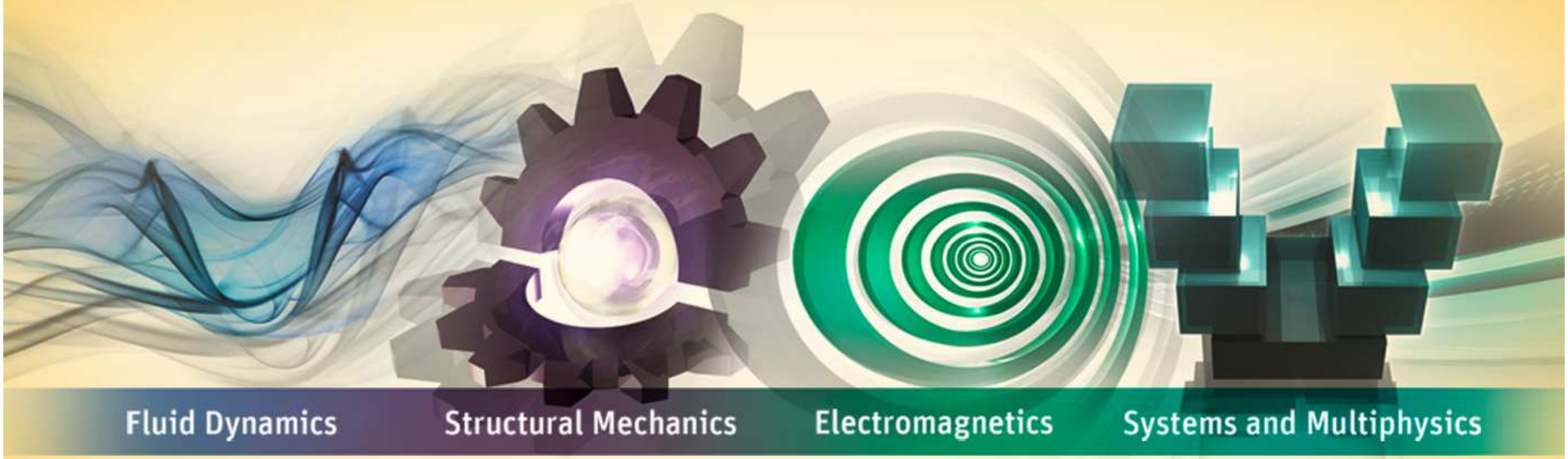
Realize Your Product Promise™

Turbulence Modeling



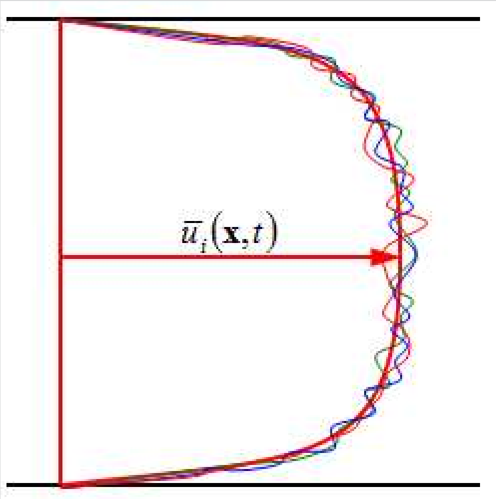
Gilles Eggenspieler, Ph.D.
Senior Product Manager

Overview

- **The Role of Steady State (RANS) Turbulence Modeling**
- **Overview of Reynolds-Averaged Navier Stokes (RANS) Modeling Capabilities in ANSYS CFD**
  - Model overview
  - Wall treatment
  - Model extensions and other interesting new features

Motivation for Steady State Turbulence Modeling

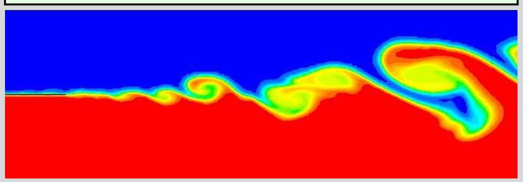
- **The majority of all flows of engineering interest are turbulent**
- **The motion of eddies in a turbulent flow is inherently unsteady and three-dimensional**
  - Even if the flow is steady in a mean flow sense
- **Steady state simulations are preferred for many engineering applications because they are easier**
  - Shorter simulation time
  - Simplified post-processing
  - In many cases, only time-averaged values are of interest
- **Turbulence models that allow steady state simulations to be performed for turbulent flows are therefore desirable and important**



Turbulent Flow Simulation Methods

DNS

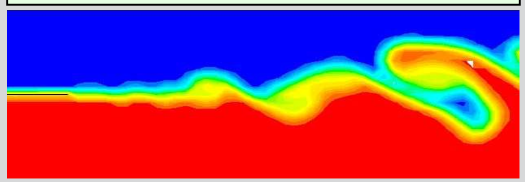
(Direct Numerical Simulation)



- Numerically solving the full unsteady Navier-Stokes equations
- No modeling is required
- A research tool only– far too much information for industrial applications

SRS

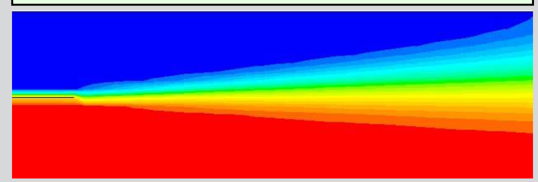
(Scale Resolving Simulations)



- Includes Large Eddy Simulation (LES)
- The motion of the largest eddies is directly resolved in the calculation, in at least a portion of the domain, but eddies smaller than the mesh are modeled
- Inherently unsteady method

RANS

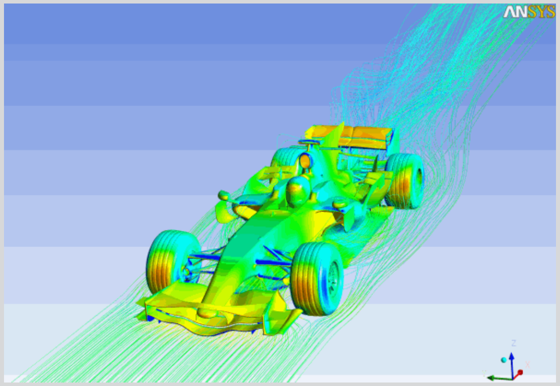
(Reynolds Averaged Navier-Stokes Simulations)



- Solve Reynolds-averaged Navier-Stokes equations (time-average)
- Steady state solutions are possible
- All turbulence is modeled. Larger eddies are not resolved
- RANS turbulence models are the only modeling approach for steady state simulation of turbulent flows
- This is the most widely used approach for industrial flows

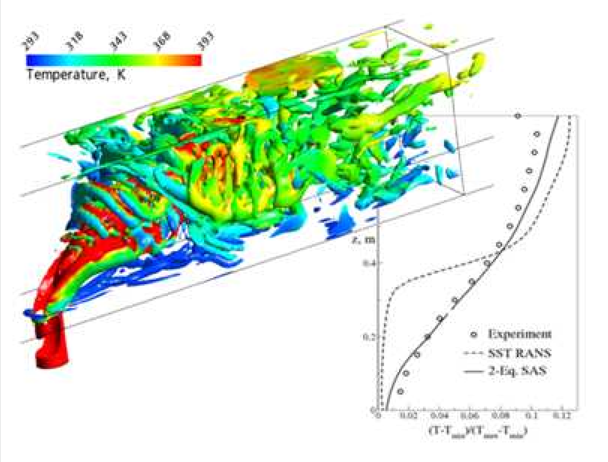
Comparison of SRS and RANS

- **RANS**
  - Advantages: For many applications, steady state solutions are preferable, and for many applications a good RANS model with a good quality grid will provide all the required accuracy
  - Disadvantages: For some flows, challenges associated with RANS modeling can limit the level of accuracy that it is possible to attain



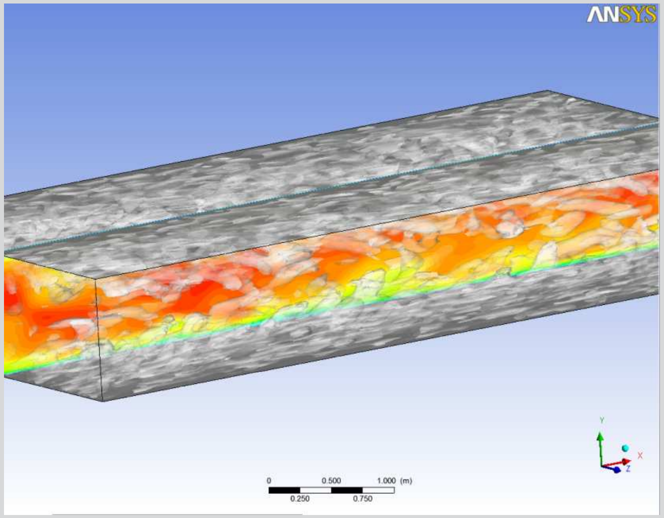
Courtesy of Red Bull Racing

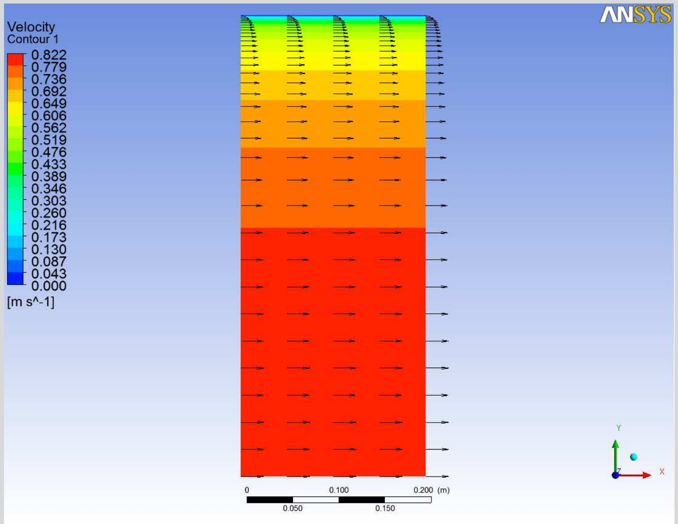
- **SRS**
  - Advantages: Potential for improved accuracy when the resolution of the largest eddies is important or when unsteady data is needed
  - Disadvantages: computationally expensive
    - Higher grid resolution required
    - Unsteady simulation with small time steps generates long run times and large volumes of data



Computational Expense: SRS vs. RANS in Wall-Bounded Flow

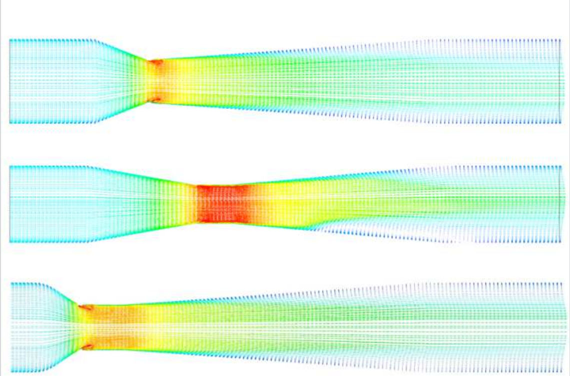
- **Example: Channel flow at Re = 114,000**
  - Boundary layer thickness, δ, equal to channel half-width
- **Top: WMLES**
  - 1.2 million cells, transient calculation, run time is order of days
- **Below: RANS**
  - 140 cells, steady calculation, run time is order of minutes
- **Important**
  - For wall-bounded flows, in a more typical 3D industrial geometry, RANS would still be 2 orders of magnitude fewer cells and run times of hours versus days.



