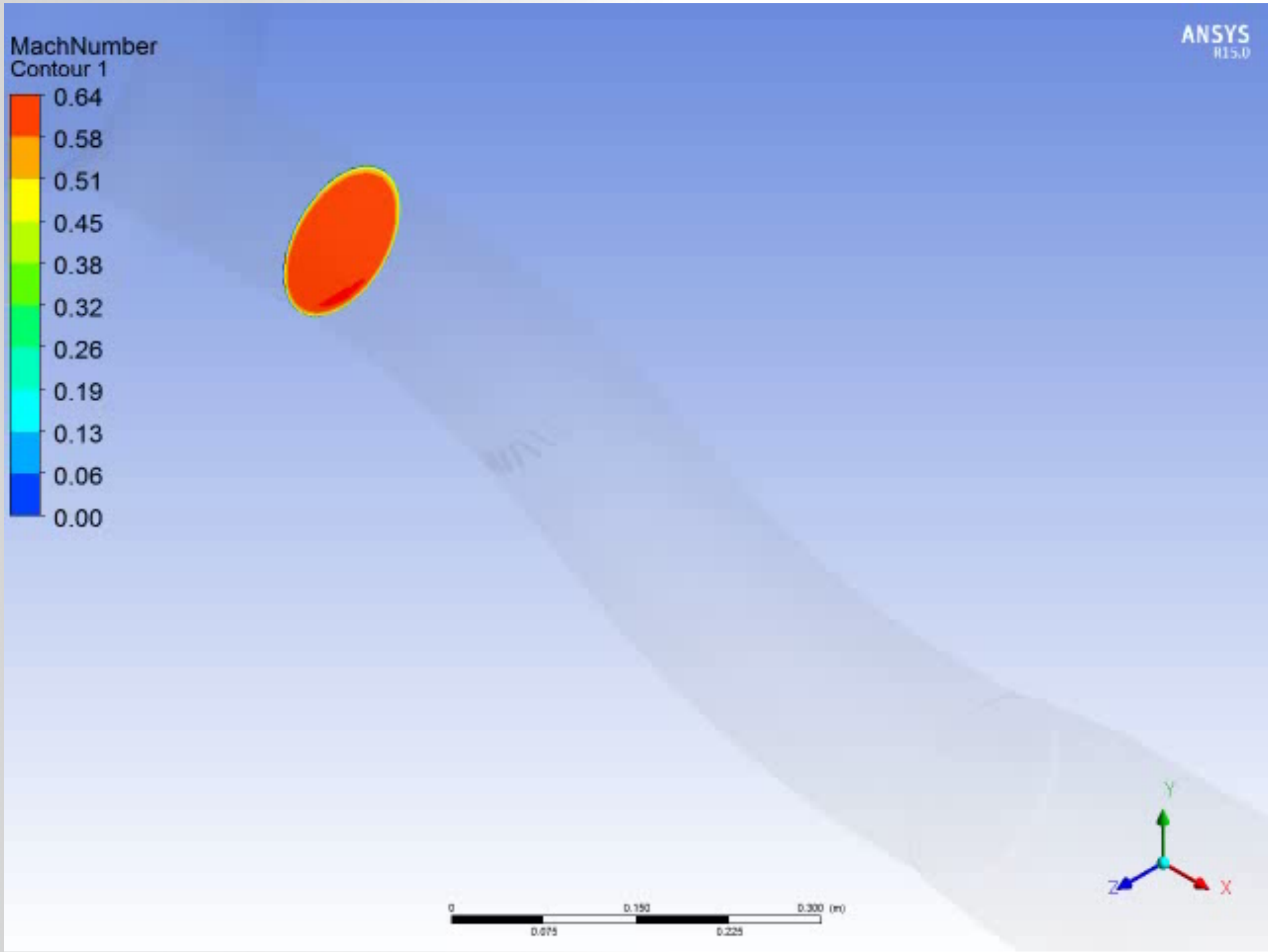




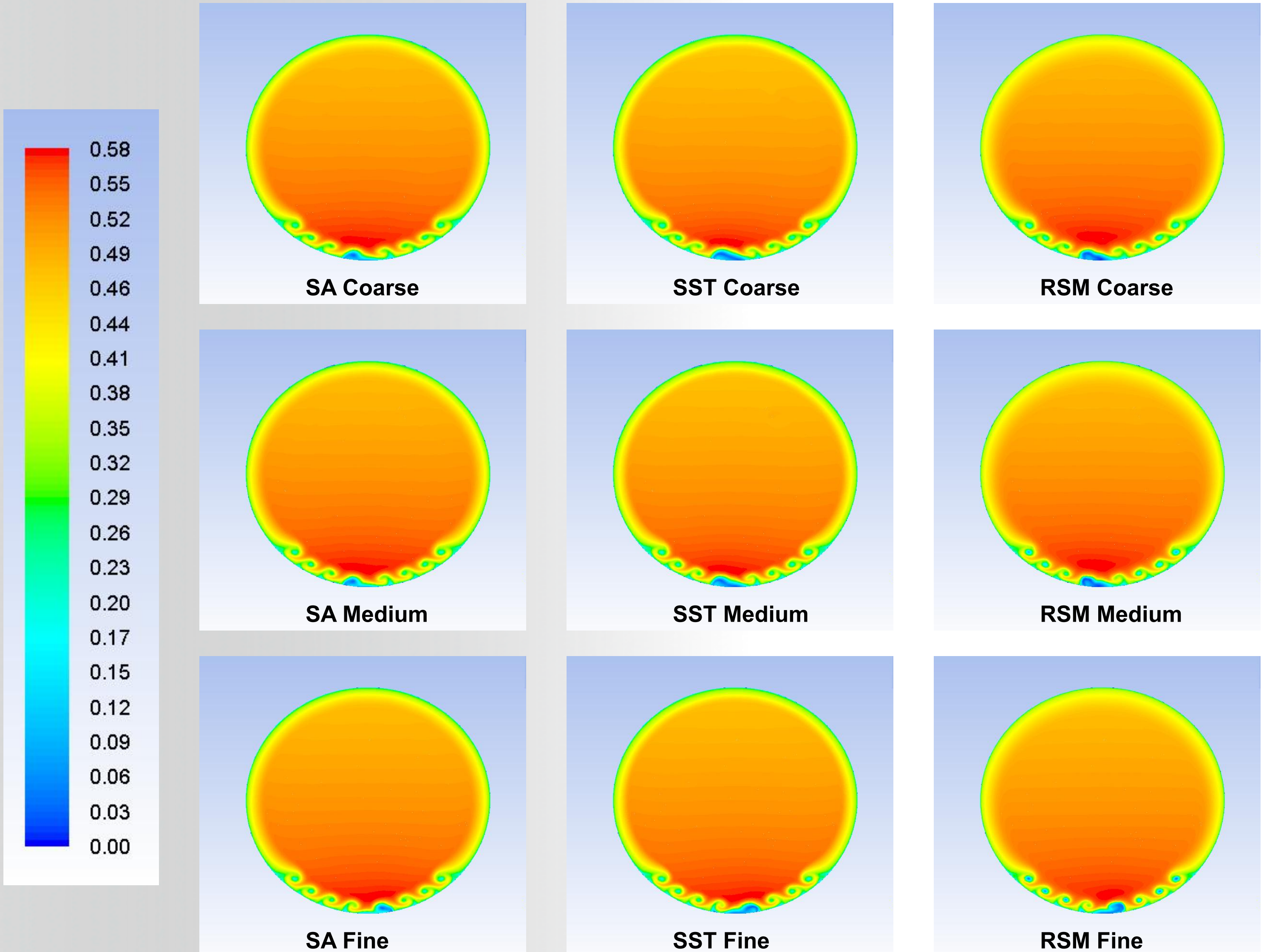


Wall Y+ ~ 1

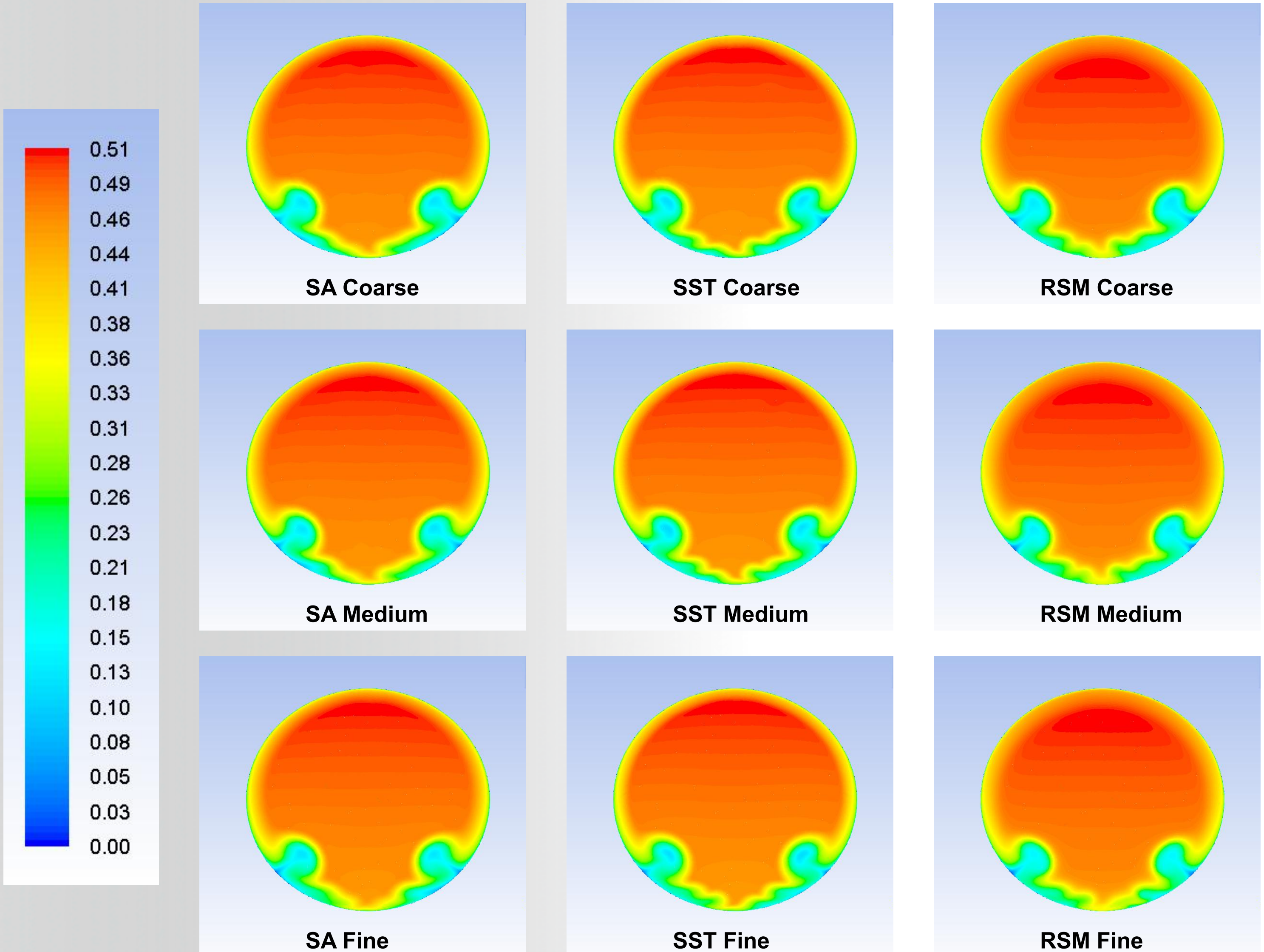
Results – *Model with VGs*



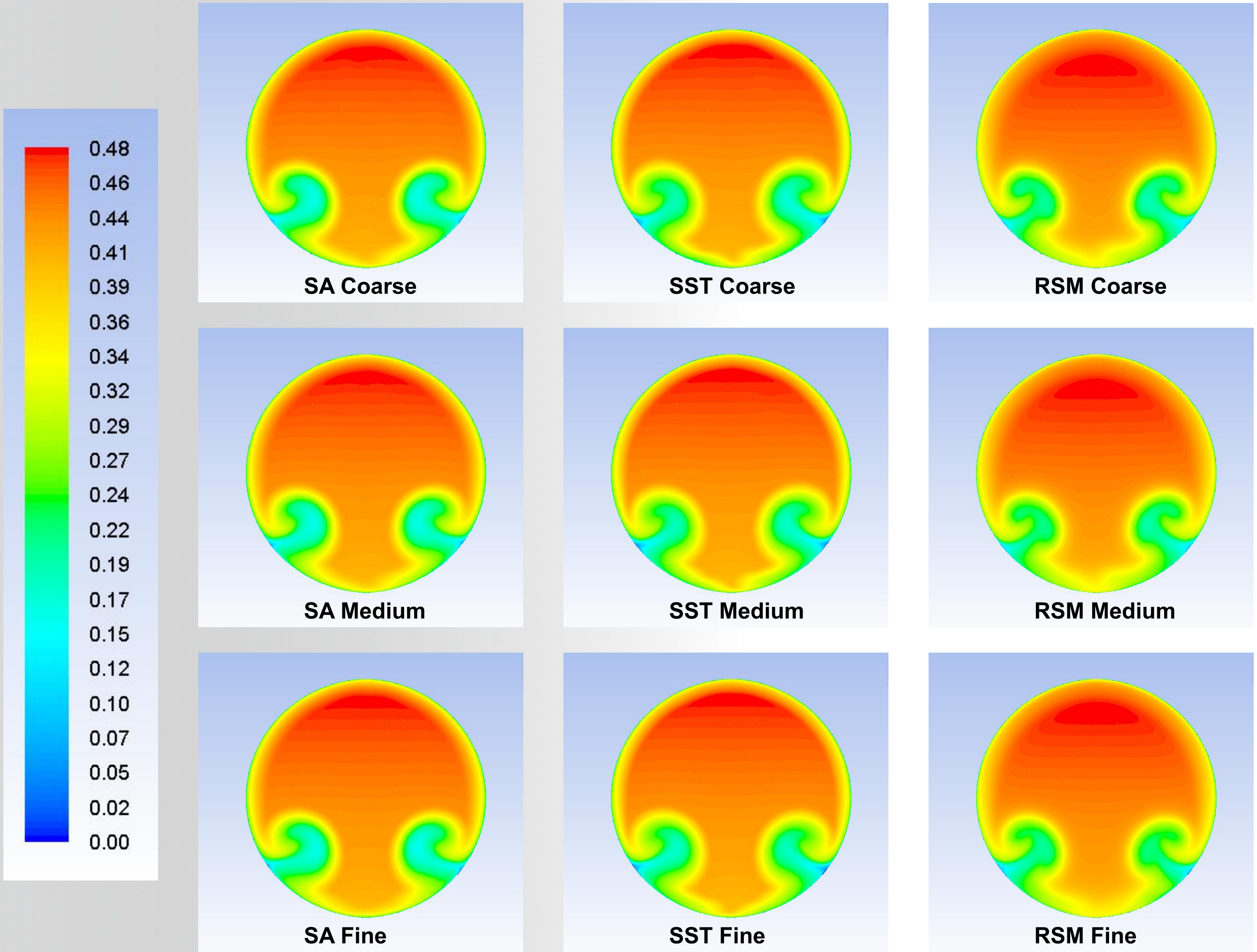
Mach Contours – s/D1 = 2



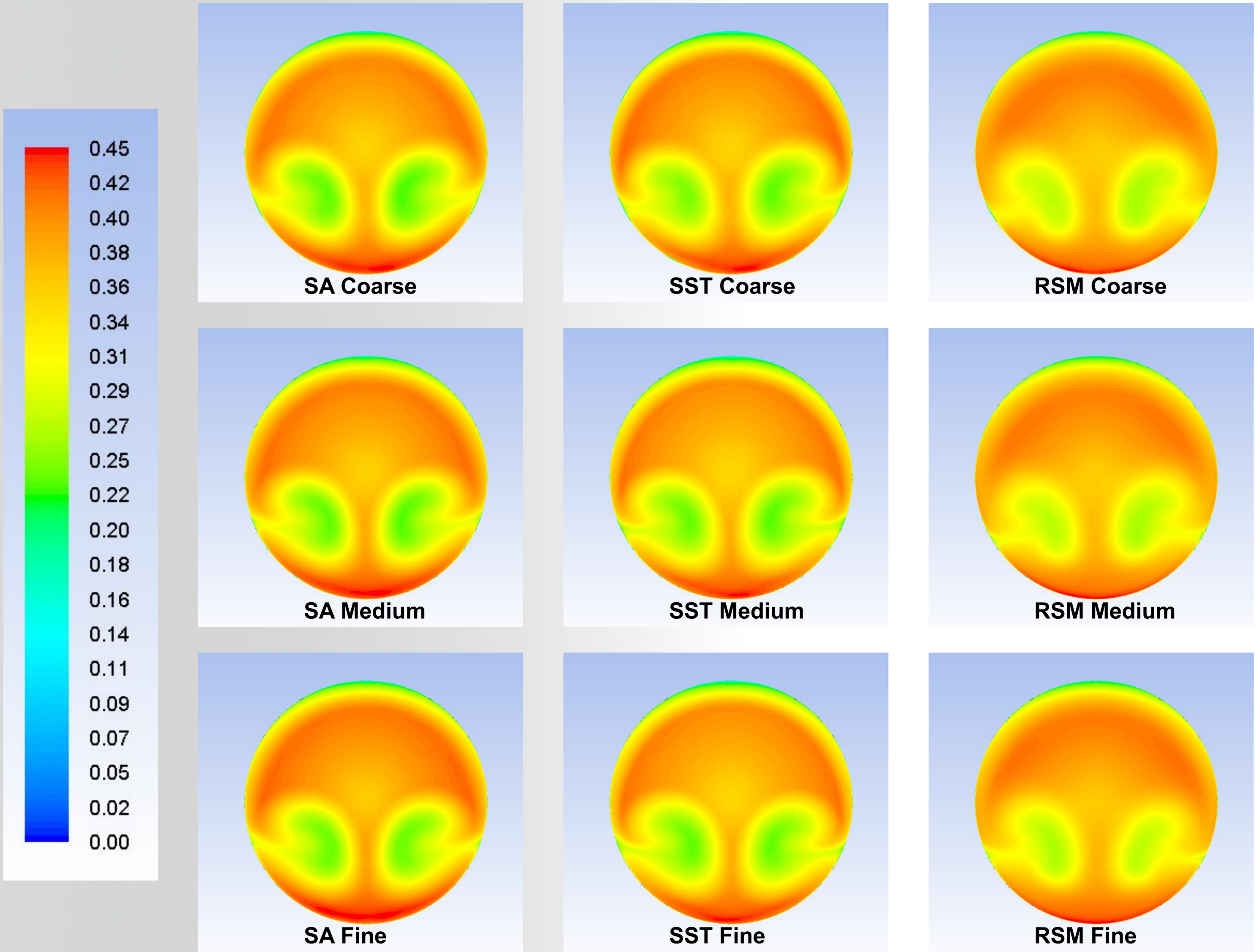
Mach Contours – s/D1 = 3



Mach Contours – s/D1 = 4



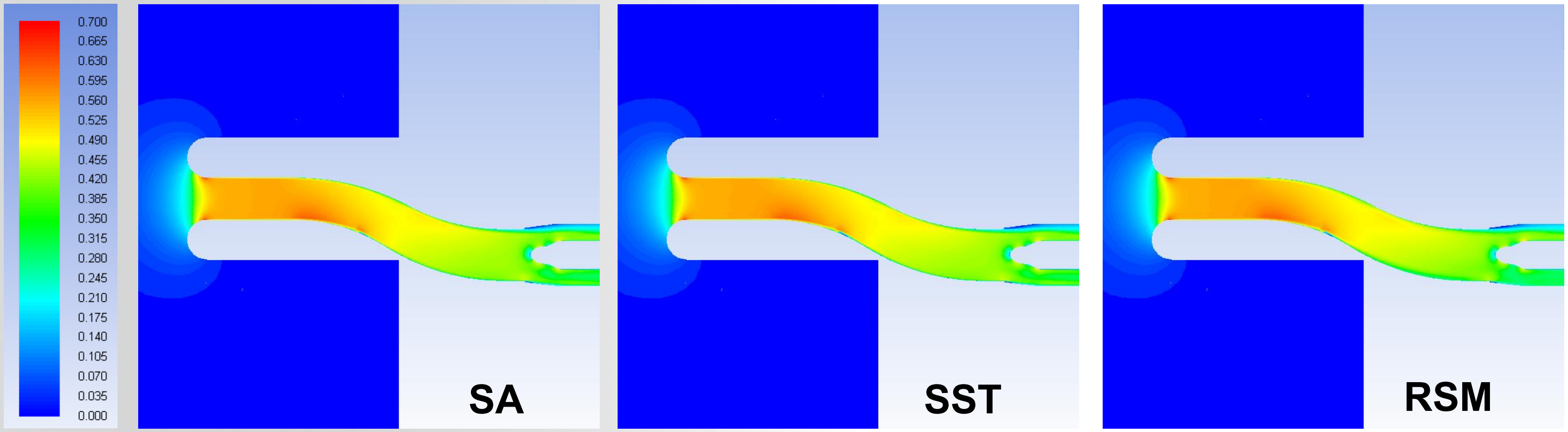
Mach Contours – AIP



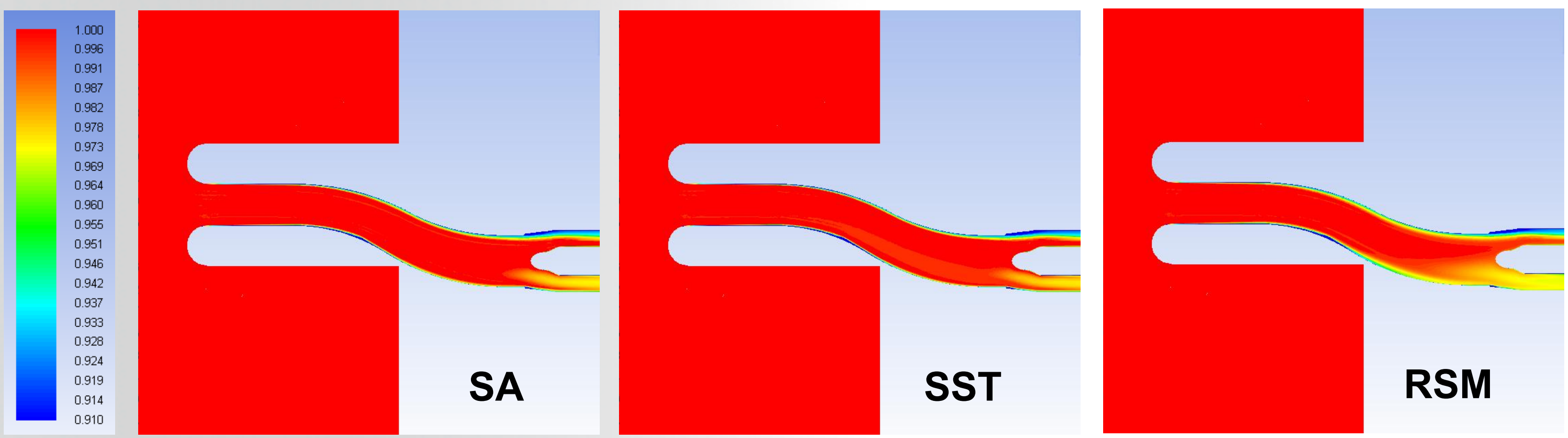
