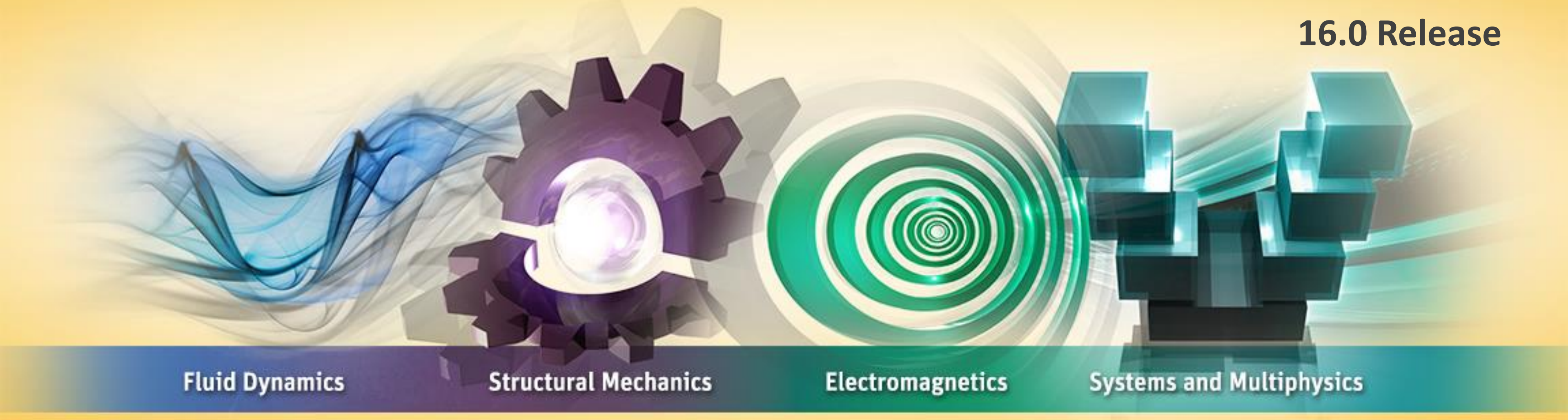
Lecture 10: Turbulence

16.0 Release



Introduction to ANSYS CFX

Realize Your Product Promise®

Introduction

Lecture Theme:

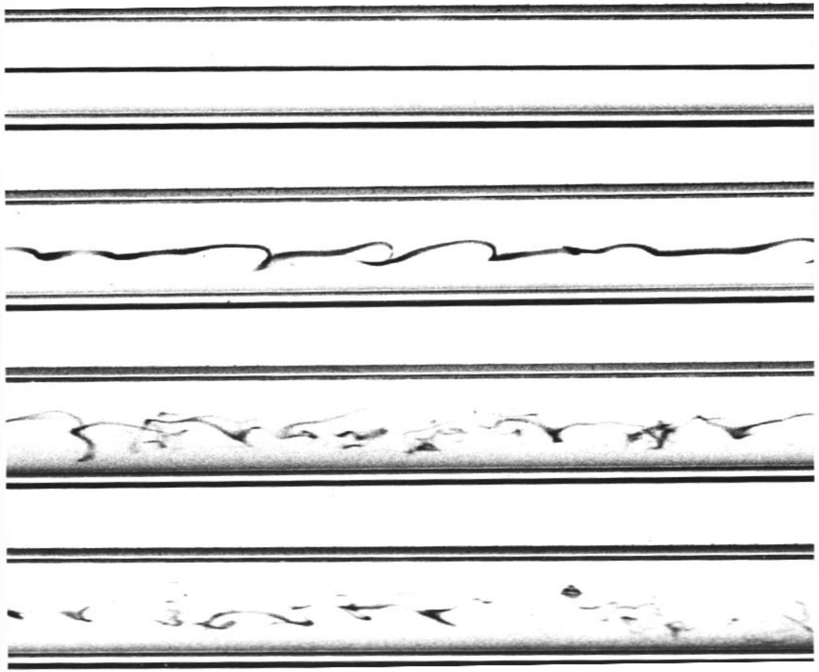
- Engineering flows are turbulent
- Successfully simulating such flows requires understanding a few basic concepts of turbulence theory and modeling

Learning Aims – you will learn:

- Basic turbulent flow and turbulence modeling theory
- You will understand the challenges inherent in turbulent flow simulation and be able to identify the most suitable model and near-wall treatment for a given problem.

Observation by O. Reynolds

- **Flows can be classified as :**
  - **Laminar:**
    - Low Reynolds number
  - **Transition:**
    - Increasing Reynolds number
  - **Turbulent:**
    - Higher Reynolds number



Observation by O. Reynolds

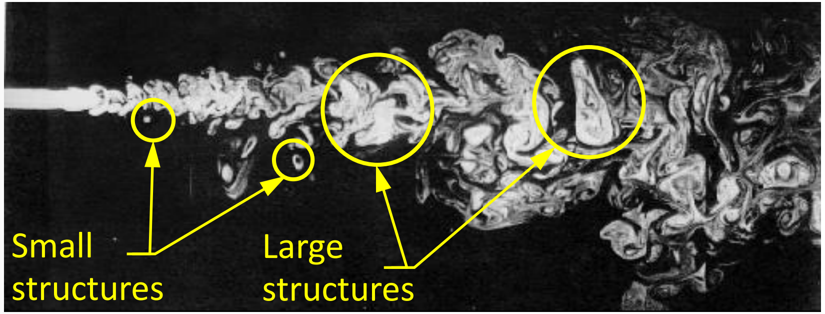
- The Reynolds number is the criterion used to determine whether the flow is laminar or turbulent

$$\mathrm{Re} = \frac{u \cdot L \cdot \rho}{\mu}$$

  - The Reynolds number is based on the characteristic length scale L, the flow velocity u and the fluid properties ρ and μ
  - Transition to Turbulence varies depending on the type of flow:
    - External flow
      - along a surface : $Re_X > 5 \times 10^5$
      - around an obstacle : $Re_L > 2 \times 10^4$
    - Internal flow : $Re_D > 2300$

Turbulent Flow Structures

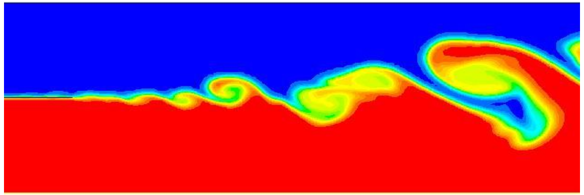
- Turbulent flow - wide range of turbulent eddy sizes. Characteristics:
  - Unsteady, tridimensional, irregular, stochastic
  - Transported quantities fluctuate in time and space
  - Unpredictability in detail
  - Large-scale coherent structures different in each flow. Small eddies more universal
  - Energy transferred from larger to smaller scale eddies
  - In smallest eddies, turbulent energy is converted to internal energy by viscous dissipation