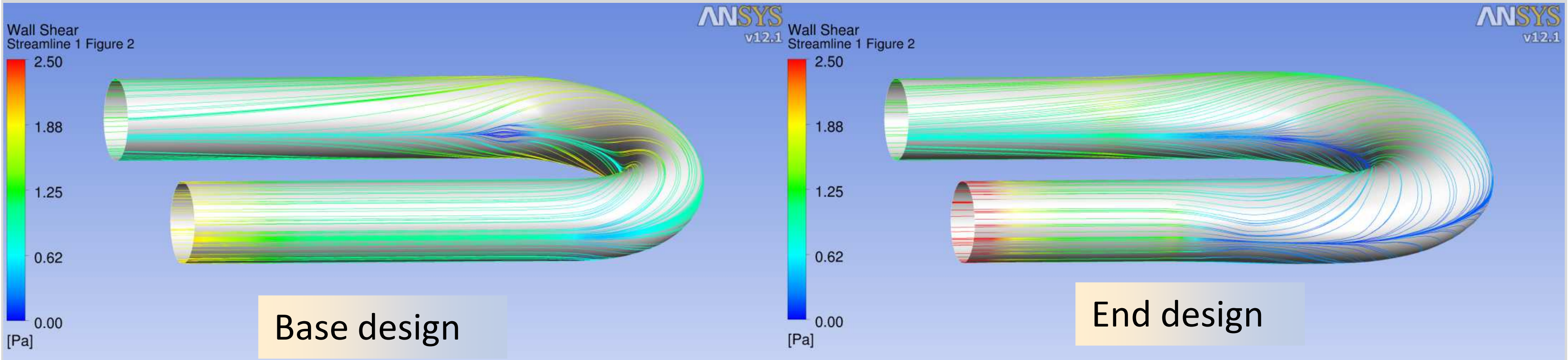


Steady RANS

- **Steady state RANS calculations will remain an important modeling practice for years to come**
  - Model the entire system versus modeling the component
  - Increase the number of simulated design points in optimization/parametric studies
- **Providing state-of-the-art RANS modeling capabilities remains an important focus of ANSYS development**



Parametric study of racecar engine intake restrictor design with SST model. Courtesy of University of Waterloo Formula Motorsports

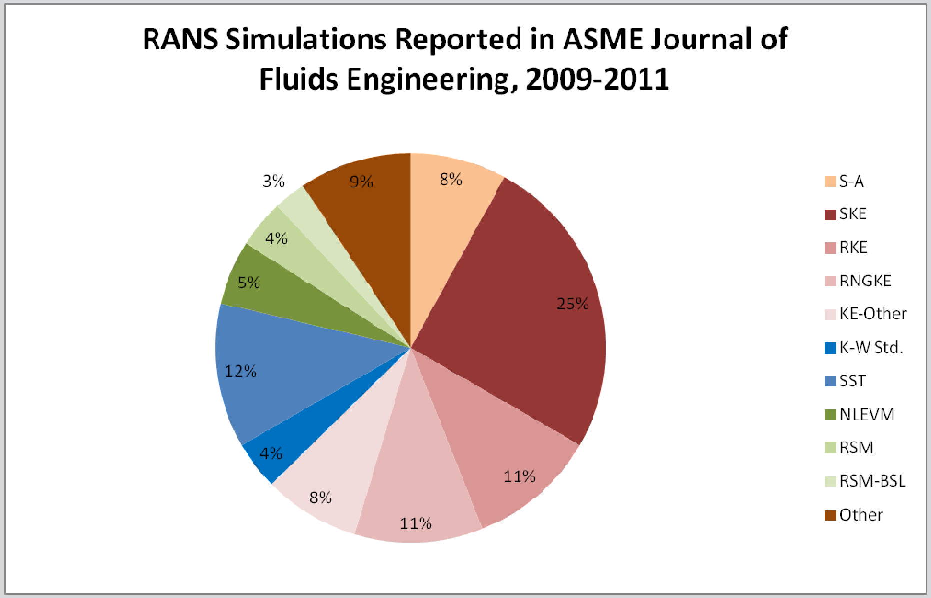


Example: Optimization study (with adjoint solver and realizable k-ε model) achieves 1/3 reduction in pressure drop in u-bend over 30 different design iterations

RANS Capabilities in ANSYS CFD

- **Models and Boundary Treatments**
- **Model Extensions**

What RANS Models are People Using?



- **Informal survey of single phase RANS model usage based on papers published in the Journal of Fluids Engineering during 2009 – 2011**
- **The CFD user community requires a broad range of models to choose from in order to meet its needs**
  - Over 2/3 of all simulations reported using some variation of 1 or 2 equation model (S-A, k-ε family, k-ω family)
  - In some applications, one model may be more dominant than others (example: aerodynamics & SST, cyclones & RSM), but for a broad range of applications, a variety of models is needed to match the appropriate model to the appropriate application

Steady RANS Turbulence Models in ANSYS

- **A wide array of models is available for steady state calculations**
  - Includes all commonly used models in CFD modeling
  - Includes useful extensions to the models such as curvature correction and EARSM
  - Important to be able to ensure whatever the application, you can choose the most suitable model
  - There is also a long list of LES/DES/SAS Hybrid Models that will be covered in later sections of the seminar

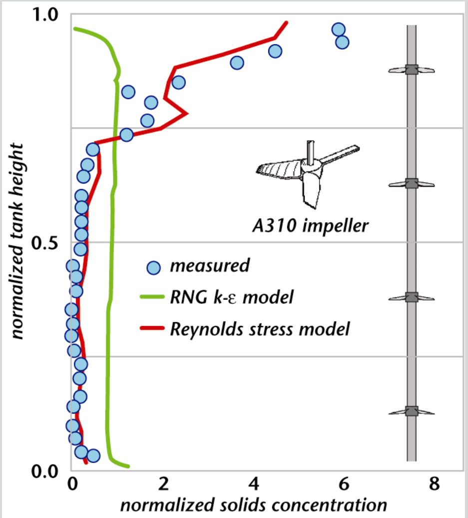
One-Equation Models Spalart-Allmaras $(k\text{-}\varepsilon)_{1E}$
Two-Equation Models k–ε (Standard, Realizable, RNG) k–ω (Standard, SST)
Curvature Correction (all 1 & 2 eqn. models) V2F (4 eqn.)*
Explicit Algebraic Reynolds Stress Model (EARSM) Reynolds Stress Models Launder-Reece-Rodi, Speziale-Sarkar-Gatski Stress-ω
k–kl–ω Transition Model SST Transition Model

* additional license required

ANSYS Models

- **It is not enough just to provide many choices**
- **More importantly, for the models that are available, emphasis is placed on**
  - Correct implementation
    - Models should be well understood and tested
  - Accurate and validated for some class(es) of applications
  - Robust performance on all mesh topologies
  - Interoperability with other physical models, e.g. multiphase, dynamic mesh, ….
  - Wall treatment

Example: Solids suspension in an tall, unbaffled tank. Reynolds stress model together with Eulerian granular multiphase model



Courtesy of the University of Bologna

Separation Prediction with the SST Model

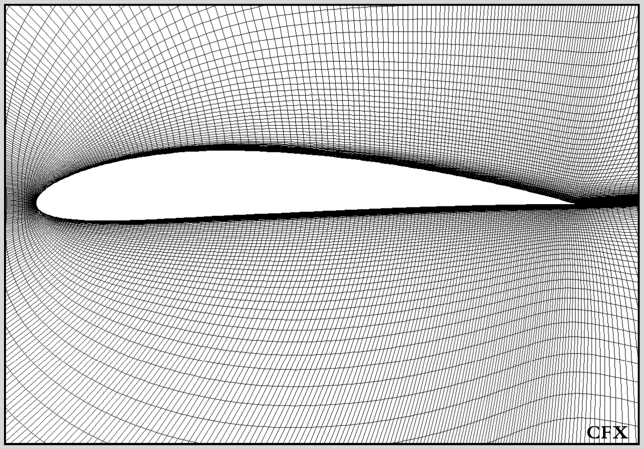
Separation is important for prediction of:

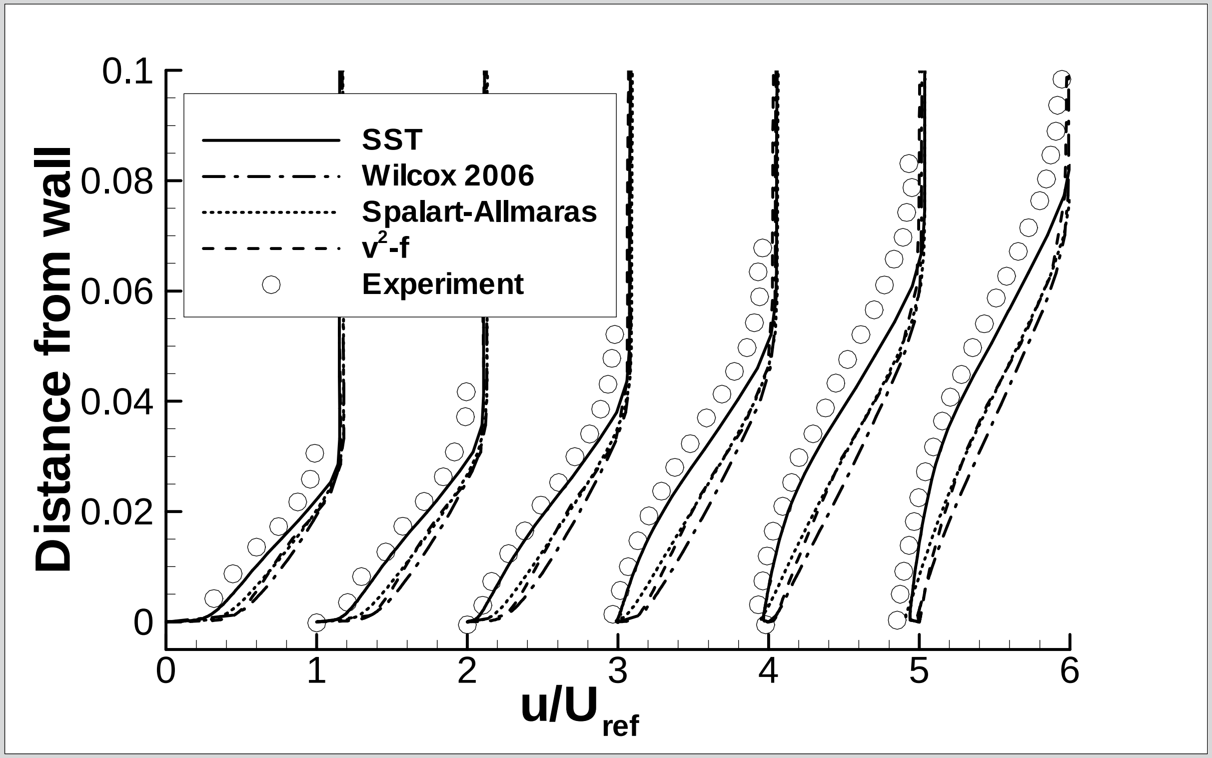
- Pressure losses in diffusers
- Stall prediction of airfoils and wings
- Prediction of performance characteristics of turbomachinery components

Motivation for SST model:

- Historically standard two-equation models miss the separation and predict attached flow even for strong pressure gradient flows
- SST model is one of the most accurate two-equation models for separation prediction.