Turbulence Model Comparison

Medium Mesh

Mach Number

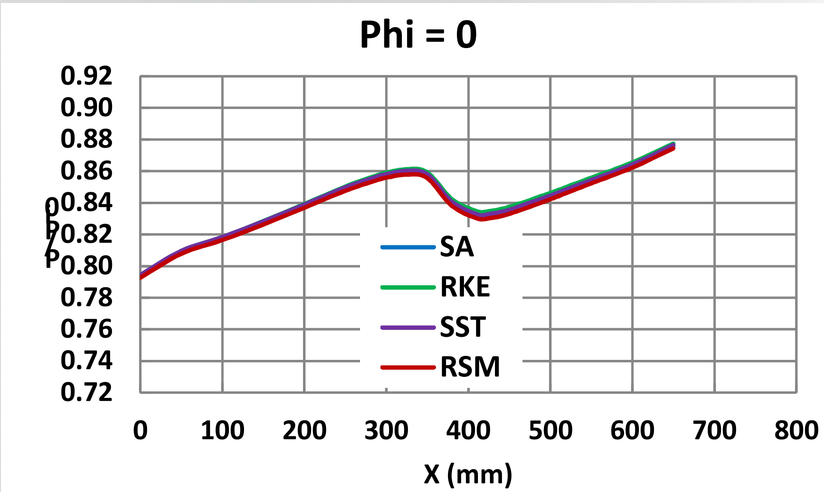


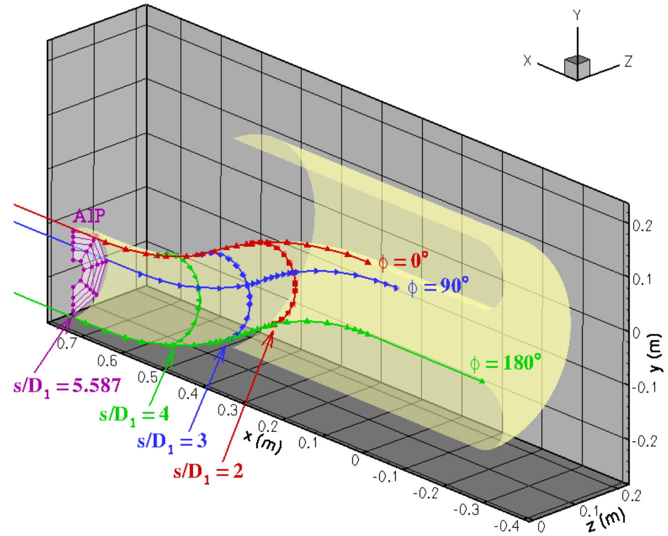
Total Pressure Ratio

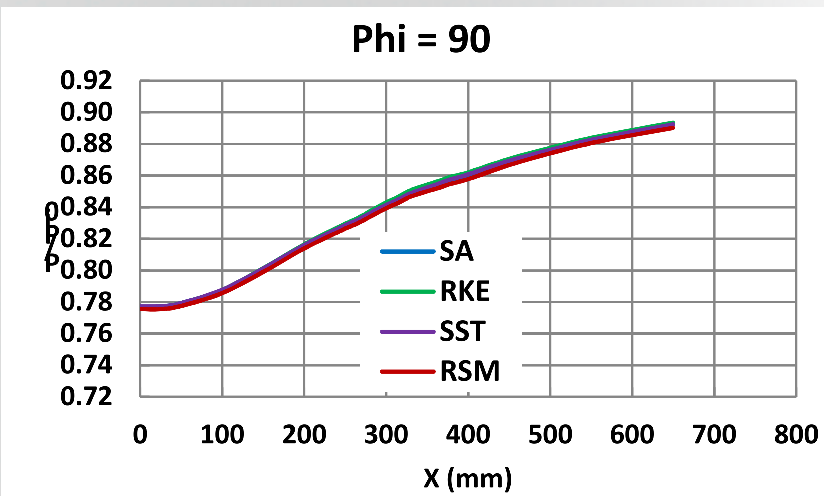


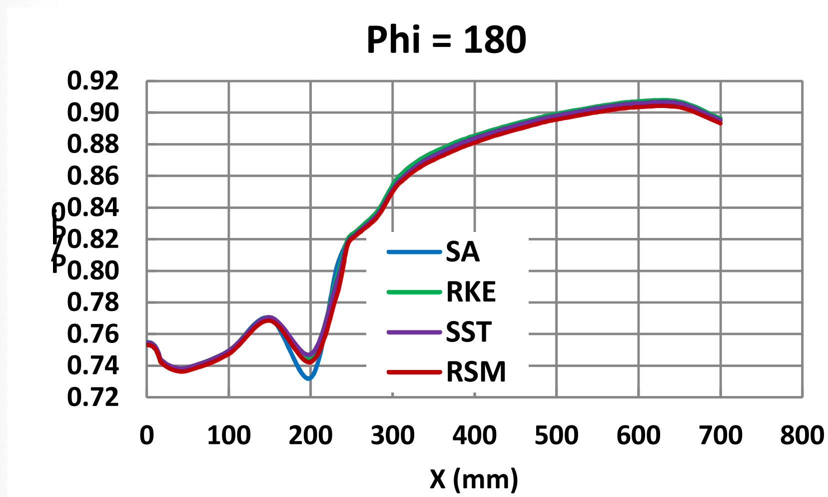
Turbulence Model Comparison

Medium Mesh

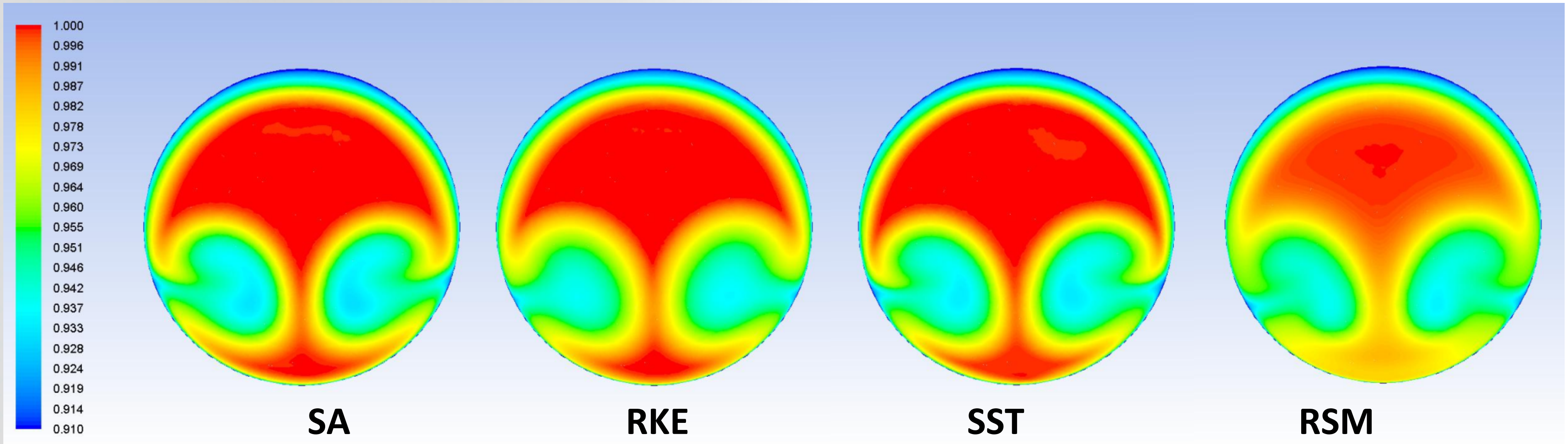






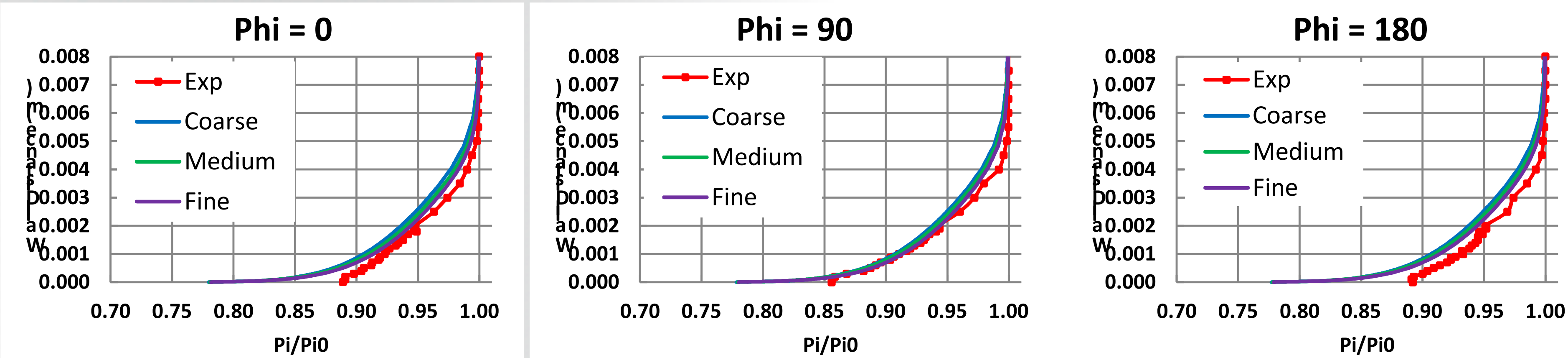


Turbulence Model Comparison