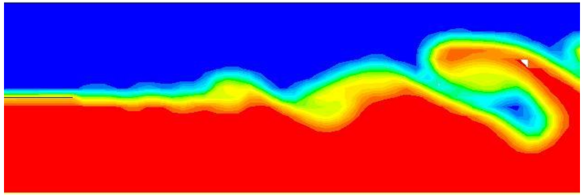
Introduction | Theory | Models | Near-Wall Treatments | Inlet BCs | Summary

Overview of Computational Approaches

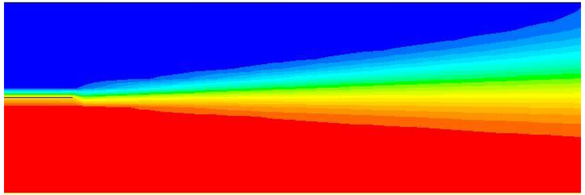
- **DNS: Direct numerical simulation**
  - Full resolution, no modeling → Too expensive for practical flows



- **LES: Large eddy simulation**
  - Large eddies directly resolved, smaller ones modeled → Less expensive than DNS, but very often still too expensive for practical applications

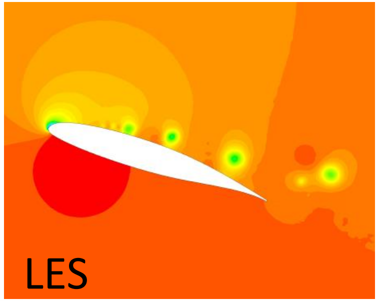


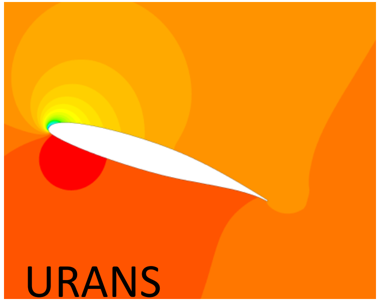
- **RANS: Reynolds averaged Navier-Stokes simulation**
  - Solution of time-averaged equations → Most widely used approach for industrial flows



RANS Modeling : Justification

- For most engineering applications it is unnecessary to resolve the details of the turbulent fluctuations
- We only need to know how turbulence affects the mean flow
- A useful turbulence model has to be:
  - applicable in wide ranges,
  - accurate,
  - simple,
  - and economical to run.



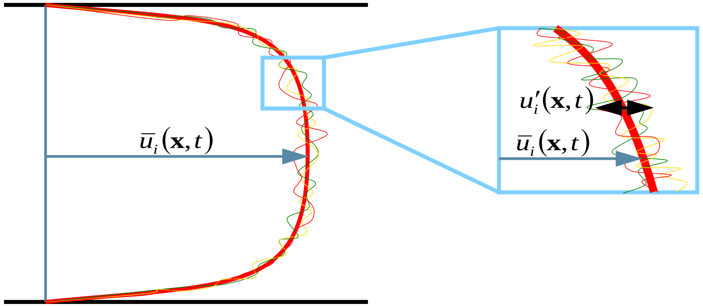


RANS Modeling : Averaging

- **Ensemble averaging allows extraction of mean from instantaneous flow properties**

$$u_i(\mathbf{x},t) = \bar{u}_i(\mathbf{x},t) + u_i'(\mathbf{x},t)$$

instantaneous average fluctuation



Example: Fully-Developed Turbulent PipeFlow Velocity Profile

- **Momentum equation becomes:**

$$\rho\left(\frac{\partial \bar{u}_i}{\partial t} + \bar{u}_k \frac{\partial \bar{u}_i}{\partial x_k}\right) = -\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\mu \frac{\partial \bar{u}_i}{\partial x_j}\right) + \frac{\partial R_{ij}}{\partial x_j}$$

$$R_{ij} = -\rho \overline{u_i' u_j'}$$

Reynolds stress tensor

  - **Introduces 6 additional unknowns in Reynolds stress tensor, R_{ij}**
  - **They must be modelled. This is the closure problem**

RANS Modeling : The Closure Problem

- The RANS models are closed by modelling the Reynolds-stress tensor, *Rij*, in one of two ways:
  - Reynolds-Stress Models (RSM)
    - The components of R_{ij} are directly solved via transport equations
    - Advantageous in complex 3D flows with streamline curvature / swirl
    - Models are complex and computational intensive
  - Eddy Viscosity Models
    - Model the components of R_{ij} using an eddy (turbulent) viscosity μ_t
    - Reasonable approach for simple turbulent shear flows: boundary layers, round jets, mixing layers, channel flows, etc.
    - The effective viscosity is given by $\mu_{eff} = \mu_t + \mu$, usually $\mu_t >> \mu$

Turbulence Models in CFX

- A large number of turbulence models are available, some for very specific applications, others can be applied to a wider class of flows with a reasonable degree of confidence

RANS Eddy-viscosity Models	**RANS Reynolds-Stress Models**	**Eddy Simulation Models (Scale Resolving Models SRS)**
Zero Equation model	SSG model	Large Eddy Simulation (LES)
Standard k-ε, k-ω model	LRR model	Detached Eddy Simulation (DES)
SST model	BSL EARSM model	Scale Adaptive Simulation SST (SAS)

k- ε Models

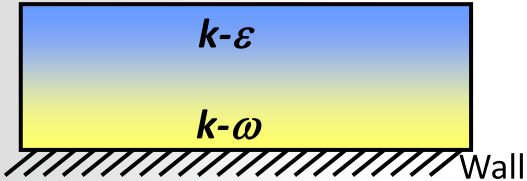
- Good compromise between numerical effort and computational accuracy
  - Two transport equations for the solution of TKI and dissipation
  - Turbulent viscosity is modeled as product of turbulent velocity and turbulent length scale
- Good predictions for many flows of engineering interest
- k- ε models not suitable for modeling
  - flows with boundary layer separation,
  - flows with sudden changes in the mean strain rate,
  - flows in rotating fluids,
  - flows over curved surfaces.

k-ω Models

- Good compromise between numerical effort and computational accuracy
  - Two transport equations for the solution of TKI and frequency
  - Turbulent viscosity is modeled as product of turbulent velocity and turbulent length scale
- Good predictions for many flows of engineering interest
- k-ω models better than k- ε models for boundary layer flows
  - separation,
  - transition,
  - low Re effects,
  - impingement.