NACA 4412 Airfoil





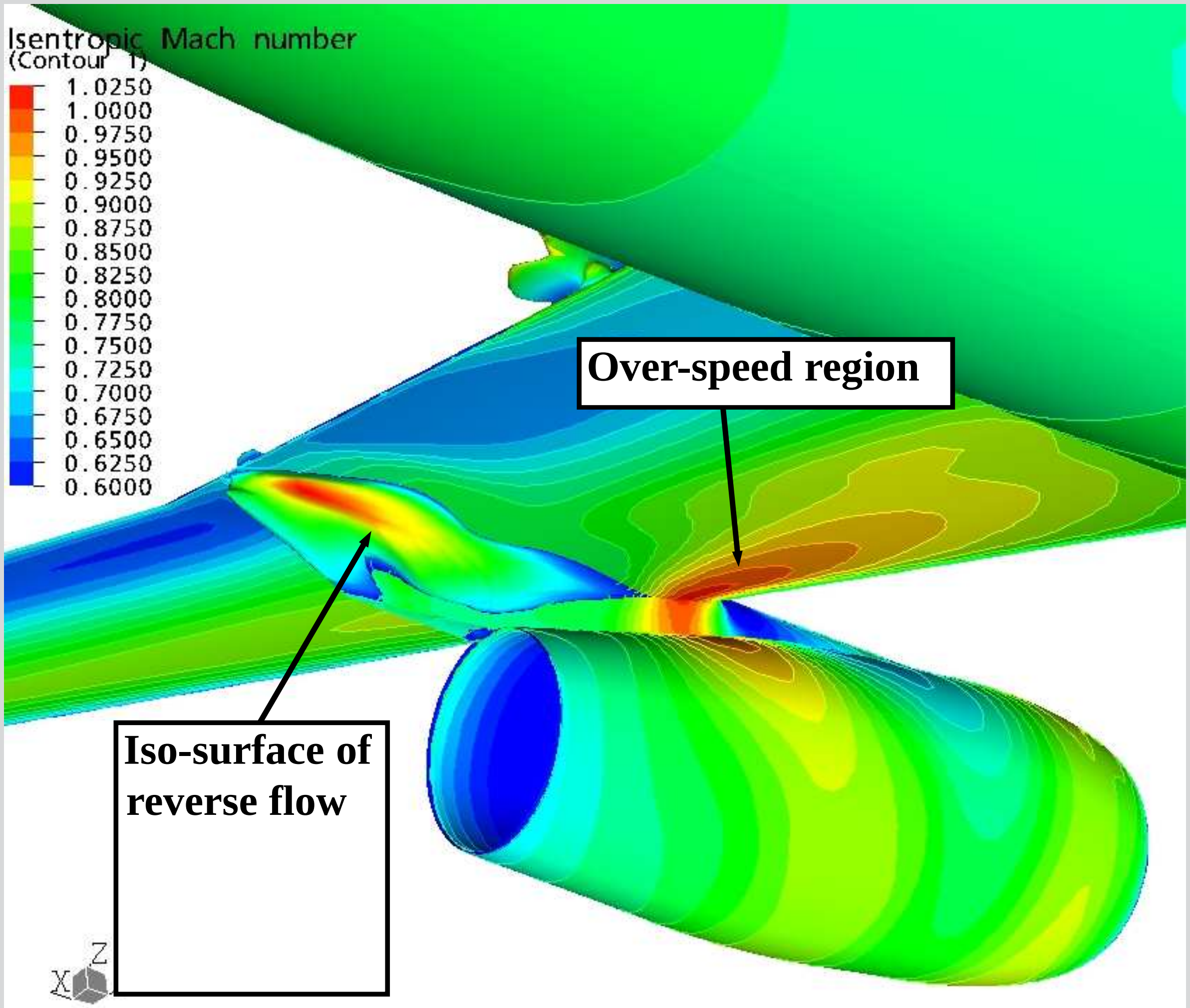
- **SST model in comparison with separated velocity profiles compared to Wilcox 2006, V2F and Spalart-Allmaras (SA) model**

AIAA Drag Prediction Workshop 2003

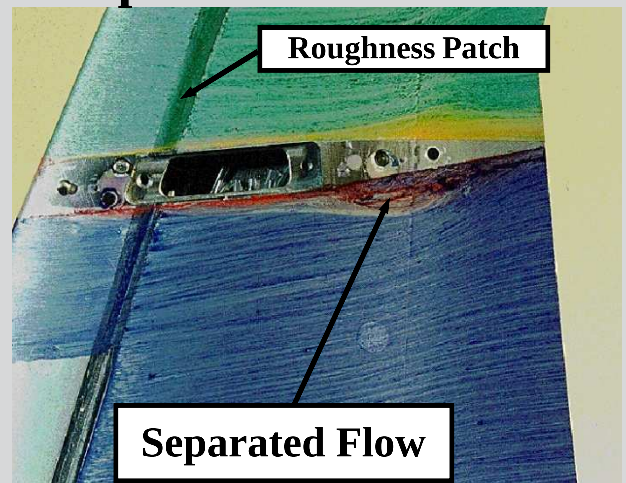
- Workshop for comparison of CFD codes for simulation of lift and drag of airplane configurations
- Simulation of installation drag of engine nacelle
- Comparison of 18 different contributions mainly from aeronautical research centers and companies.
- Comparison with experimental data for DLR-F6 wing-body and wing-body-pylon-nacelle configuration
- http://aaac.larc.nasa.gov/tsab/cfdlarc/aiaa-dpw/Workshop2/workshop2.html

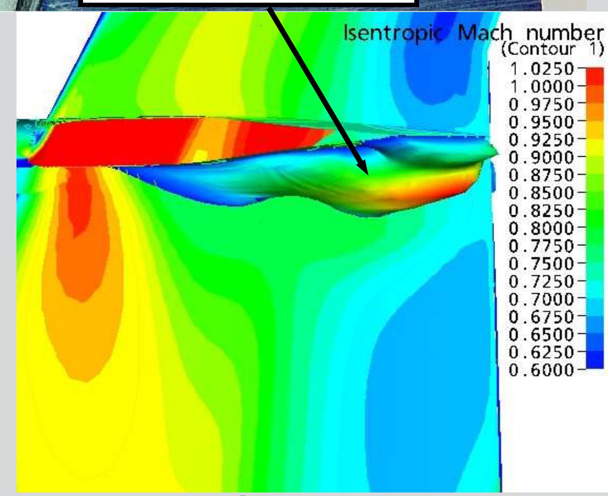
Part of this work was supported by research grants from the European Union under the FLOMANIA project

Lower Surface Flow Visualization



Experimental Oil Flow

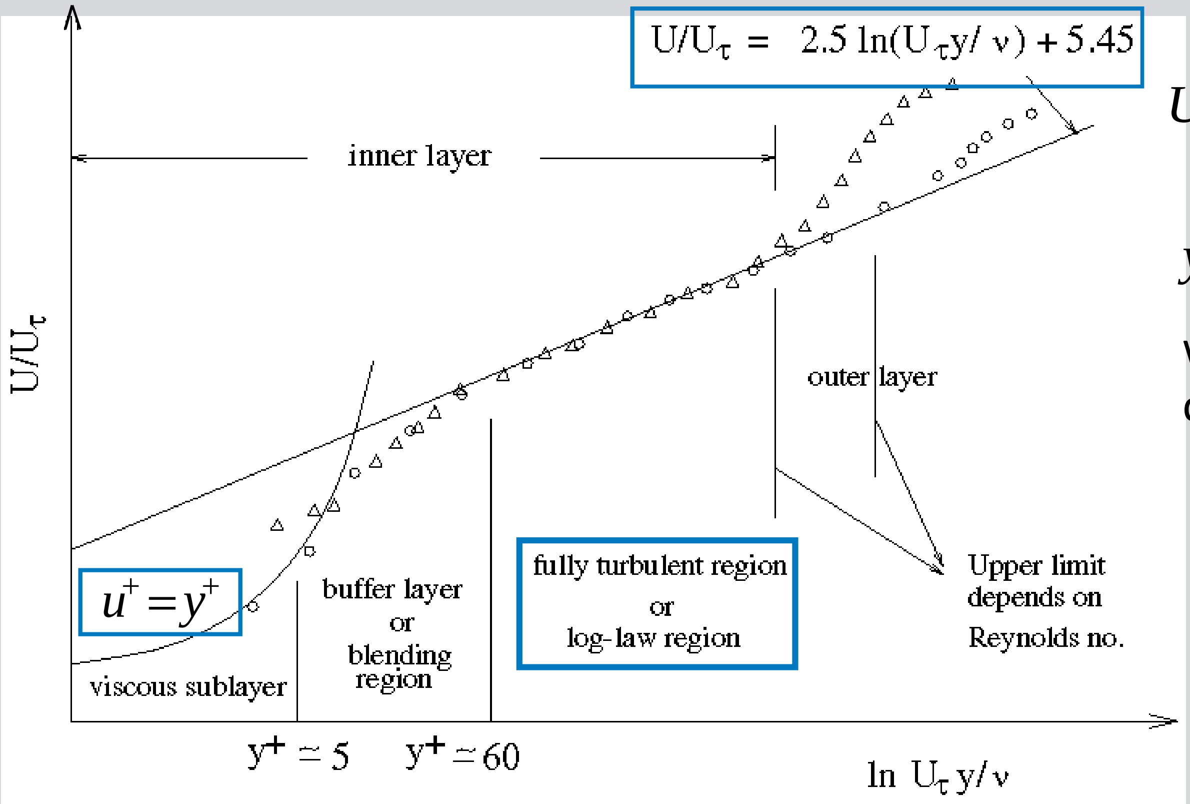




CFX 5

Near Wall Turbulence and the Law of the Wall

- **The law of the wall describes the relationship between the velocity profile and wall shear in turbulent boundary layers**
- **Close to the wall, in the inner part of the boundary layer, with the appropriate normalization, there is a universal velocity profile**
- **This universal behavior forms the basis for near wall modeling in RANS**