Medium Mesh

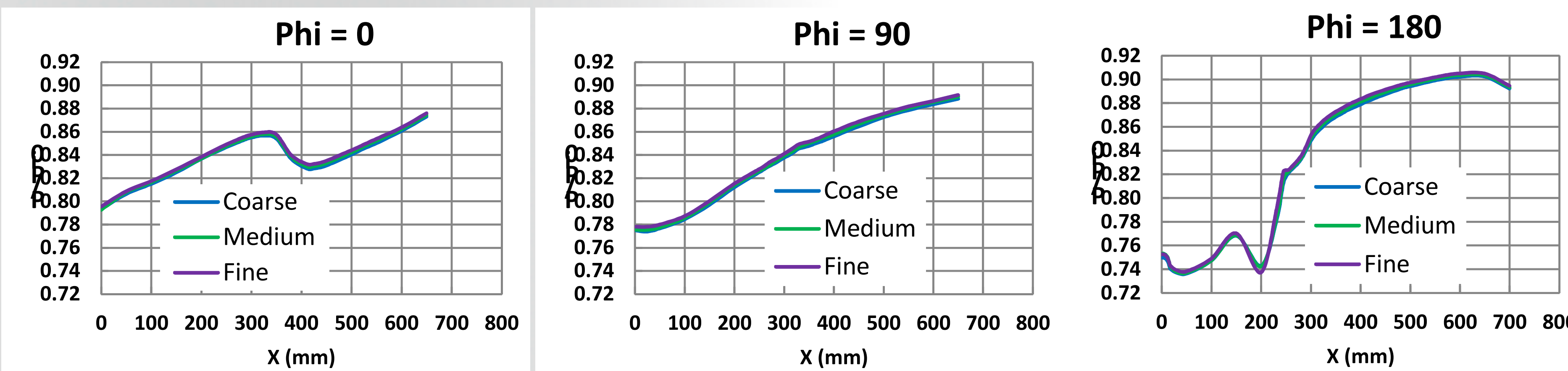


Total pressure ratio at the AIP

Mesh Independence Study – *RSM*



Boundary layer profiles at the duct entrance. Comparison with experiment



Streamwise pressure distributions along duct walls

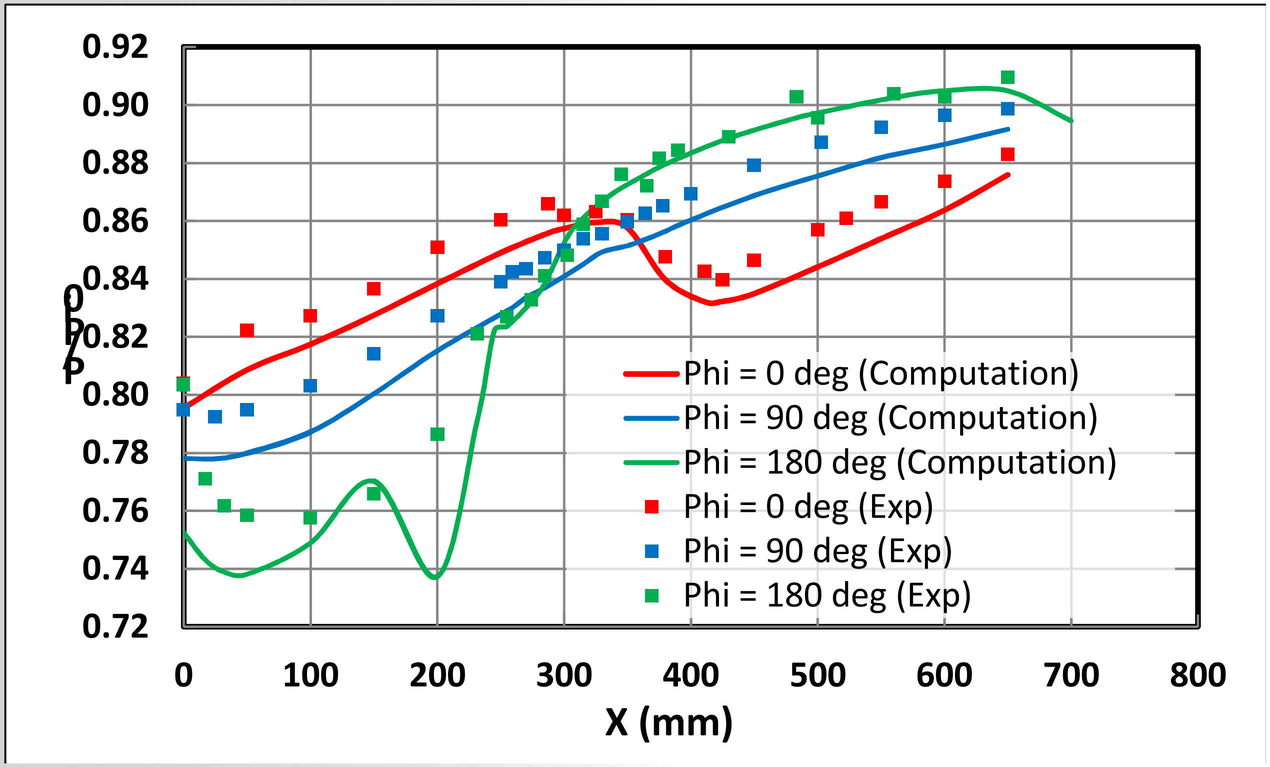
  September 29, 2014

Observations – *Model with VGs*

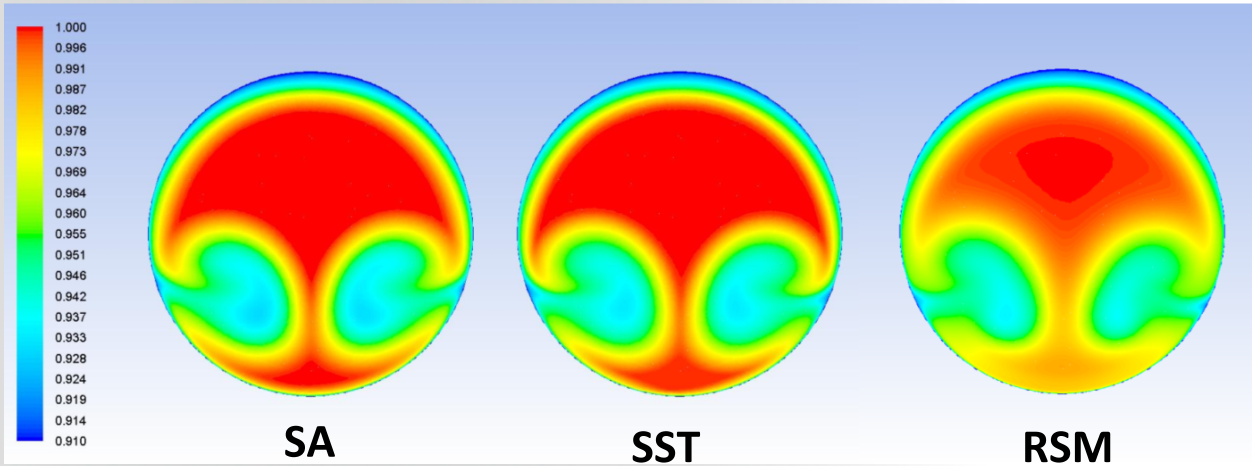
Models	SA			RKE		SST			RSM		