Shear Stress Transport (SST) Model

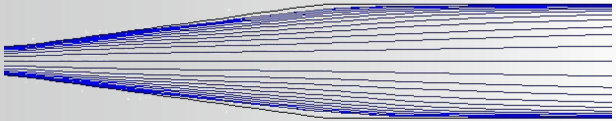
- **Relative performances of k-ε & k-ω models depend on the region of flow:**
  - **k-ω model performs much better for boundary layer flows**
  - **original k- ω model is sensitive to the free-stream conditions but k-ε is not**
- **SST model overcomes the shortcomings of k-ε and k-ω models:**
  - **blends between the two models according to the distance from the wall**
  - **accounts for transport of turbulent shear stress**
    - Prevents over-prediction of eddy viscosity
    - Improves prediction of onset & degree of separation from smooth surfaces



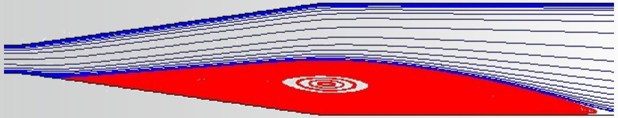
SST Model: Validation Example

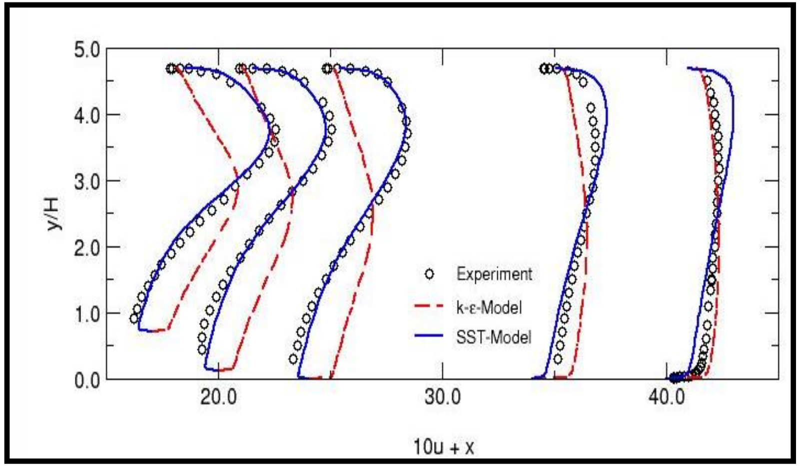
The SST model predicts well the onset and the amount of flow separation

Standard k-ε fails to predict separation



SST result and experiment



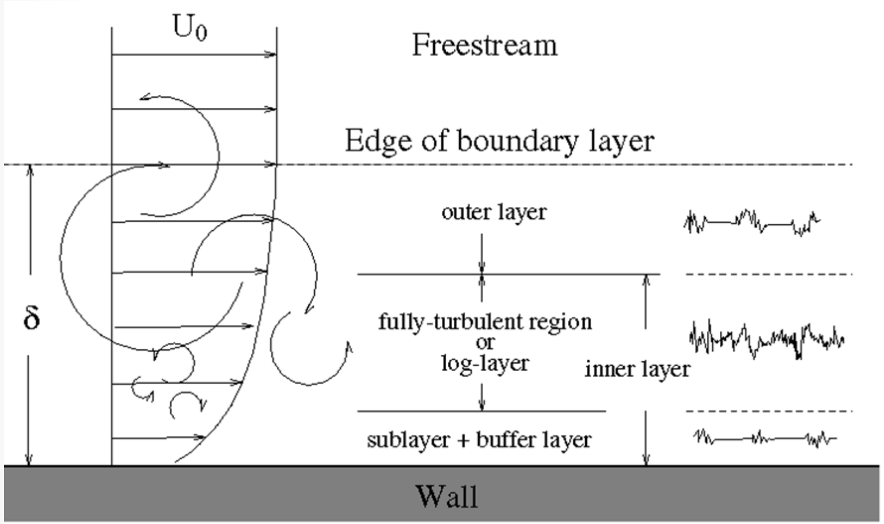


Experiment Gersten et al.

Introduction | Theory | Models | Near-Wall Treatments | Inlet BCs | Summary

Turbulence Near the Wall

- **Walls are main source of turbulence**
- **Velocity profile near wall is important:**
  - **Pressure drop, separation, recirculation, shear effects & heat transfer**
- **Accurate near-wall modeling is important**
  - **Turbulence models suited to flow outside the boundary layer**
  - **Need special treatments near walls**



Turbulence Near the Wall

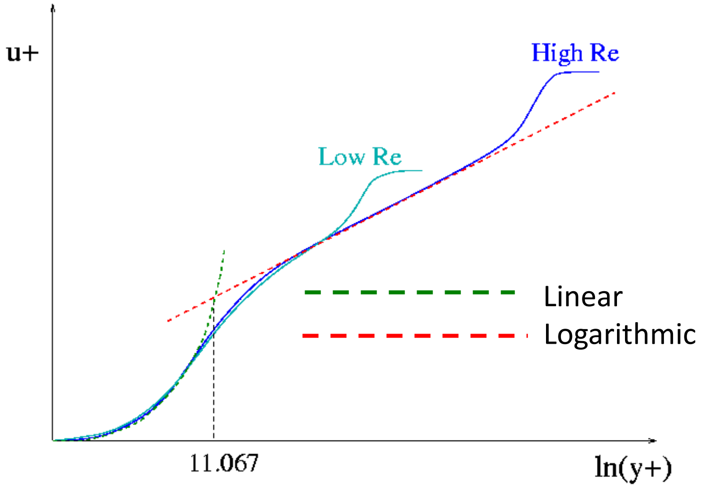
- **Plot of dimensionless velocity vs dimensionless distance gives a profile that is the same for all Re – “Law of the Wall”**
  - **non-dimensional velocity**

$$u^{+} = \frac{u}{\sqrt{\tau_{Wall} / \rho}}$$

  - **non-dimensional distance from the wall**

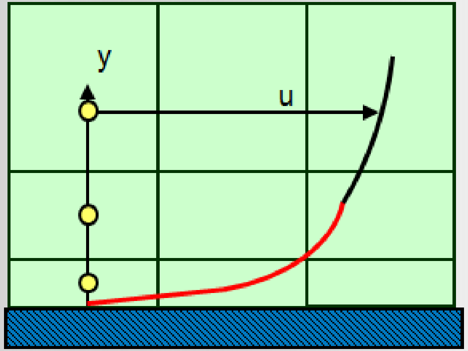
$$y^{+} = \frac{y\sqrt{\tau_{Wall} / \rho}}{\nu}$$