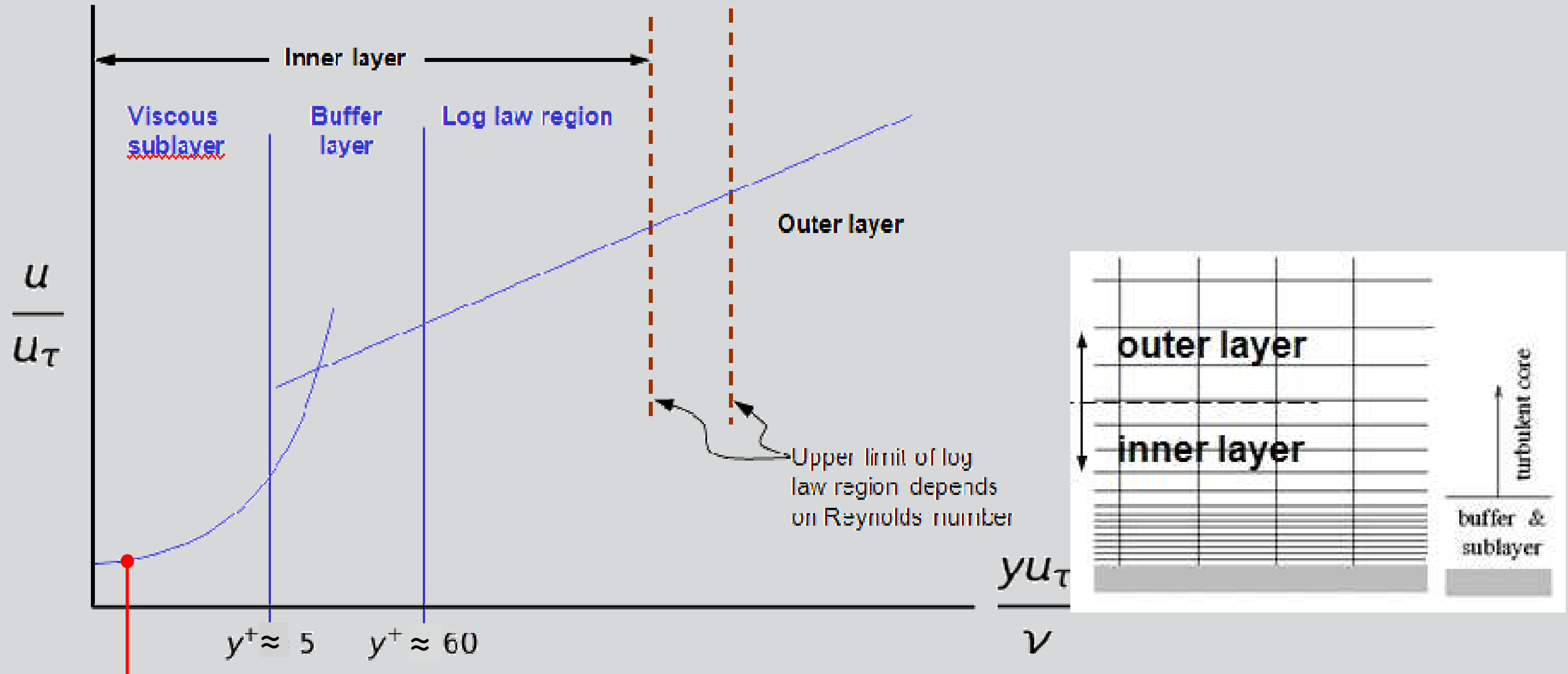
$$U_\tau = \sqrt{\frac{\tau_w}{\rho}}$$ ← Wall shear stress

$$y^+ = \frac{y U_\tau}{\nu} \qquad u^+ = \frac{u}{U_\tau}$$

where *y* is the normal distance from the wall

Viscous Sublayer Modeling Approach

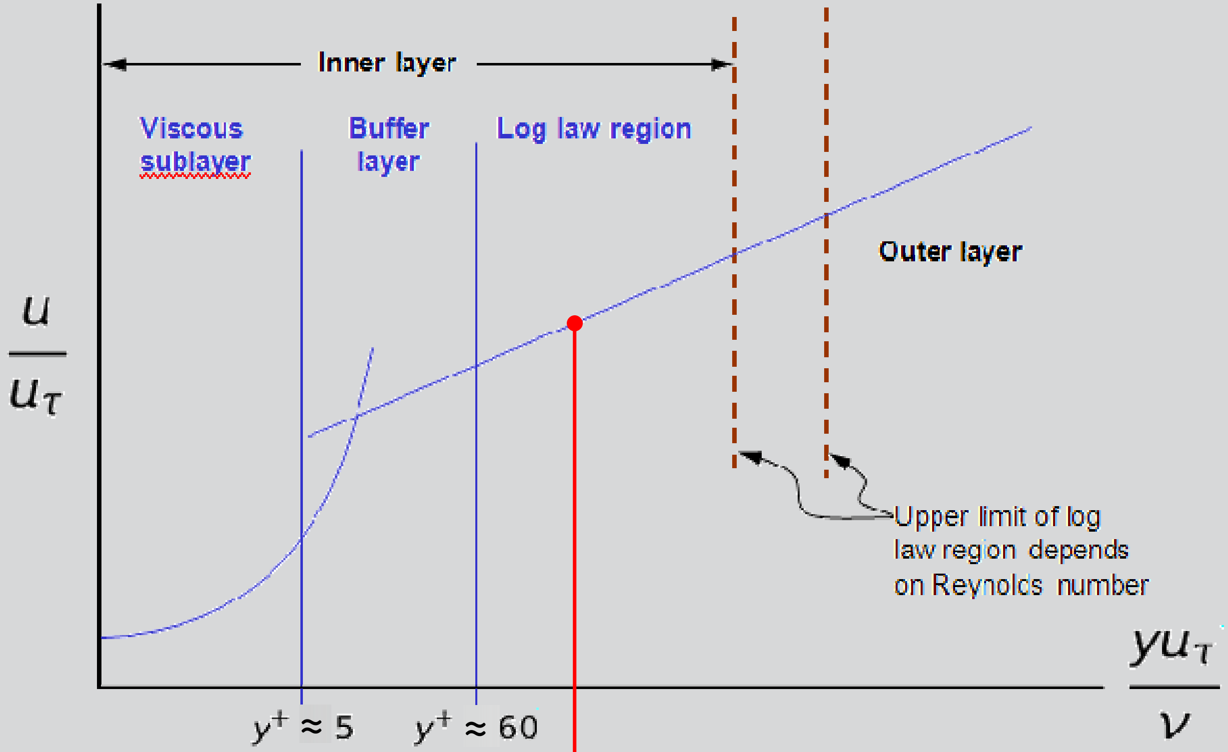
- **Used in cases where meshes that resolve the viscous sublayer can be afforded or are absolutely necessary (flow separation, laminar-turbulent transition, heat transfer…)**

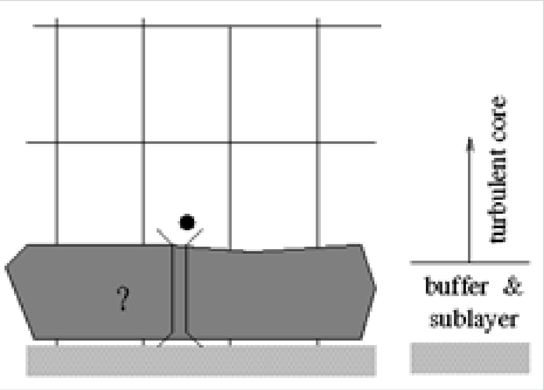


1st cell centroid at $y^+ \sim 1$, moderate grid stretching to ensure there are enough grid points across the entire boundary layer

Wall Function Modeling Approach

- **Cases where high near-wall resolution is unaffordable. Wall functions bridge the gap between the wall and the log region where the first cell centroid is located**





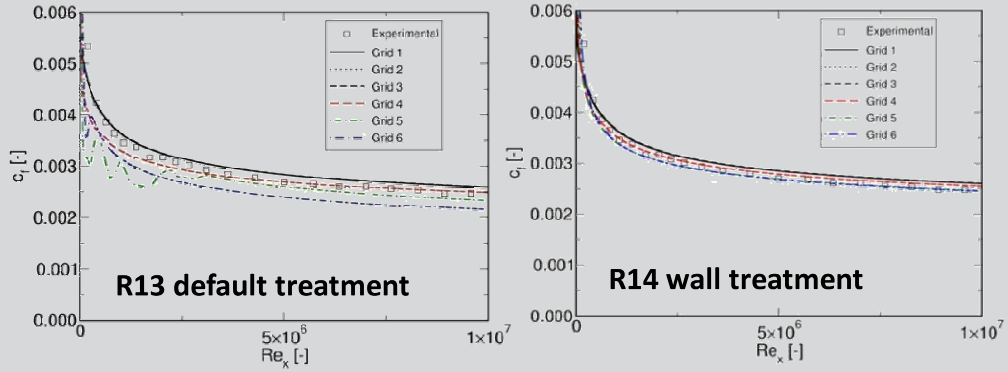
1st cell centroid located in log law region

The importance of y^+ insensitive wall treatment

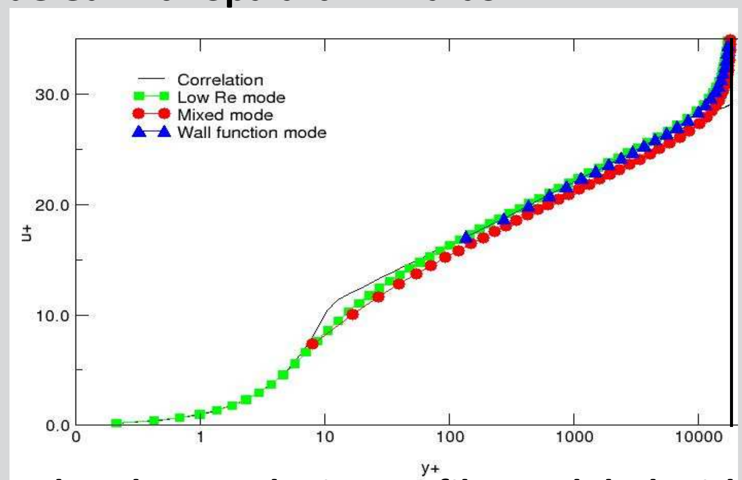
- In practice, maintaining a prescribed value of y+ in wall-adjacent cells throughout the domain for industrial cases is challenging
- Maintaining a value of y+ for the first grid point such that it is located in the log law region when using wall functions can be especially problematic when refining the grid
- Grid refinement can be a critical component of achieving a grid-independent solution, which is one of the fundamental concepts in CFD best practices, therefore y^+ insensitive wall treatments are a critical requirement for RANS models in industrial CFD

Y^+ Insensitive Treatments in ANSYS CFD

- Y^+ insensitive wall modeling treatments are available for all RANS models in ANSYS CFD
- New enhanced wall treatment for Spalart-Allmaras model in R14
- Enhanced wall treatment and scalable wall functions for k-ε family of models
- Automatic wall treatment for SST and k-ω models



Sensitivity of the skin friction coefficient to mesh density in an incompressible flat boundary layer modeled with Spalart-Allmaras