Mesh	Coarse	Medium	Fine	Coarse	Medium	Coarse	Medium	Fine	Coarse	Medium	Fine
PR	**0.9765**	**0.9772**	**0.9769**	**0.9745**	**0.9770**	**0.9765**	**0.9770**	**0.9770**	**0.9716**	**0.9727**	**0.9739**
DC	**0.0216**	**0.0227**	**0.0220**	**0.0197**	**0.0209**	**0.0233**	**0.0231**	**0.0230**	**0.0168**	**0.0167**	**0.0167**

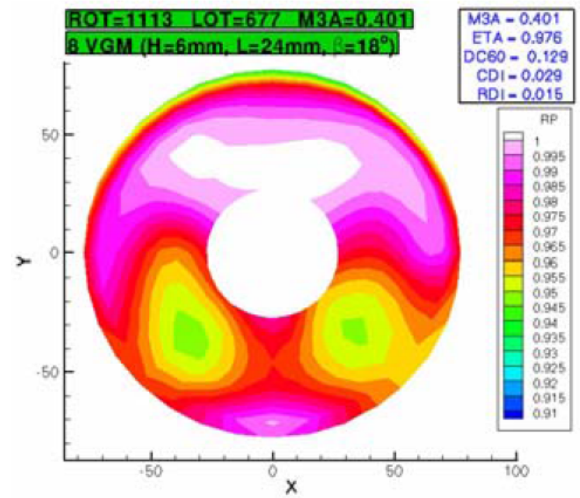
- All models predicted similar surface static pressure trends