- **Behaviour allows the development of wall functions**

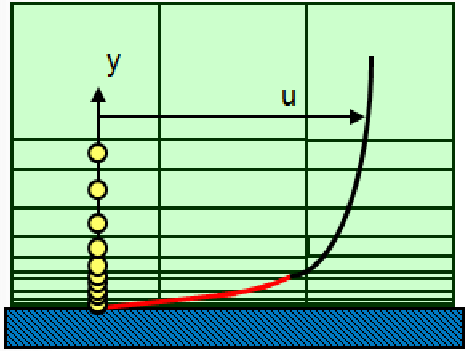


Turbulence Near the Wall

- **Wall functions avoid need for mesh to resolve the profiles near wall**



Logarithmic-based Wall functions used to resolve boundary layer



Near-wall resolving approach used to resolve boundary layer

Boundary layer

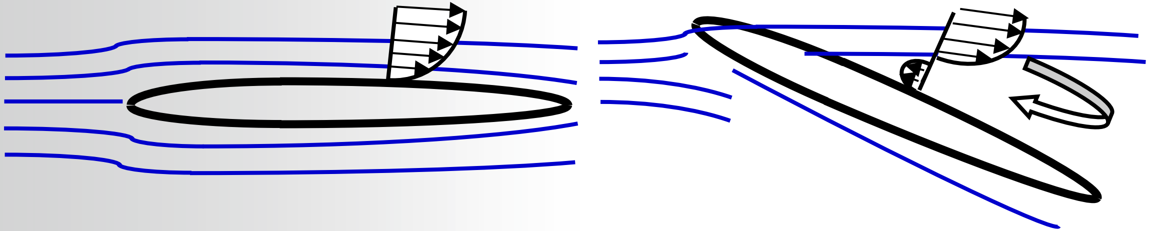
- **Yplus reports dimensionless distance to first vertex adjacent to wall**
  - **tells us in which region of boundary layer it lies**

Location of The First Vertex

- **Automatic wall functions (ω-based turbulence models, e.g. SST)**
  - **Switches between wall function & low-Re wall treatment as mesh is refined**
  - **Wall functions**
    - Wall-adjacent vertices should be in the log-law layer: **$y+ \approx$ 20-200**
  - **Resolved wall treatment**
    - Wall-adjacent vertices should be within the viscous sublayer:
    - **$y+ \approx 1$** with a minimum of 10 nodes in boundary layer
- **Scalable wall functions (k-ε models)**
  - **No switch. Wall-adjacent vertices should be in the log-law layer: $y+ \approx$ 20-200**
  - **In highly refined mesh, the first vertex is shifted virtually to $y+ = 11.067$, linear to logarithmic transition.**
  - **Further refinement near the wall has no effect**

Limitations of Wall Function

- In some situations, such as boundary layer separation, logarithmic-based wall functions do not correctly predict the boundary layer profile



→ **logarithmic-based wall functions should not be used**

→ **resolving the boundary layer can provide accurate results**

Wall Function and Heat Transfer

- Heat flux at the wall, q_w , is given by:

$$q_w = \frac{\rho c_p u^*}{T^+}\left(T_w - T_f\right) = h_c\left(T_w - T_f\right)$$

- T_w is the wall temperature and T_f is the near-wall temperature in the fluid

- The equations for u^* and the non-dimensional temperature,T^+ , depend on the type of wall function
  – For scalable wall functions T^+ follows a log-law relationship
  – For automatic wall functions the correlation between T^+ and wall distance blends between the viscous sublayer and the log law

Inlet Turbulence Conditions

- When turbulent flow enters a domain, turbulent boundary conditions must be specified
- Several options exist for the specification of turbulence quantities
  - Explicitly input k and either ε or ω
  - Turbulence intensity and length scale
  - Turbulence intensity and turbulent viscosity ratio
  - Turbulent Intensity: $I = \frac{u'}{\bar{u}} \approx \frac{1}{\bar{u}}\sqrt{\frac{2k}{3}}$
  - Turbulent viscosity ratio: μ_t / μ

Introduction | Theory | Models | Near-Wall Treatments | Inlet BCs | Summary

Inlet Turbulence Conditions

- If you have absolutely no idea of the turbulence levels in your simulation, you could use following values of turbulence intensities and length scales:
  - Usual turbulence intensities range from 1% to 5%
  - The default turbulence intensity value of 0.037 (that is, 3.7%) is sufficient for normal turbulence through a circular inlet, and is a good estimate in the absence of experimental data

	Low	Medium	High
I	1%	5%	10%
μ_t/μ	1	10	100

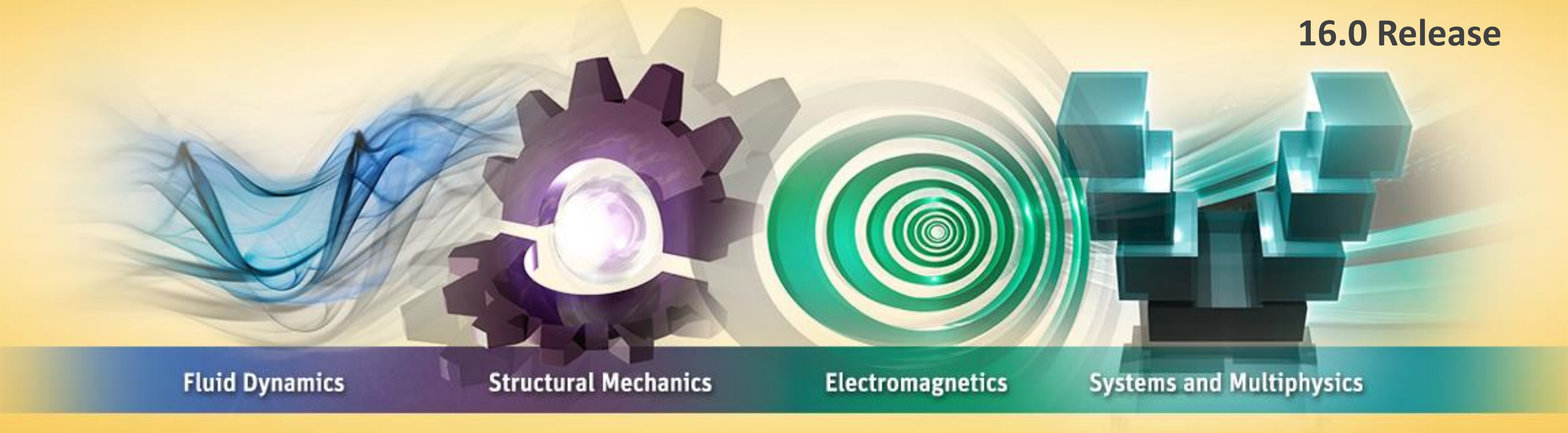
  - Some more details for the specification of the inlet turbulence conditions can be found in the appendix

Summary - Turbulence Modeling Guidelines

- Calculate Re to determine whether the flow is turbulent
- Estimate y^+ before generating the mesh
- The SST model is good choice for most flows
- Use the RSM or the SST model with Curvature Correction for highly-swirling flows
- Consider using SRS models, LES, DES and SAS, if you need to resolve the turbulence structure, e.g. for acoustics/vibration applications
- Use the default model parameters unless you are confident that you have better values!