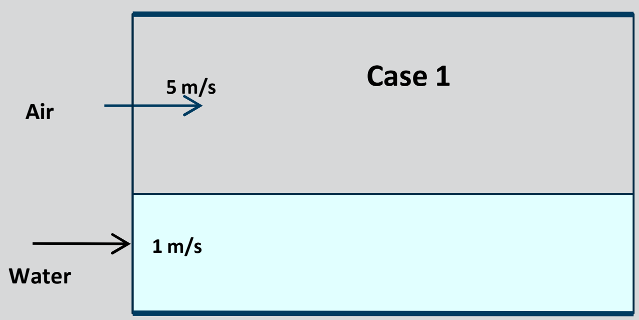
Boundary layer velocity profile modeled with standard k-ε for three different mesh densities using Enhanced Wall Treatment

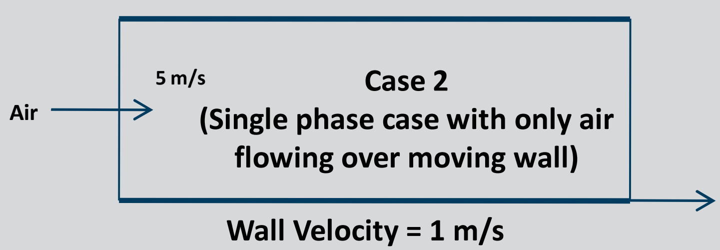
RANS Model Extensions

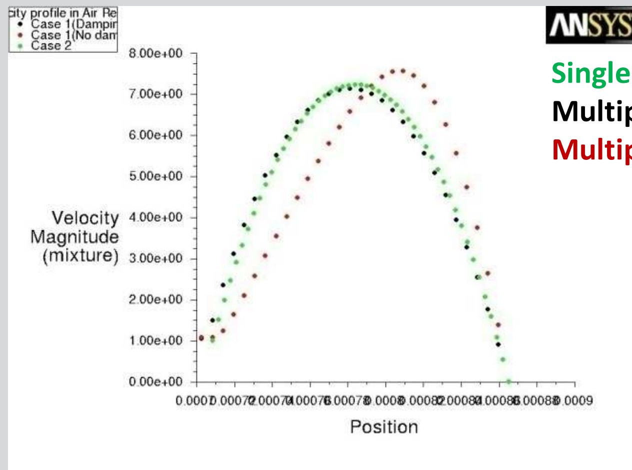
- **Turbulence Damping at Free Surface**
- **Wall Functions at Boundary of Porous Medium**
- **Curvature Correction for all 1- and 2-Equation Models**
- **Explicit Algebraic Reynolds Stress Model (EARSM)**

Turbulence Damping for Free Surface Flows

Special turbulence treatment available for SST and k-ω models accurately represents the effect of the free surface on turbulence, allowing accurate calculation of the velocity profile

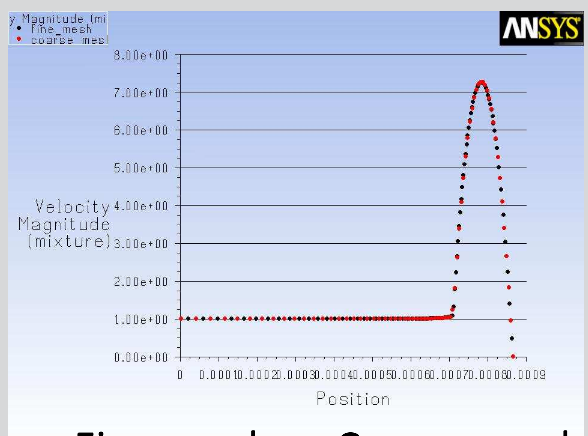






Single phase
Multiphase + damping
Multiphase + No damping

Velocity profile in air region



Fine mesh : 77520 cells

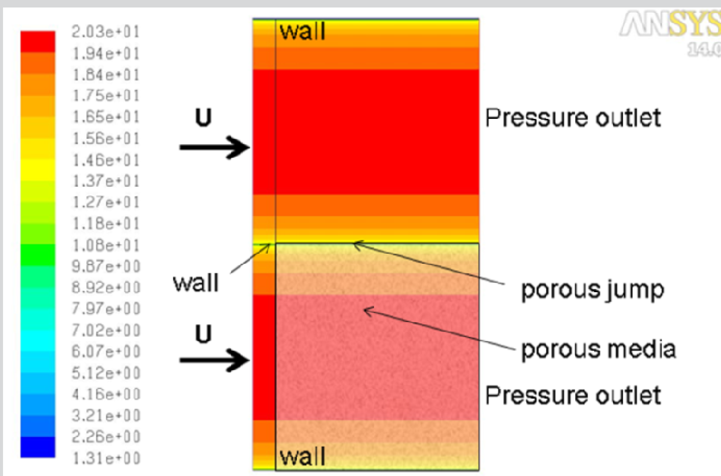
Coarse mesh: 19380 cells

Fine mesh vs Coarse mesh with turbulence damping

  May 14, 2012

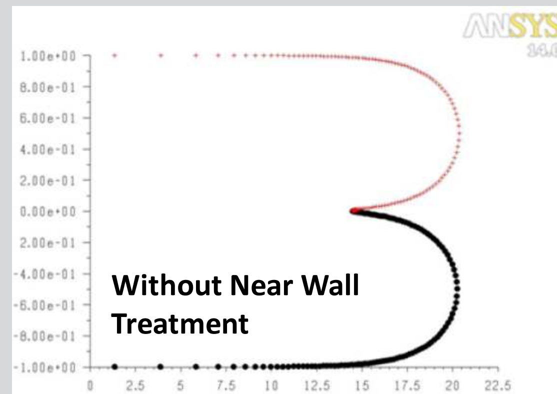
Turbulent Near Wall Treatment at Porous Medium Interface

- **Improved accuracy for turbulence near porous jump interfaces (Fluent beta feature)**
  - Use wall functions to include the effects of solid porous material on the near-wall turbulent flow on the fluid side of porous jump interfaces



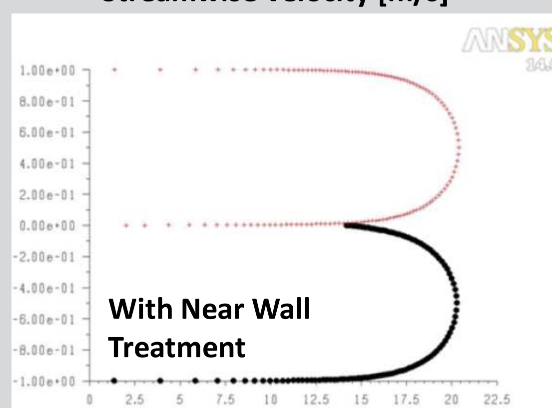
Contours of velocity showing the impact of a porous jump on velocity in bordering cells

y[m]



Streamwise Velocity [m/s]

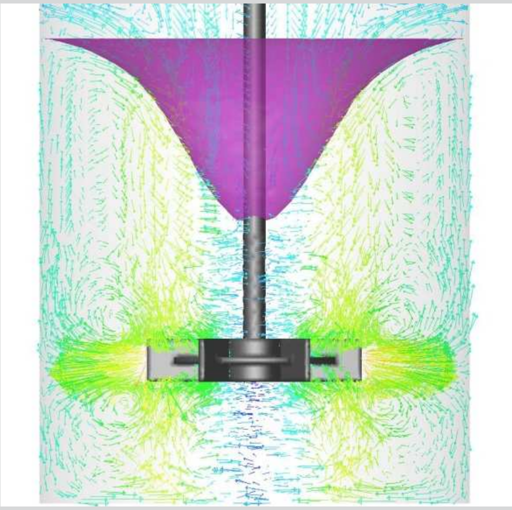
y[m]

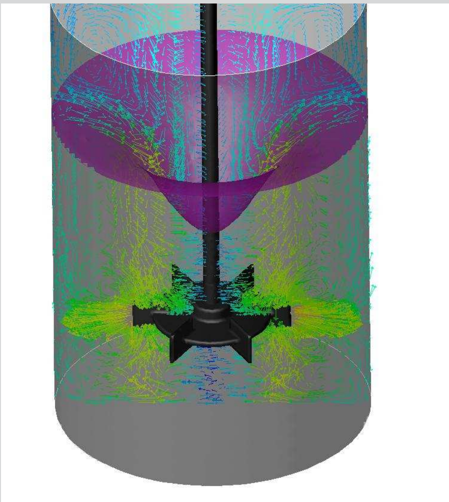


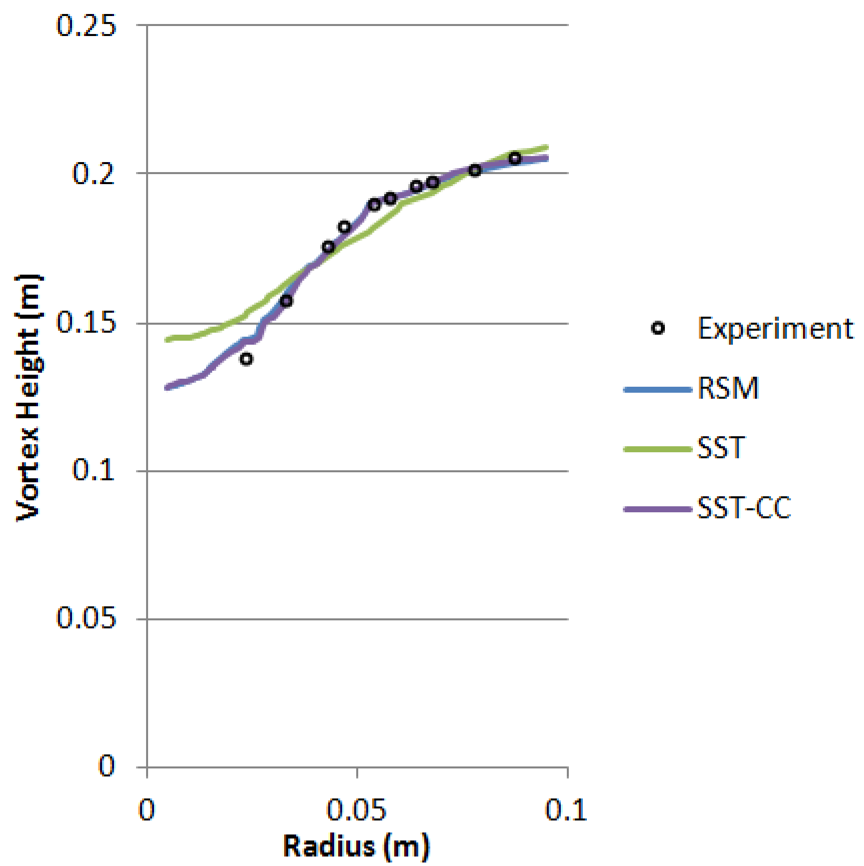
Streamwise Velocity [m/s]

Curvature Correction for One and Two Equation Models

- **Option to apply a correction term sensitive to rotatation and streamline curvature for one and two equation RANS models**
- **Can offer comparable accuracy to Reynolds Stress models with less computational effort for swirl dominated flows**