- Predictions of static pressure along the wall in the region where VGs installed are slightly different between turbulence models
- Even the coarse mesh appears to give a mesh independent solution with RSM
- It is expected that the results from RSM should be closer to experimental measurement as RSM captures turbulence anisotropy

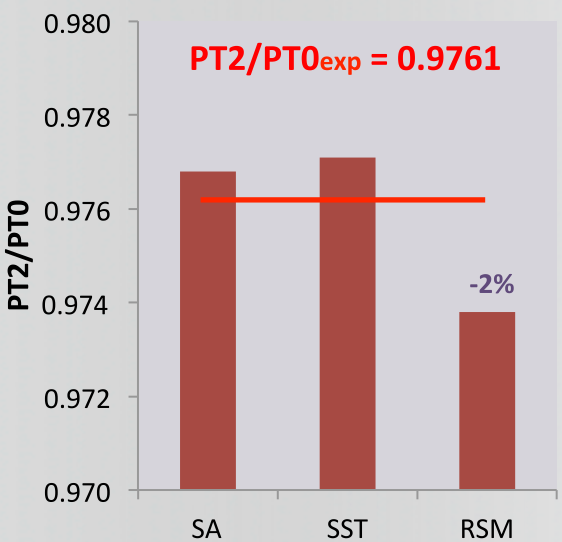
Comparison with Experiment – *RSM, Fine mesh*