Appendices

16.0 Release



Introduction to ANSYS CFX

Realize Your Product Promise®

Turbulent Flow Structures (1)

Characteristics of the Turbulent Structures:

- **Length Scale:** $l \qquad [m]$
  - Describing size of large energy-containing eddies in a turbulent flow
  - In many cases defined by a relation $$l \sim \frac{k^{3/2}}{\varepsilon}$$
- **Turbulent kinetic energy:** $$k = \frac{1}{2}\left(\overline{u'^2} + \overline{v'^2} + \overline{w'^2}\right) \qquad [m^2/s^2]$$
- **Velocity Scale:** $\sqrt{k} \qquad [m/s]$
- **Time Scale:** $\dfrac{l}{\sqrt{k}} \qquad [s]$

Turbulent Flow Structures (2)

Characteristics of the Turbulent Structures:

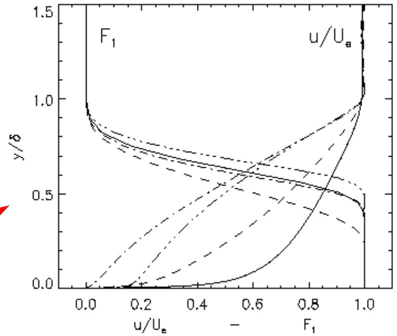
- **Turbulent dissipation:** $$\varepsilon \sim k^{3/2} / l \qquad [m^2 / s^3]$$
- **Turbulent frequency:** $$\omega \approx \frac{\varepsilon}{k} \propto \frac{1}{\tau} \qquad [1/s]$$
- **Turbulent Reynolds number:** $$\mathrm{Re}_t = \frac{\sqrt{k}\, l}{\nu} \sim \frac{k^2}{\nu\varepsilon} \qquad [-]$$
- **Turbulent Intensity:** $$I = \frac{u'}{\bar{u}} \approx \frac{1}{\bar{u}} \sqrt{\frac{2k}{3}} \qquad [-]$$

SST Model: Blending k-ω and k-ε Models (1)

Two transport equations

- **turbulent kinetic energy**

$$\frac{\partial(\rho k)}{\partial t}+\frac{\partial(\rho \bar{U}_j k)}{\partial x_j}=P_k-\beta^{*}\rho k\omega+\frac{\partial}{\partial x_j}\left[(\mu+\frac{\mu_t}{\tilde{\sigma}_\kappa})\frac{\partial k}{\partial x_j}\right]$$



- **turbulent frequency**

$$\frac{\partial(\rho\omega_i)}{\partial t}+\frac{\partial(\rho \bar{U}_j \omega)}{\partial x_j}=\tilde{\alpha}\frac{\omega}{k}P_k-\tilde{\beta}\rho\omega^2+(1-F_1)\rho\frac{2}{\sigma_\omega}\frac{\partial k}{\partial x_j}\frac{\partial \omega}{\partial x_j}+\frac{\partial}{\partial x_j}\left[(\mu+\frac{\mu_t}{\tilde{\sigma}_\omega})\frac{\partial \omega}{\partial x_j}\right]$$

Blending function F1 switches from k-ε to k-ω

- 1 near the wall
- 0 towards edge of boundary layer

SST Model: Turbulent shear stress (2)

- k-ω model
  - does not predict properly the onset & degree of separation from smooth surfaces because it does not account for the transport of turbulent shear stress and so over-predicts eddy viscosity
- SST model
  - accounts for this transport by means of a limiter in the formulation of eddy viscosity

$$-\overline{u_i' u_j'} = \nu_t \left| \frac{\partial \bar{u}}{\partial y} \right| \quad \text{and} \quad \nu_t = \frac{a_1 k}{\max\left(SF_2, a_1 \omega\right)}$$

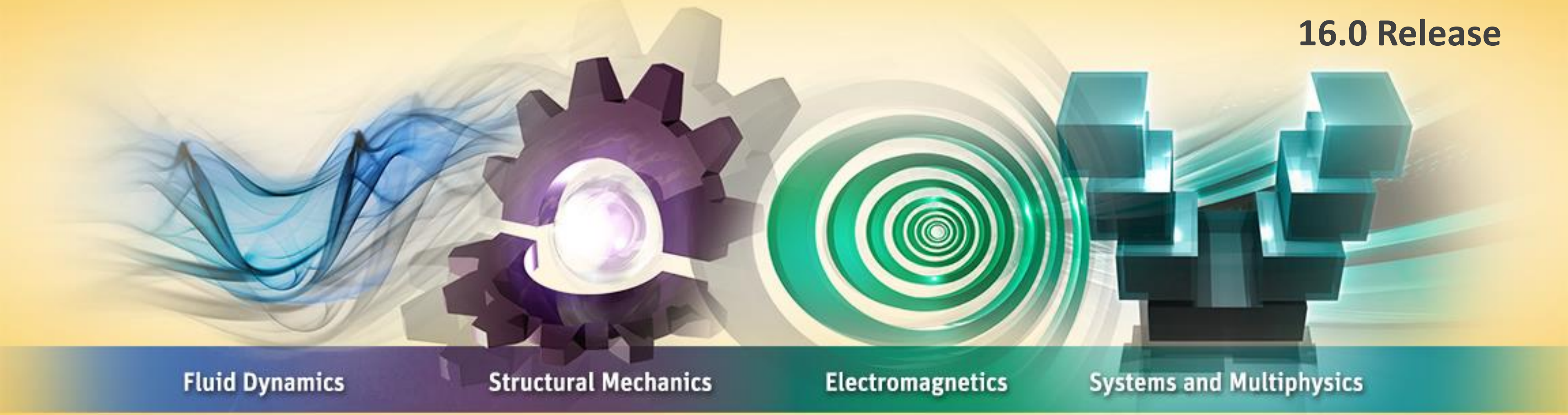
- Where shear stress, S, dominates
- → BSL k-ω + limiter = SST