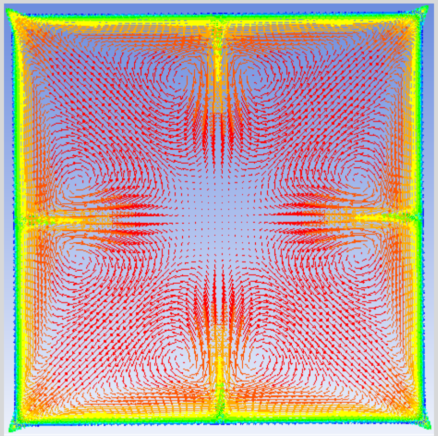


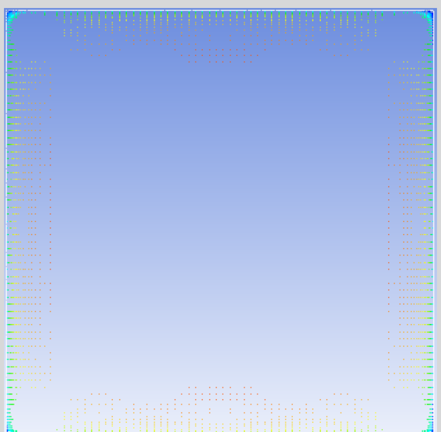


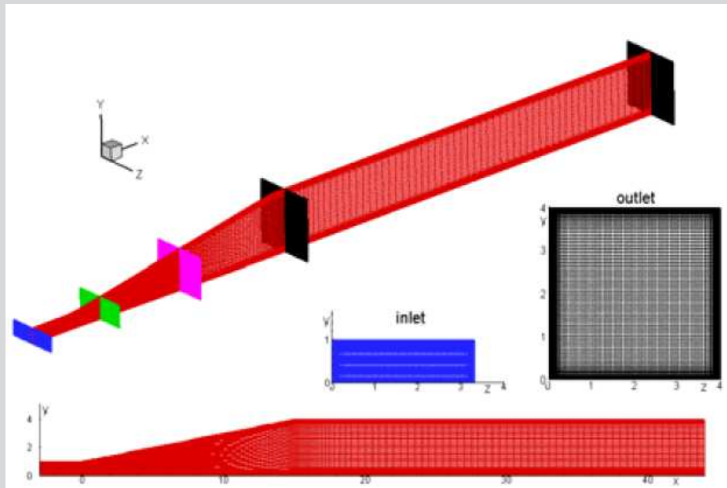
Example: Prediction of the vortex free surface in an unbaffled mixing tank

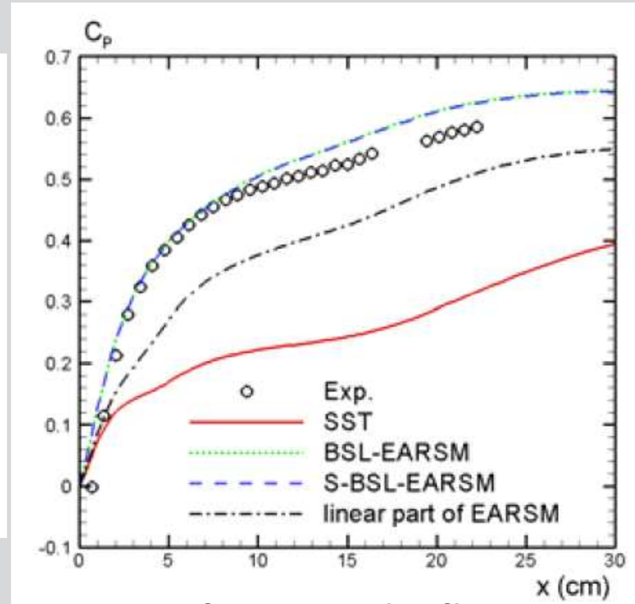
Explicit Algebraic Reynolds Stress Model (EARSM)

- **Non-linear algebraic expansion of Reynolds stress tensor allows two-equation model to capture anisotropic effects such as stress induced secondary flows in rectangular ducts**









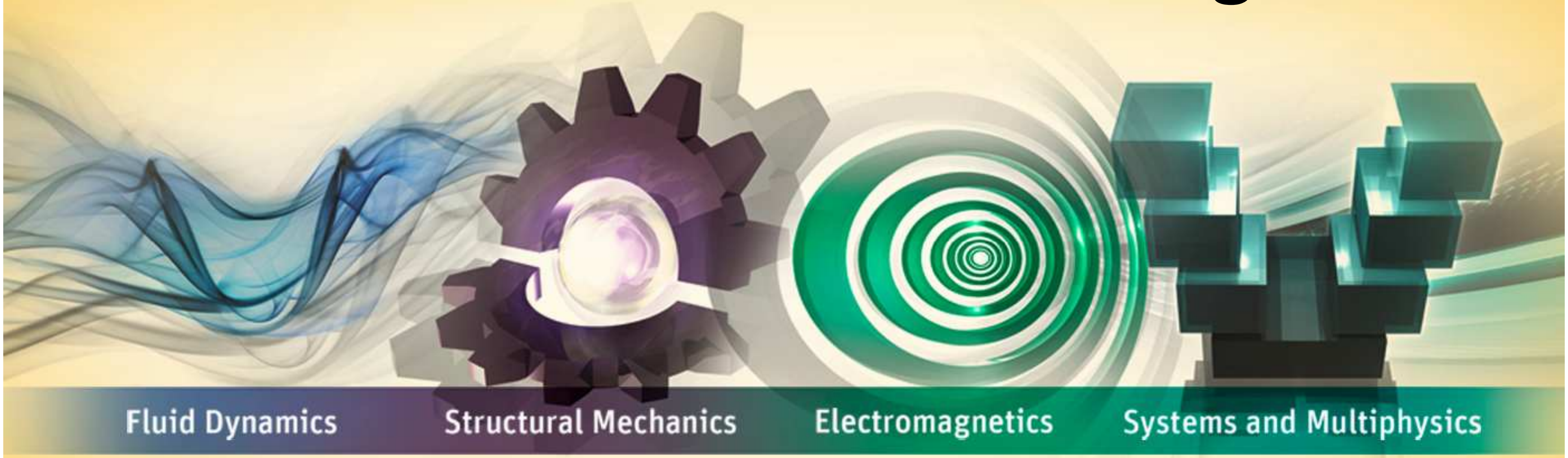
Left: In-plane component of velocity vectors for Periodic flow in a square duct. EARSM (above) predicts secondary flow patterns with velocity ~2.4 percent of bulk velocity. SST (below) predicts no secondary flow
Above and Right: Flow in a rectangular, asymmetric diffuser. EARSM correctly predicts pressure coefficient on bottom surface

Summary and Conclusions

- **Steady state RANS simulations will remain the dominant simulation method for turbulent flows for many years**
  - While increasing use of LES and other scale resolving simulation methods for engineering applications is predicted, RANS will still maintain important advantages in some areas
- **ANSYS strives to provide RANS models for use which are**
  - Accurate
  - Robust
  - Y^+ insensitive wall treatment
  - Interoperable with other physical models
- **Developments in recent ANSYS releases extend the range of capabilities of the core turbulence models**
  - Curvature correction, EARSM, free surface turbulence damping, porous media near wall treatment

Realize Your Product Promise™

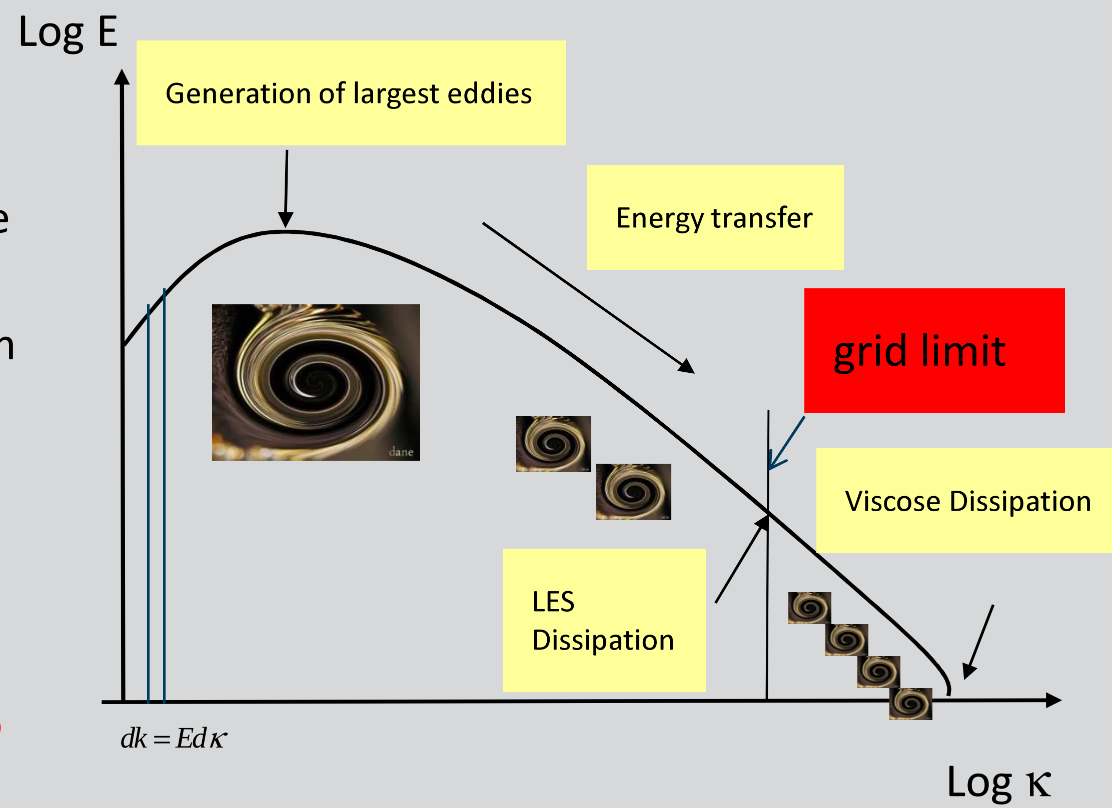
Large Eddy Simulation and Hybrid RANS-LES Turbulence Modeling



F.R. Menter
ANSYS Germany GmbH

Large Eddy Simulation (LES)

- **Role of LES:**
  - Turbulent spectrum cannot be resolved down to the dissipative scales (Kolmogorov scales)
  - Energy has to be dissipated from the spectrum at grid limit
  - LES Eddy Viscosity provides required damping
  - LES does not model the small scales – it just dissipates them
  - Everything of importance has to be resolved!



$$\varepsilon_{LES} = \varepsilon_{DNS} \longrightarrow \varepsilon_{DNS} = \nu \frac{\partial u_i}{\partial x_j}\frac{\partial u_i}{\partial x_j} \qquad \varepsilon_{LES} = \nu_t^{LES} \frac{\partial \hat{u}_i}{\partial x_j}\frac{\partial \hat{u}_i}{\partial x_j}$$