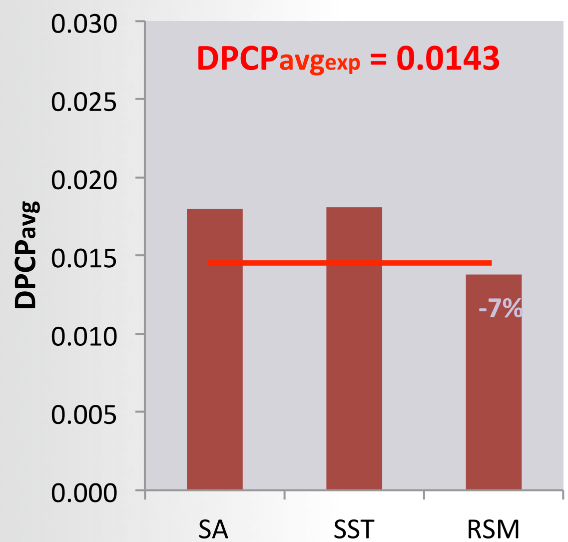


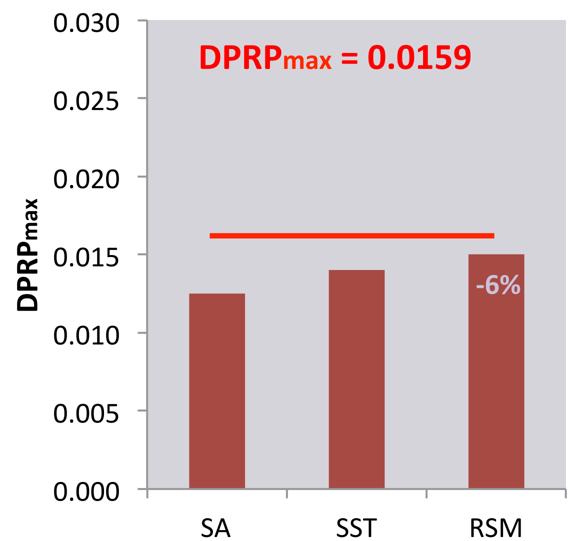
Comparison with Experiment – *RSM, Fine mesh*