$$-\overline{u_i' u_j'} = a_1 k$$

Introduction | Theory | Models | Near-Wall Treatments | Inlet BCs | Summary

Appendix: Pipe Expansion with Heat Transfer

16.0 Release



Introduction to ANSYS CFX

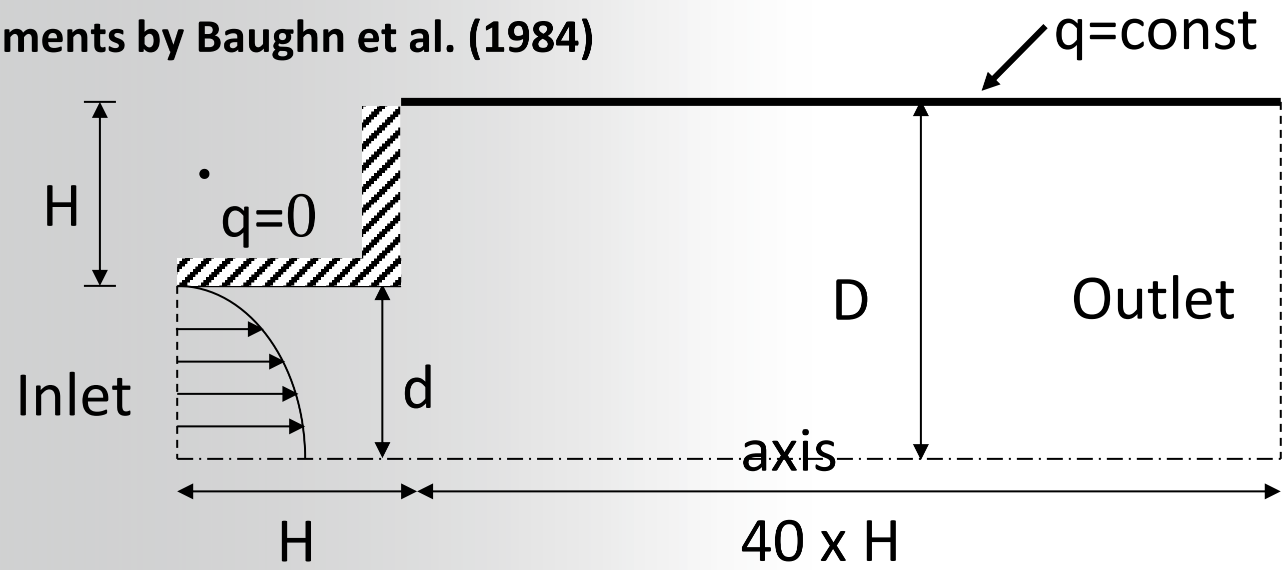
Realize Your Product Promise®

Example: Pipe Expansion with Heat Transfer

Reynolds Number Re_D= 40750

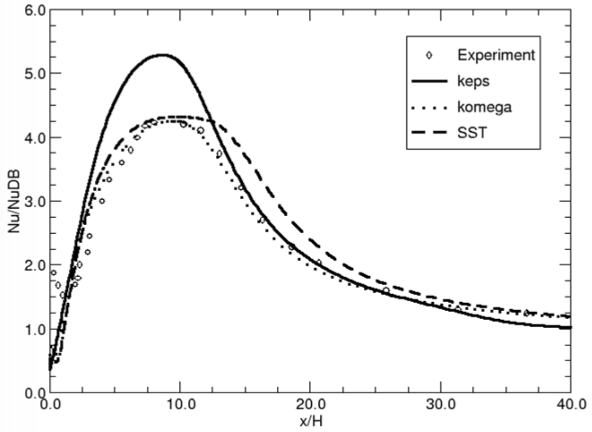
Fully Developed Turbulent Flow at Inlet

Experiments by Baughn et al. (1984)



Example: Pipe Expansion with Heat Transfer

- Plot shows dimensionless distance versus Nusselt Number
- Best agreement is with SST and k-ω models
- Better in capturing flow recirculation zones accurately

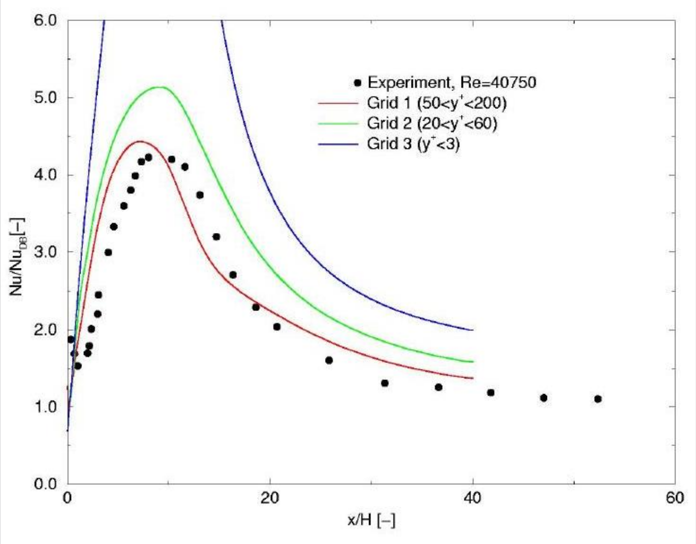


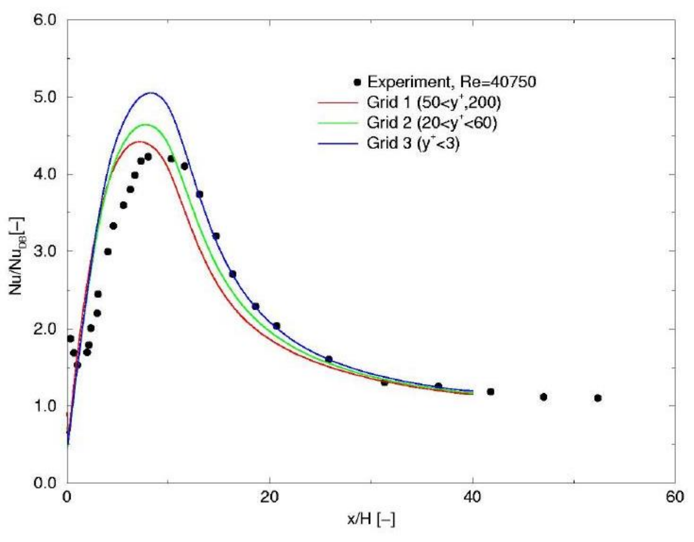
Introduction | Theory | Models | Near-Wall Treatments | Inlet BCs | Summary