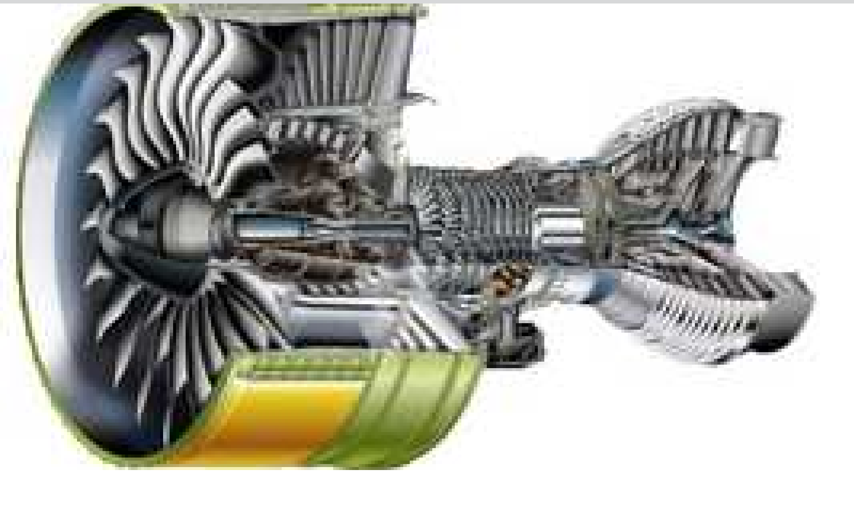
LES – Smagorinsky Modell

$$\nu_t^{LES} = (c\Delta)^2 S$$

LES - Wall Bounded Flows

ANSYS

- A single Turbine (Compressor) Blade ($Re=10^5$-10^6) with hub and shroud section
- Need to resolve turbulence in boundary layers
- Need to resolve laminar-turbulent transition

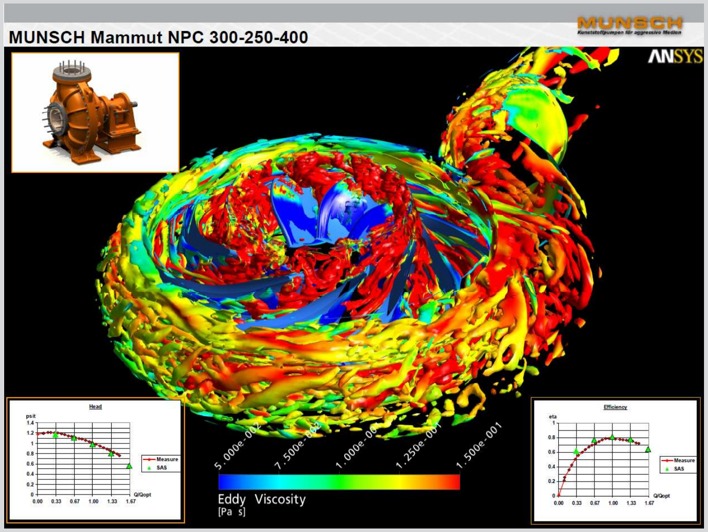


Method	Number of Cells	Number of time steps	Inner loops per Δt.	CPU Ratio
RANS	$\sim 10^6$	$\sim 10^2$	1	1
LES	$\sim 10^9$	$\sim 10^4$	10	10^6

Therefore Hybrid RANS-LES Methods

Motivation for Scale-Resolving Simulations (SRS)

- Accuracy Improvements over RANS
  - Flows with large separation zones (stalled airfoils/wings, flow past buildings, flows with swirl instabilities, etc.)
- Additional information required
  - Acoustics - Information on acoustic spectrum not reliable from RANS
  - Vortex cavitation – low pressure inside vortex causes cavitation – resolution of vortex required
  - Fluid-Structure Interaction (FSI) – unsteady forces determine frequency response of solid.

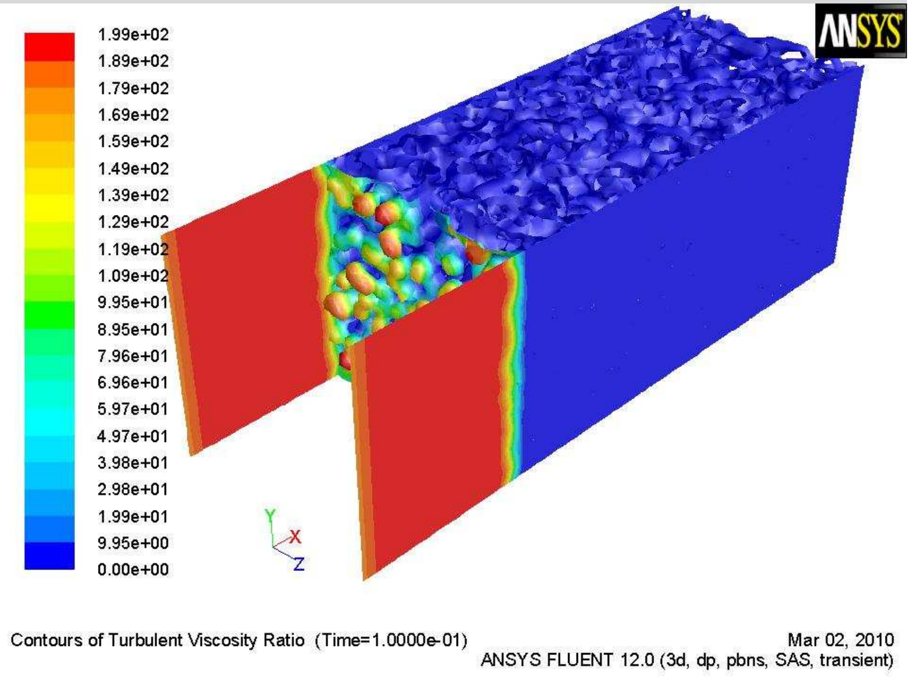


Scale-Resolving Simulation (SRS)

- SRS refers to all turbulence models, which resolve at least a portion of the turbulence spectrum in at least a part of the domain
  - Scale-Adaptive Simulation (SAS)
  - Detached Eddy Simulation (DES)
  - Large Eddy Simulation (LES)
  - Wall-modelled LES (WMLES)
  - Embedded and Zonal LES (ELES, ZFLES)
  - Other RANS-LES hybrids
- SRS is a field of intense research and many new model formulations/combinations are explored
- In ANSYS CFD R14, the most promising new approaches were selected and implemented

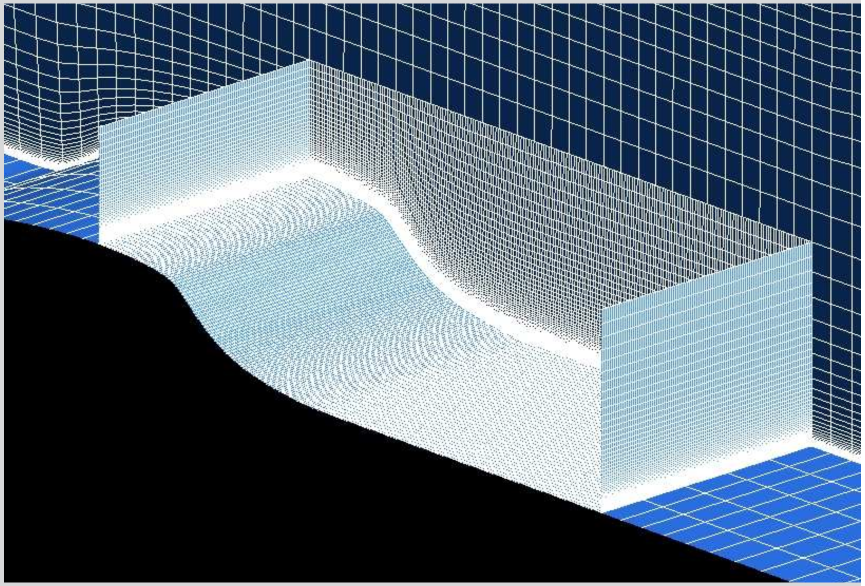
Embedded LES and Zonal Forced LES

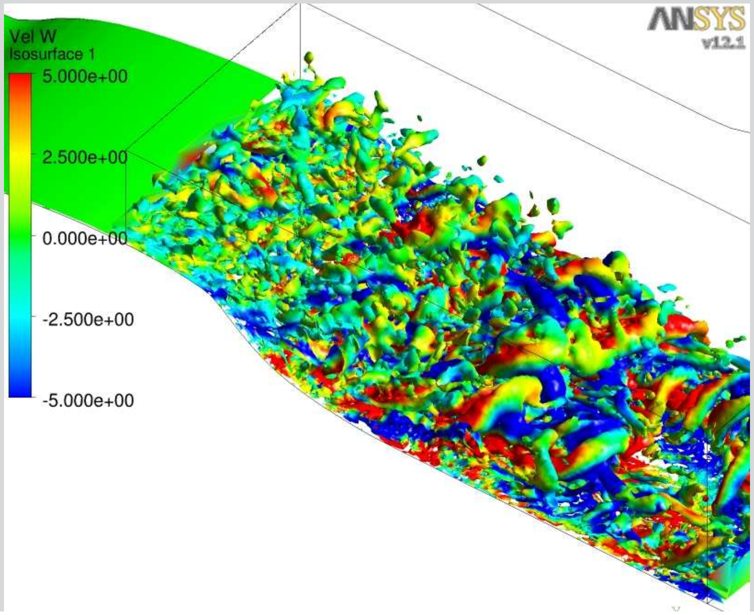
- In many flows an area where (WM)LES is required is embedded in a larger RANS region
- In such cases, a zonal method is advantageous
- RANS and LES regions are separately defined and use different models
- Synthetic turbulence is generated at the interface to convert RANS to LES turbulence



ANSYS-Fluent and ANSYS-CFX

Flow over a wall mounted hump, Geometry and Grid





Geometry:

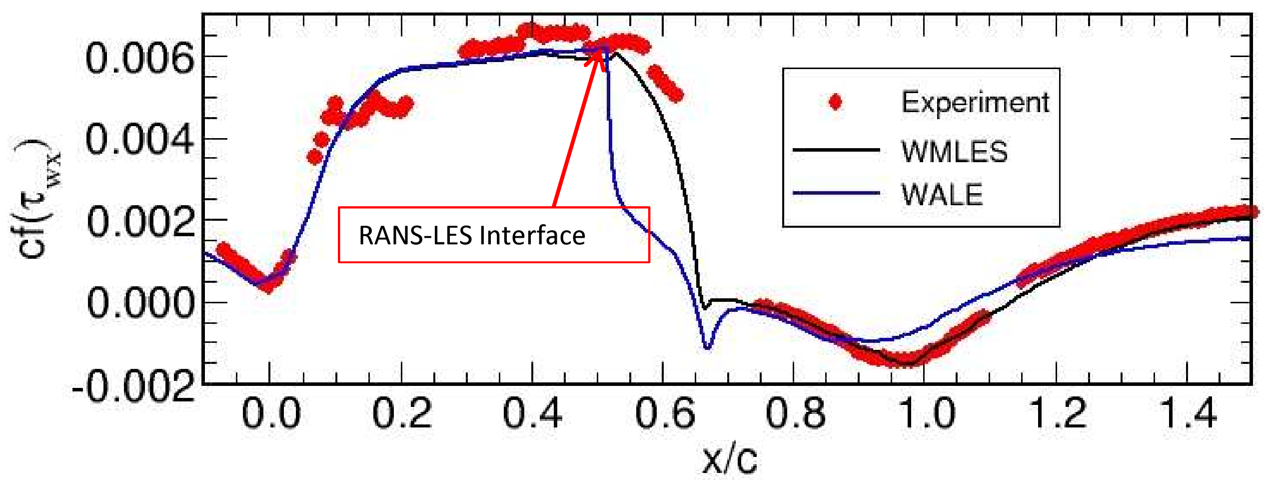
- Spanwise extent:
  - 3.16 H (bump height)
  - 5.6 $\delta_{interface}$ (δ – boundary layer thickness).