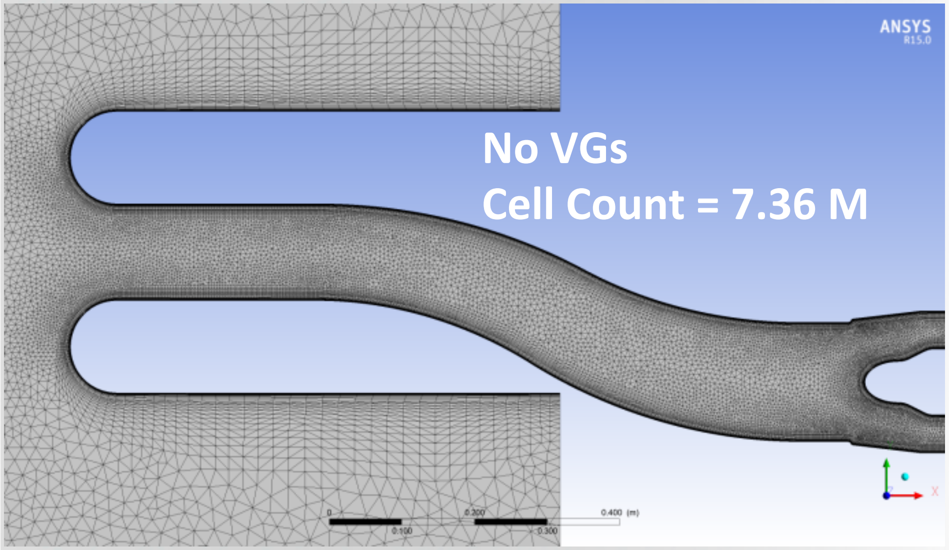


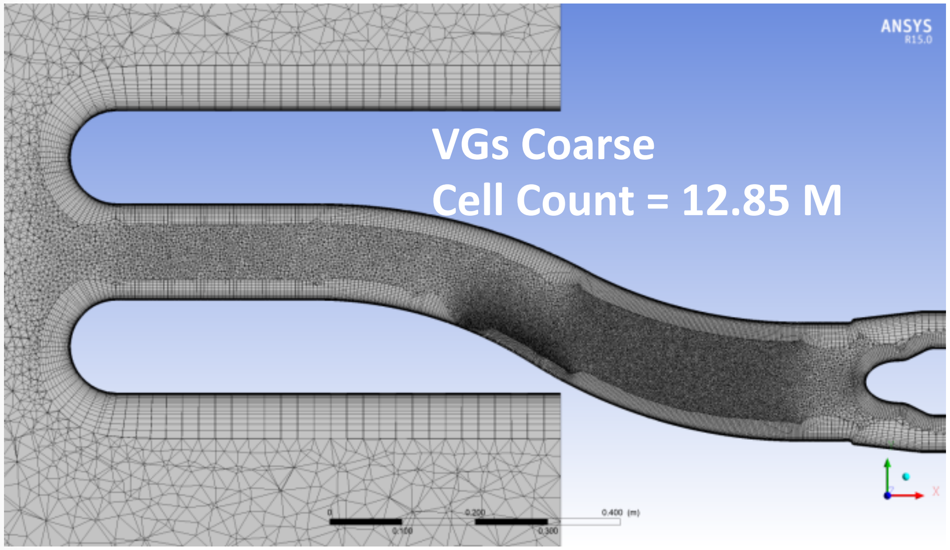




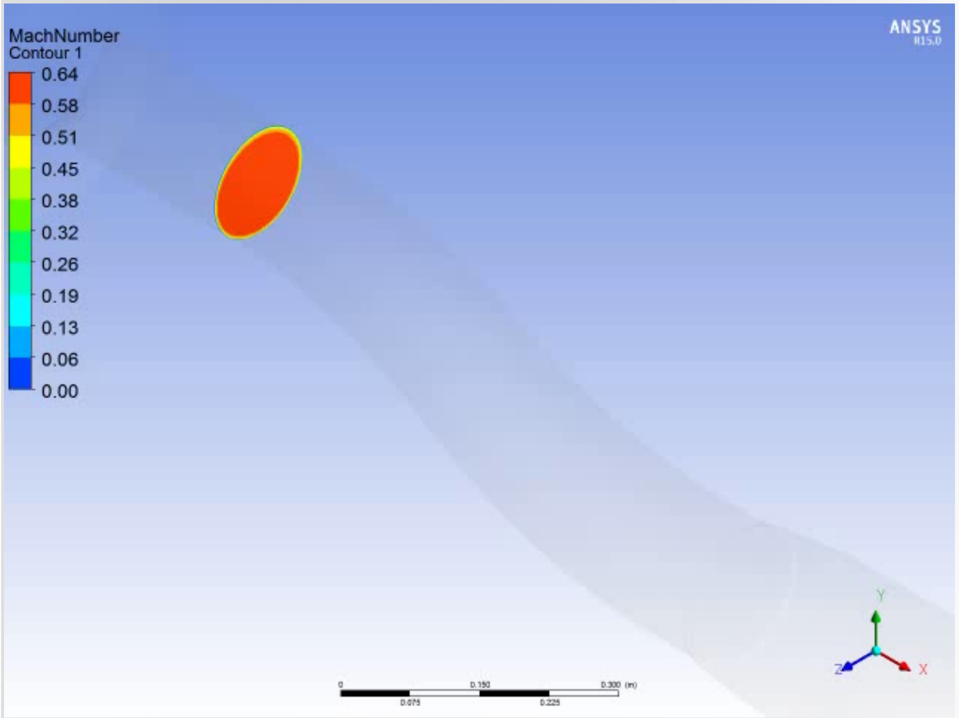


Results – Comparison with and without VGs

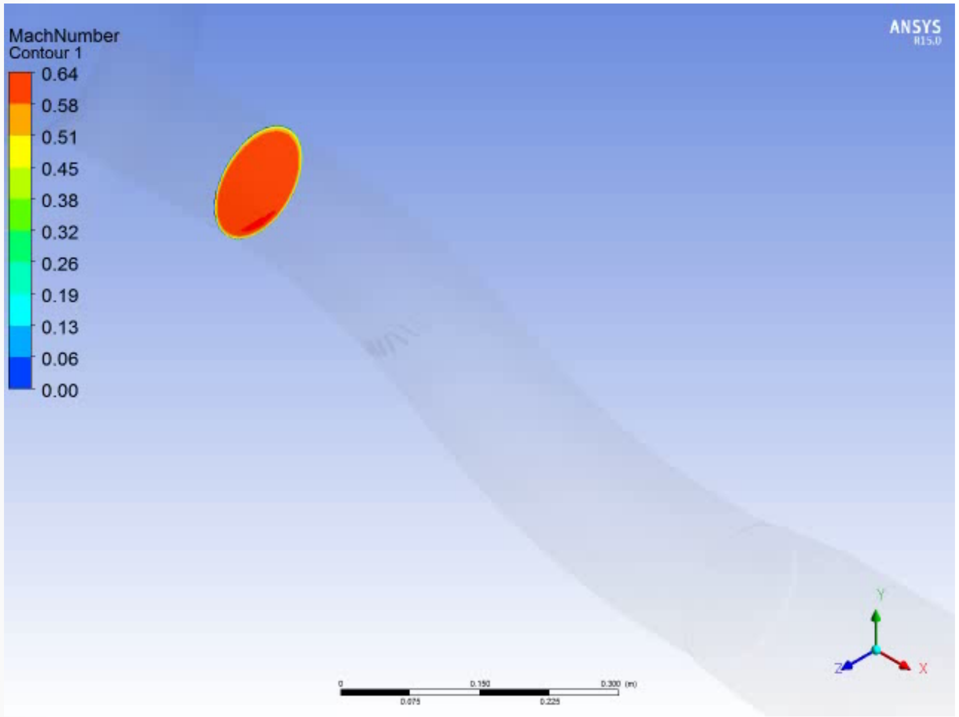




Effect of Vortex Generators

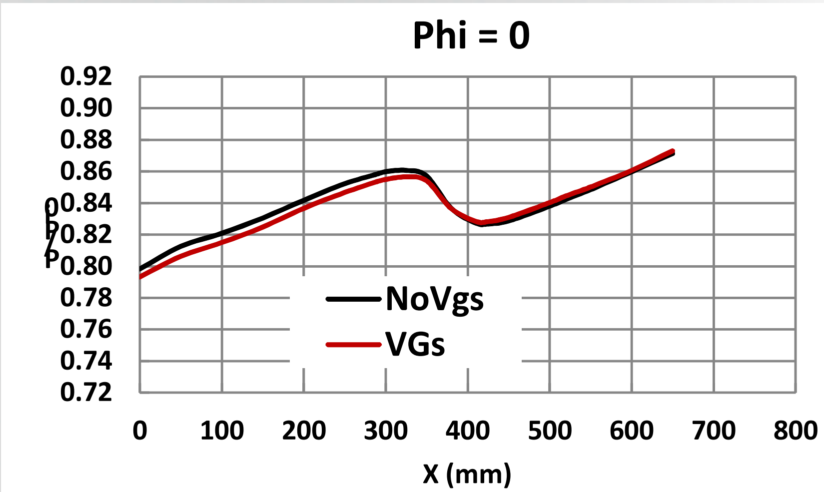


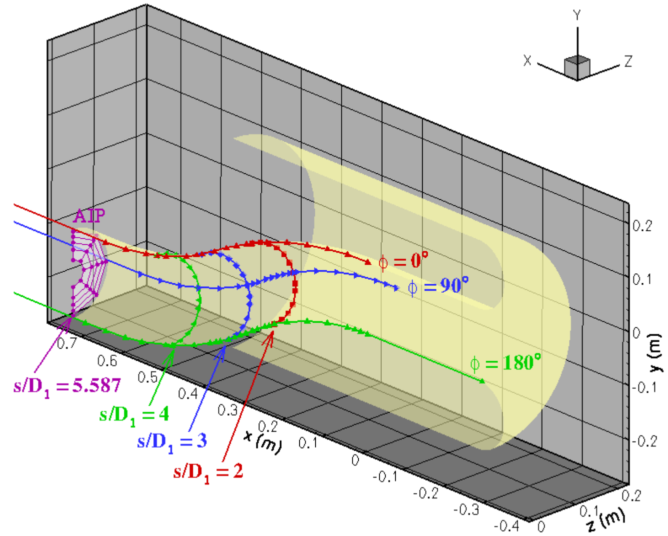
Baseline

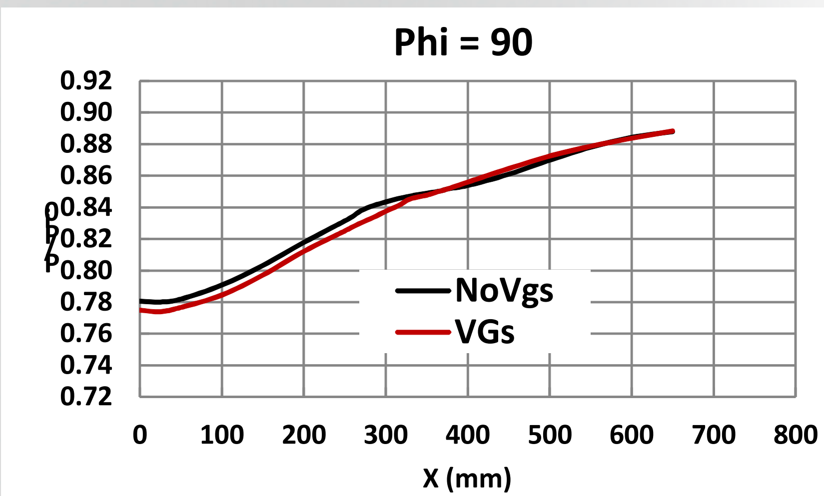


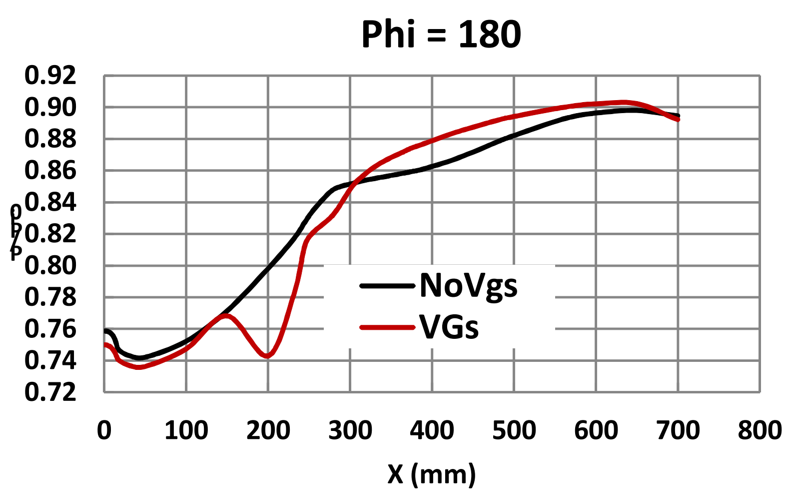
With VGs

Effect of Vortex Generators - RSM



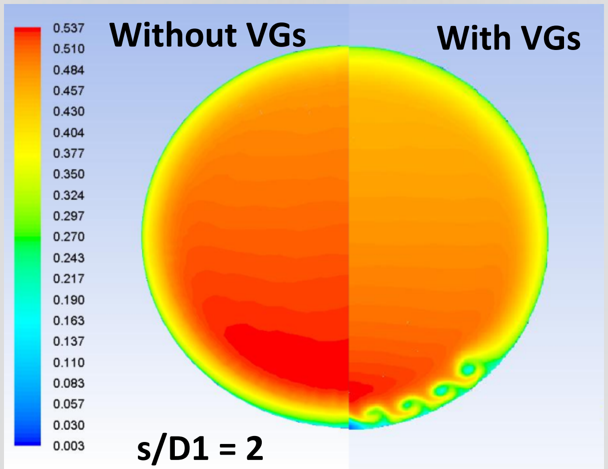


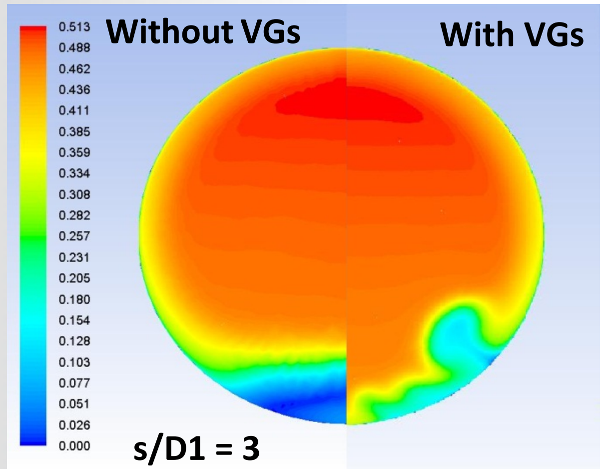


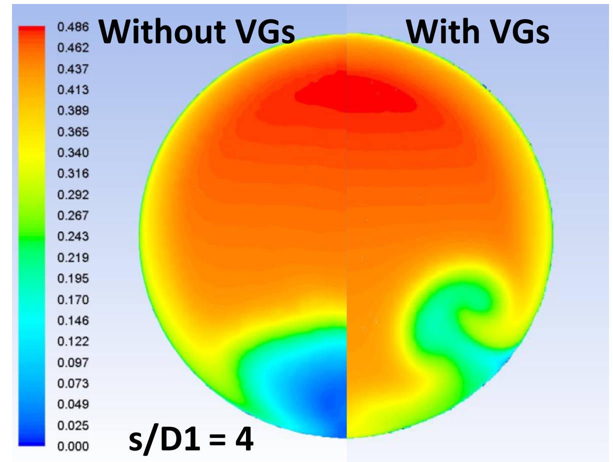


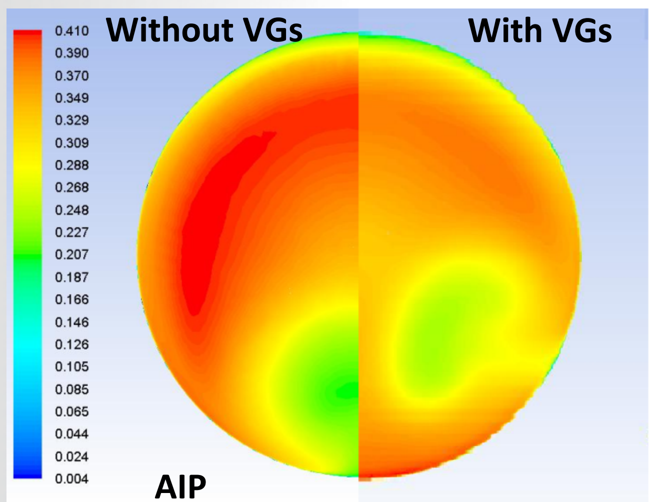
Effect of Vortex Generators - RSM

Mach Number









Observations

Mesh	Without VGs				With VGs			
Models	SA	RKE	SST	RSM	SA	RKE	SST	RSM
PR	0.9728	0.9747	0.9724	0.9695	0.9765	0.9745	0.9765	0.9716
DC	0.0276	0.0311	0.0317	0.0252	0.0216	0.0197	0.0233	0.0168

- The separation is significantly reduced by the VGs
- Large difference in the static pressure distribution (particularly along phi = 180 line) between the cases with and without VGs
- Pressure recovery does not change much after installing VGs
- Distortion Coefficient is reduced by the VGs

Summary

- Pressure-based coupled solver is a robust and effective method for simulating subsonic flow in an S-shaped intake diffuser
  - less memory and CPU intensive than traditional density-based approaches
- Reynolds Stress Model is better suited for steady-state analysis of this type of flows, as compared to one- and two-equation RANS models
- A grid-independence study is carried out to provide an additional verification of the CFD results
- Numerical predictions of total pressure recovery are in excellent agreement with experiment in both configurations without and with VGs
- Distortion Coefficient predictions are within the confidence interval of experimental data
- 2nd AIAA Propulsion Aerodynamics Workshop summary presentation highlighted ANSYS Fluent RSM predictions as one of the most accurate among participating CFD codes

ANSYS

On October 28th we have a NASA Tech Briefs webinar with a guest presentation by SpaceX

TECH BRIEFS MEDIA GROUP
an SAE International Company

SpaceX: Revolutionizing the Design of Advanced Rockets and Spacecraft through Computer Aided Engineering

Add to Calendar