Pipe Expansion With Heat Transfer

Heat transfer predictions

- **Wall function approach (k-ε Model)**



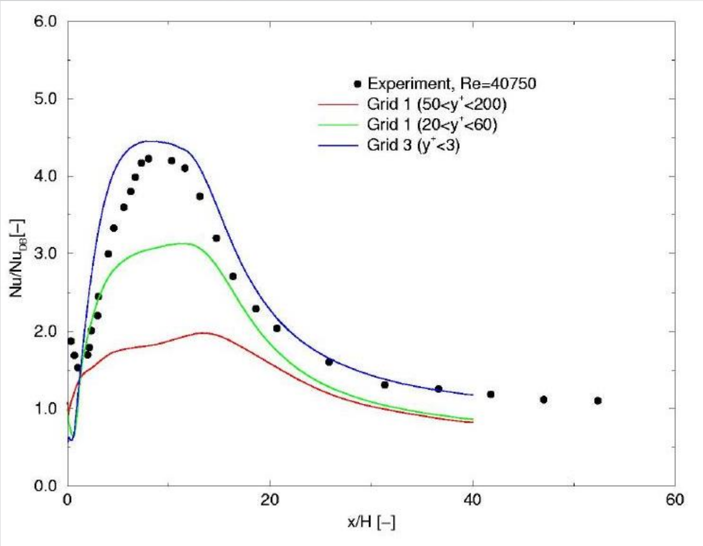


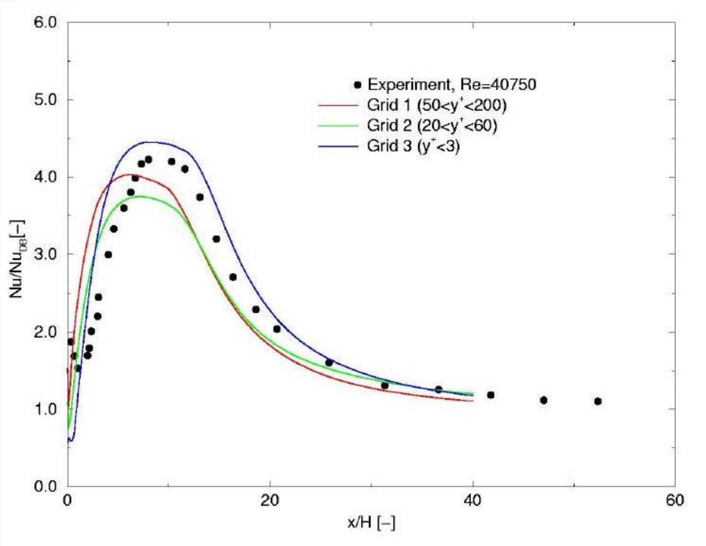
Standard ⟷ **Scalable**

Pipe Expansion With Heat Transfer

Heat transfer predictions

- **Automatic Wall treatment (SST-Model)**



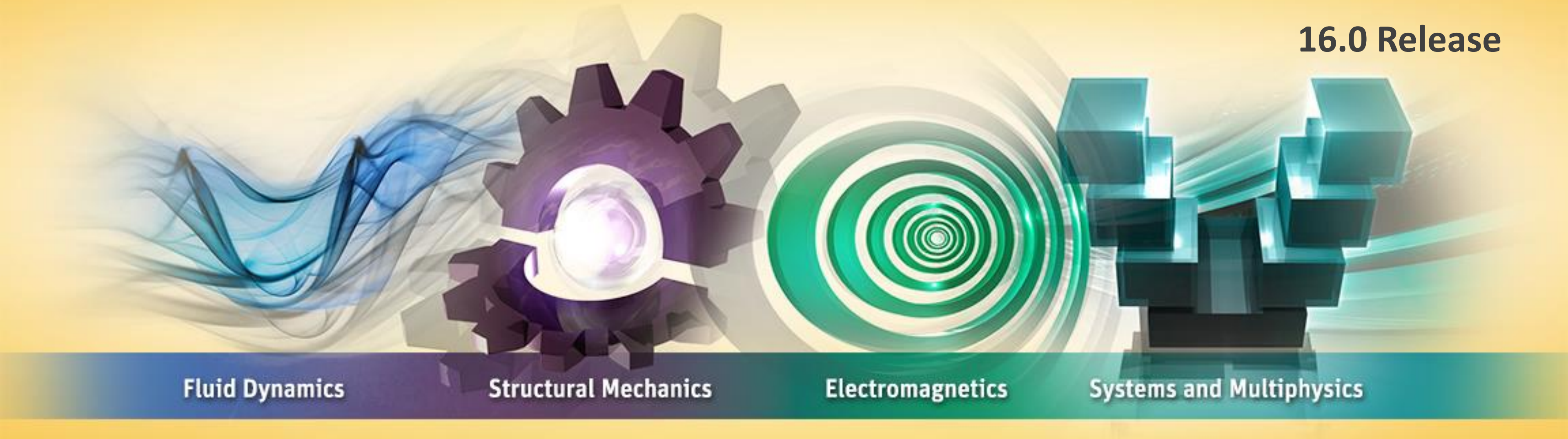


Low Re **Automatic**

ANSYS

Appendix: Summary of RANS Models

16.0 Release



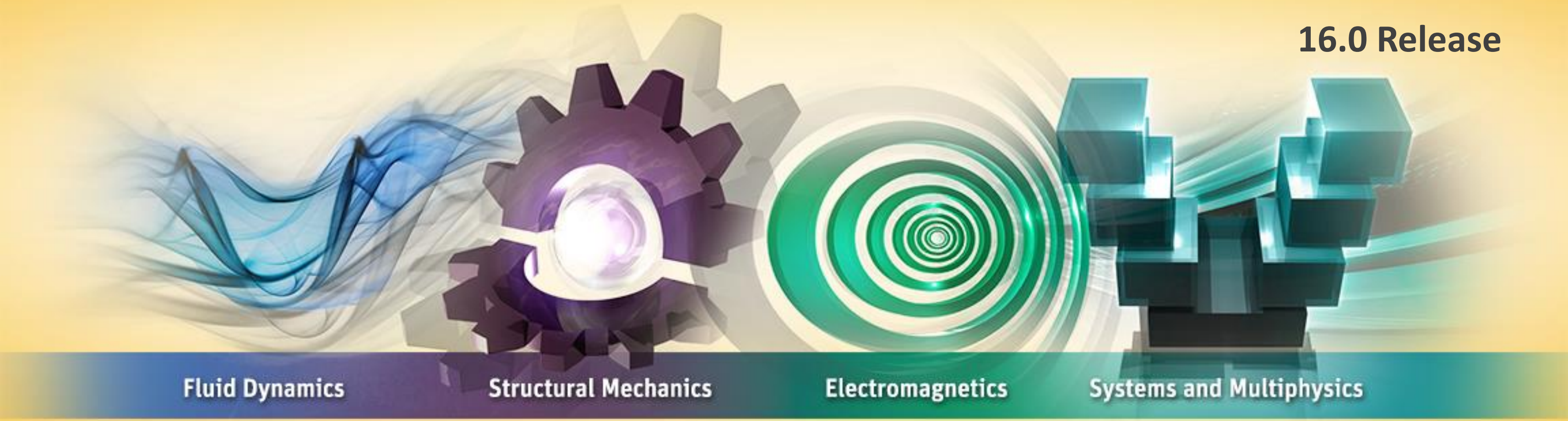
Introduction to ANSYS CFX

Appendix: RANS Turbulence Model

Model	Description
Standard k–ε	The baseline two-transport-equation model solving for k and ε. This is the default k–ε model. Coefficients are empirically derived; valid for fully turbulent flows only. Options to account for viscous heating, buoyancy, and compressibility are shared with other k–ε models.
RNG k–ε	A variant of the standard k–ε model. Equations and coefficients are analytically derived. Significant changes in the ε equation improves the ability to model highly strained flows. Additional options aid in predicting swirling and low Reynolds number flows.
Standard k–ω	A two-transport-equation model solving for k and ω, the specific dissipation rate (ε / k) based on Wilcox (1998). This is the default k–ω model. Demonstrates superior performance for wall-bounded and low Reynolds number flows. Shows potential for predicting transition. Options account for transitional, free shear, and compressible flows.
SST k–ω	A variant of the standard k–ω model. Combines the original Wilcox model for use near walls and the standard k–ε model away from walls using a blending function. Also limits turbulent viscosity to guarantee that $\tau_T \sim k$. The transition and shearing options are borrowed from standard k–ω. No option to include compressibility.
RSM	Reynolds stresses are solved directly using transport equations, avoiding isotropic viscosity assumption of other models. Use for highly swirling flows. Quadratic pressure-strain option improves performance for many basic shear flows.