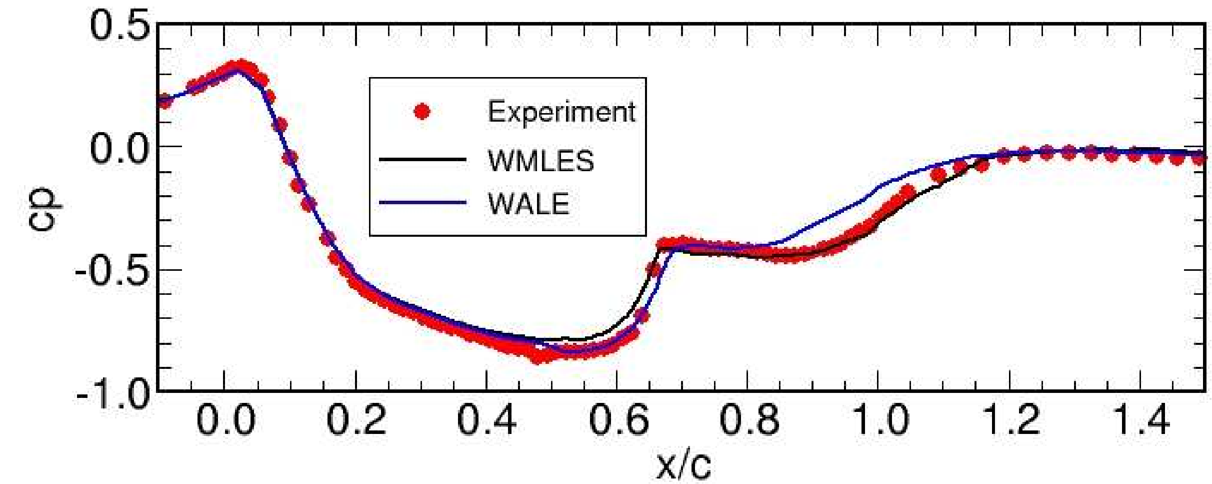
Grid:

- RANS grid with only 5 cells in spanwise direction
- LES grid: 200x100x100 (2 million)
- Grid resolution per inlet boundary layer ($\Delta x/\delta$=10, $\Delta z/\delta$~20, NY~40.

Flow over a wall mounted hump Wall Shear Stress and Wall Pressure

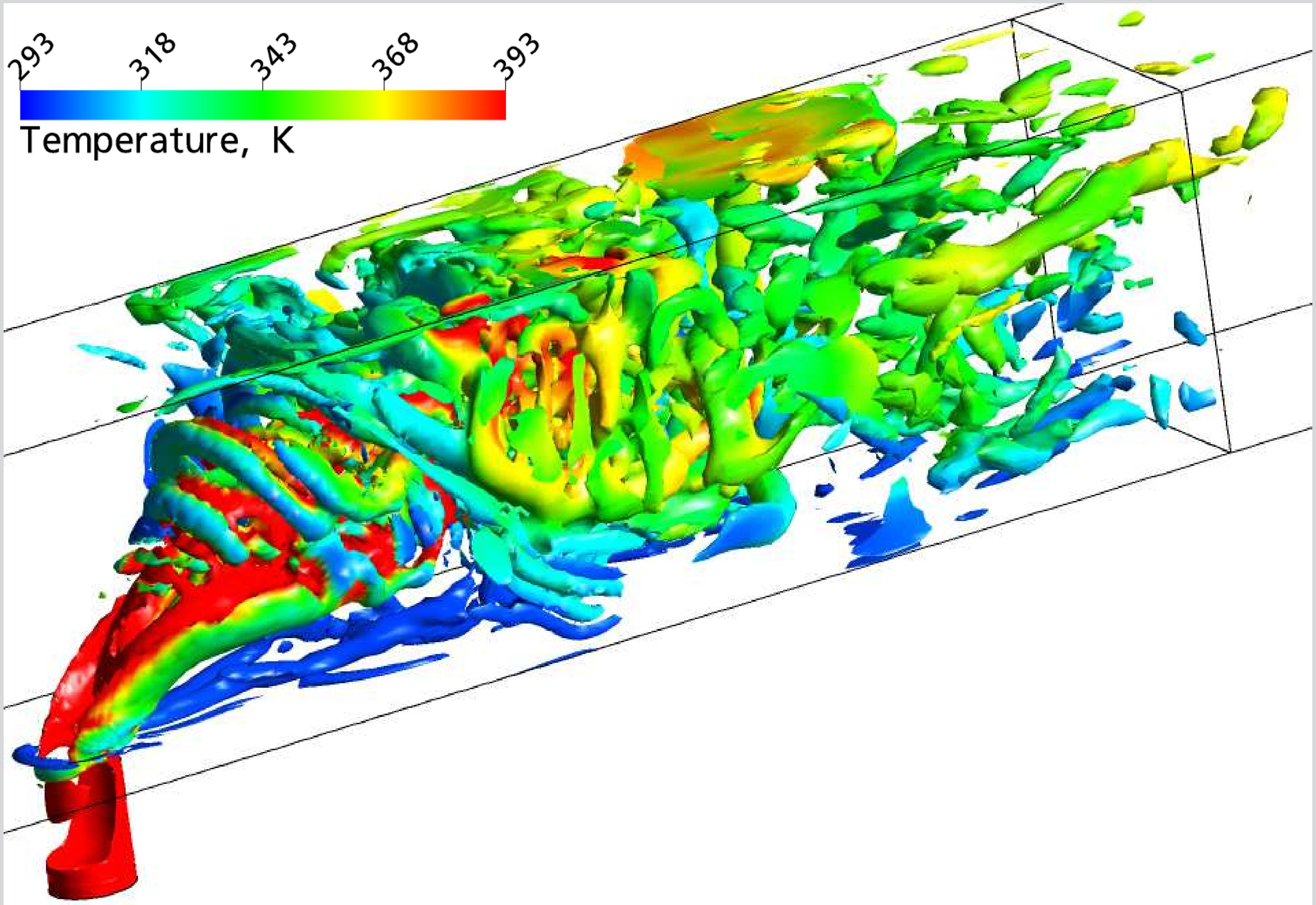
- The Re number at the RANS-LES interface is $Re_{\Theta}=7000$
- If the simulation in the LES region is carried out with a standard LES model (WALE) the solution is lost immediately after the interface
- The WMLES formulation is able to carry the solution smoothly across and provide a good agreement with the data for two different time steps (CFL~0.5 and CFL~0.12)



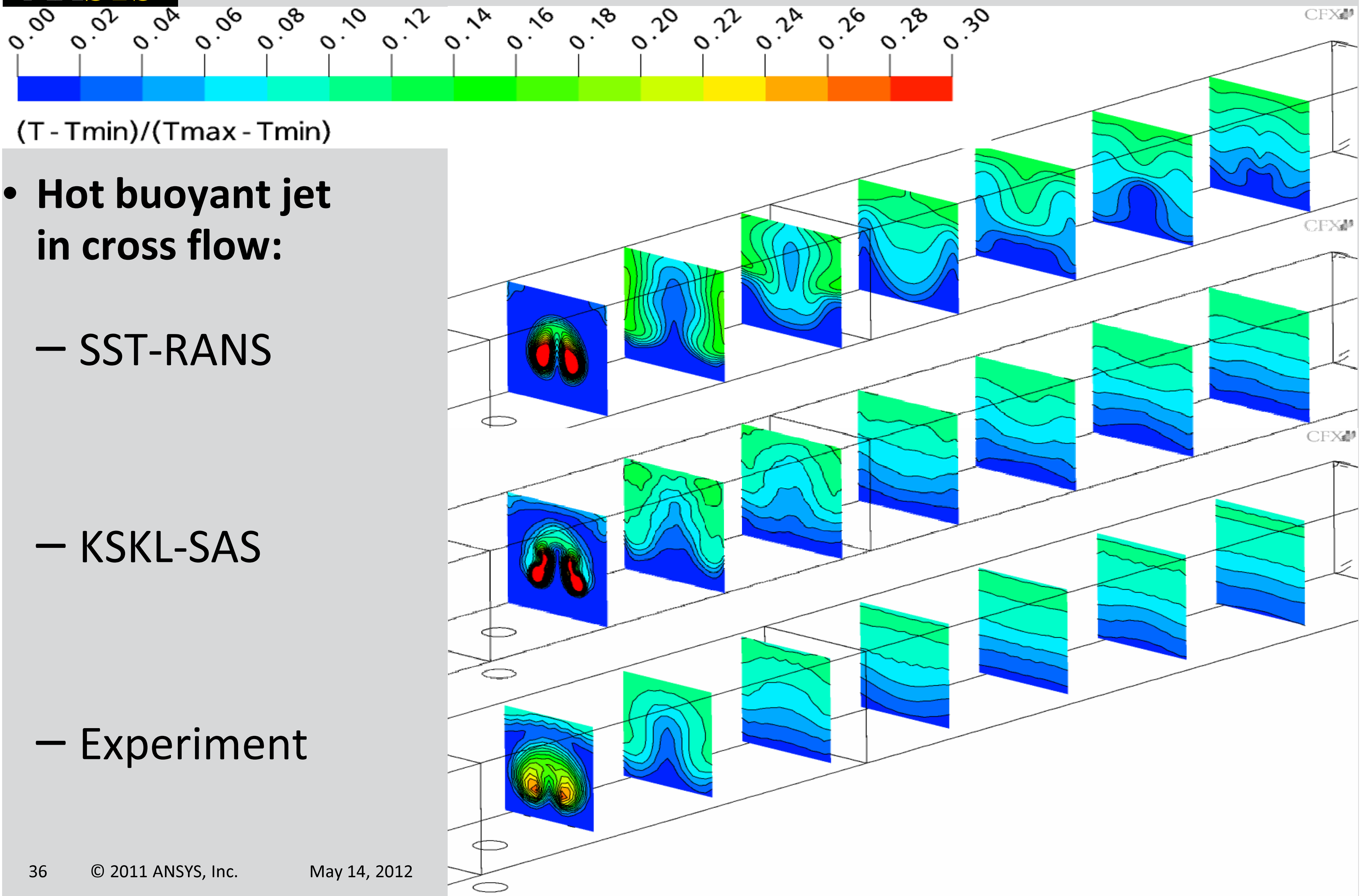


3-D Transient Flows: SAS

- **Hot buoyant jet in cross flow in a channel (ETH)**



Temperature Distribution



- **Hot buoyant jet in cross flow:**
  - SST-RANS
  - KSKL-SAS
  - Experiment