Live Presentation - Tuesday, October 28, 2014 • 1:00 PM ET

This presentation discusses how SpaceX uses Finite Element Analysis and Computational Fluid Dynamics to revolutionize the rocket and spacecraft industry. It touches upon how Falcon 9 v1.1 and Dragon engine components were designed at faster than industry rates via multi-physics analysis, which includes structural, dynamic, thermal, fluid, and electromagnetic analyses.

SpaceX designs, manufactures and launches advanced rockets and spacecraft. The company was founded in 2002 to revolutionize space technology, with the ultimate goal of enabling people to live on other planets.

SpaceX has gained worldwide attention for a series of historic milestones. It is the only private company ever to return a spacecraft from low-Earth orbit, which it first accomplished in December 2010. The company made history again in May 2012 when its Dragon spacecraft attached to the International Space Station, exchanged cargo payloads, and returned safely to Earth — a technically challenging feat previously accomplished only by governments. Since then Dragon has delivered cargo to and from the space station multiple times, providing regular cargo resupply missions for NASA.

Speakers:

Andy Sadhwani
Senior Propulsion Analyst
Space Exploration Technologies

Andy Sadhwani is a Senior Propulsion Analyst at Space Exploration Technologies and has been involved with all SpaceX rocket engine programs to date. He holds degrees in Aerospace and Mechanical Engineering from Stanford and Carnegie Mellon and carries over 10 years of experience in finite element analysis and computational fluid dynamics.

October 28th 1pm Eastern US time

Topics to be covered include:

- CFD for drag and pressure drop
- Rapid geometry updates and parametric studies.
- Ensuring NASA margins of safety were met.
- Mechanical static & transient structural, static & transient thermal, random vibration, and harmonic analyses were used to provide a holistic view of structural & thermal margins.
- Ease of use for designers and quickly training new users

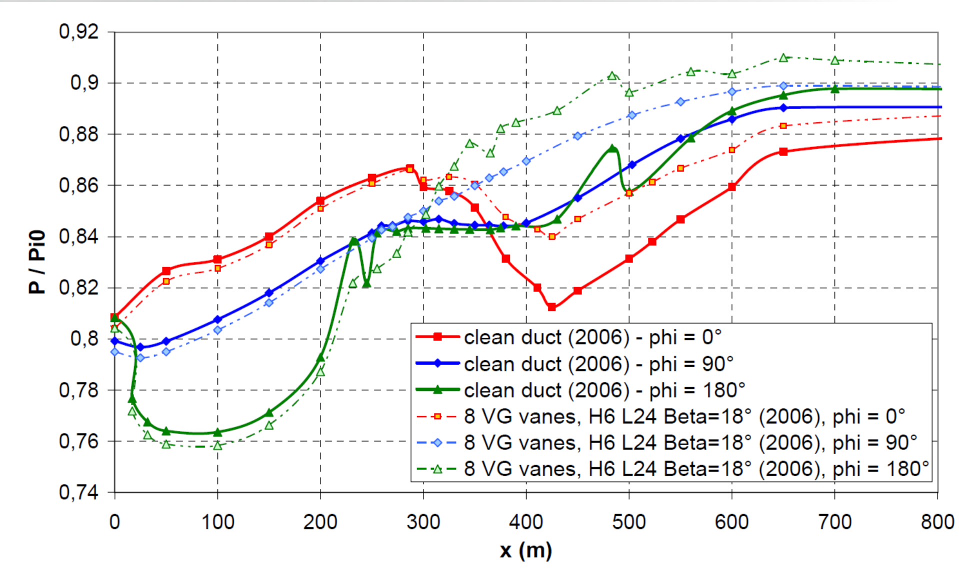
Register to watch for free at:

www.ansys.com/spacexweb

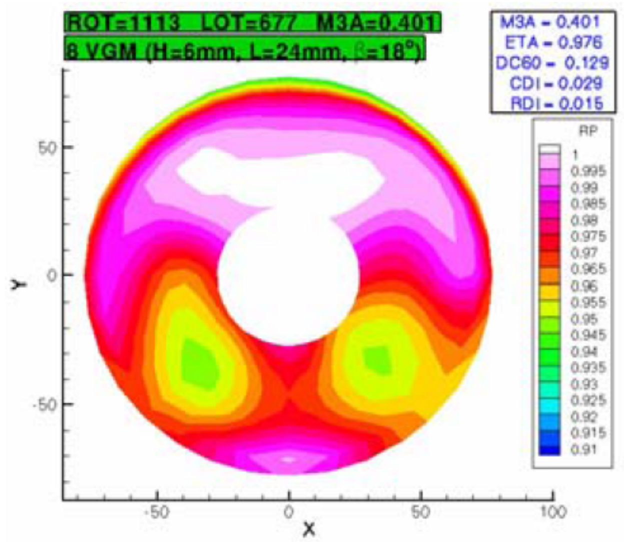
Thank you!!

Appendix

Experimental Comparison



Static Pressure Distribution



Total Pressure Ratio. AIP