Appendix: Prediction of First Cell Height

16.0 Release

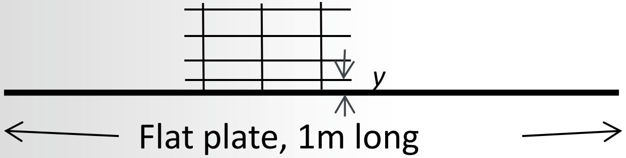


Introduction to ANSYS CFX

Example in predicting near-wall cell size

- During the pre-processing stage, you will need to know a suitable size for the first layer of grid cells (inflation layer) so that y+ is in the desired range.
- The actual flow-field will not be known until you have computed the solution (and indeed it is sometimes unavoidable to have to go back and remesh your model on account of the computed y+ values).
- To reduce the risk of needing to remesh, you may want to try and predict the cell size by performing a hand calculation at the start. For example:

Air at 20 m/s
ρ = 1.225 kg/m^3
μ = 1.8x10^{-5} kg/ms



The question is what height (y) should the first row of grid cells be. We will use SWF, and are aiming for $Y^+ \approx 50$

For a flat plate, Reynolds number gives Re = 1.4x10^6

(Recall from earlier slide, flow over a surface is turbulent when Re > 5x10^5)

Example in predicting near-wall cell size [2]

- A literature search suggests a formula for the skin friction on a plate:

$$C_f = 0.058\,\mathrm{Re}_l^{-0.2}$$

- Use this value to predict the wall shear stress τ_{w}:

$$\tau_w = \frac{1}{2} C_f \rho U_\infty^2$$

- From τ_w compute the velocity u_t:

$$U_\tau = \sqrt{\frac{\tau_w}{\rho}}$$

- Rearranging the equation shown previously for y^+ gives a formula for the first cell height, y, in terms of u_t
- In this example we are aiming for y+ of 50, hence our first cell height y should be approximately 1 mm.

Example in predicting near-wall cell size [3]

- **For Conjugate Heat Transfer Simulations one would need a y^+ value of 1. Let's estimate the first grid node for $y^+ = 1$:**

V= 20 m/s , ρ = 1.225 kg/m^3 , μ = 1.8x10^{-5} kg/ms

$$Re_l = \frac{\rho V L}{\mu}$$ → Re_l = 1.4x10^6

$$C_f = 0.058\, Re_l^{-0.2}$$ → C_f =0.0034

$$\tau_w = \frac{1}{2} C_f \rho U_\infty^2$$ → τ_w = 0.83 kg/ms^2

$$U_\tau = \sqrt{\frac{\tau_w}{\rho}}$$ → U_τ = 0.82 m/s

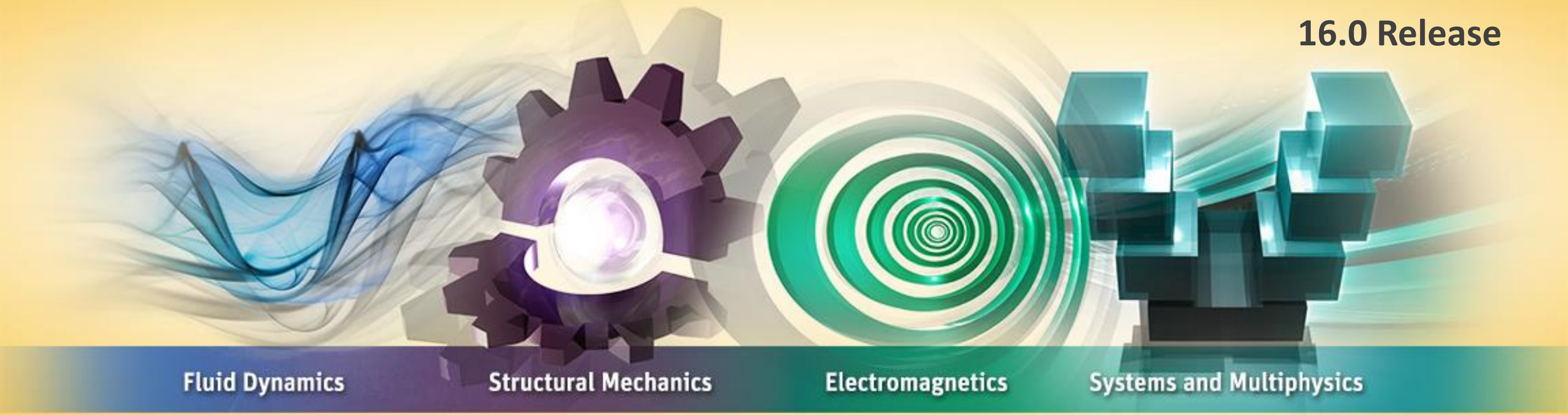
$$\upsilon = \frac{\mu}{\rho} = 1.469 \times 10^{-5}$$

$$y = \frac{y^+ \upsilon}{U_\tau} = 1.8 \times 10^{-5} m$$ → y = 0.02 mm

aiming for y+ of 1:
our first cell height y should be ≈ 0.02 mm

Appendix: Inlet Turbulence Conditions

16.0 Release



Introduction to ANSYS CFX

Realize Your Product Promise®

Inlet Turbulence Conditions

Default Intensity and Autocompute Length Scale

- The default turbulence intensity of 0.037 (3.7%) is used together with a computed length scale to approximate inlet values of k and ε. The length scale is calculated to take into account varying levels of turbulence.
- In general, the autocomputed length scale is not suitable for external flows

Intensity and Autocompute Length Scale

- This option allows you to specify a value of turbulence intensity but the length scale is still automatically computed. The allowable range of turbulence intensities is restricted to 0.1%-10.0% to correspond to very low and very high levels of turbulence accordingly.
- In general, the autocomputed length scale is not suitable for external flows

Intensity and Length Scale

- You can specify the turbulence intensity and length scale directly, from which values of k and ε are calculated

Inlet Turbulence Conditions [2]

Low Intensity = 1%

- This defines a 1% intensity and a viscosity ratio equal to 1

Medium Intensity = 5%

- This defines a 5% intensity and a viscosity ratio equal to 10
- This is the recommended option if you do not have any information about the inlet turbulence

High Intensity = 10%

- This defines a 10% intensity and a viscosity ratio equal to 100

Specified Intensity and Eddy Viscosity Ratio

- This defines a 10% intensity and a viscosity ratio equal to 100
- Use this feature if you wish to enter your own values for intensity and viscosity ratio

k and ε

- Specify the values of k and ε directly

Zero Gradient

- Use this setting for fully developed turbulence conditions