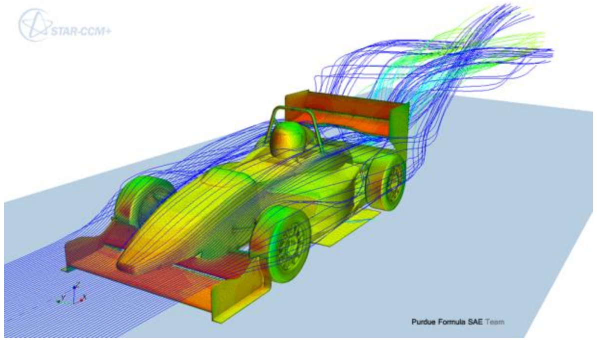
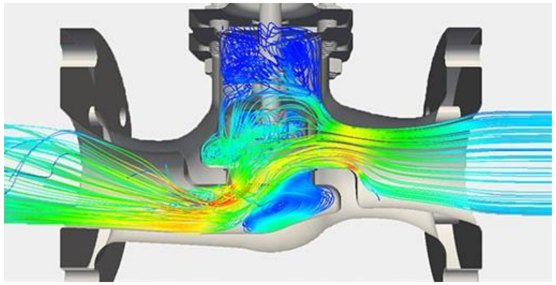
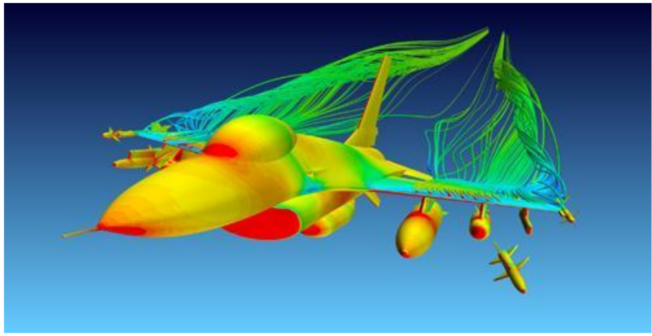
ME - 733
Computational Fluid Mechanics
Fluent Lecture 3
Turbulence Modelling







Dr./ Ahmed Nagib Elmekawy

Oct 28, 2018

Lecture Outline

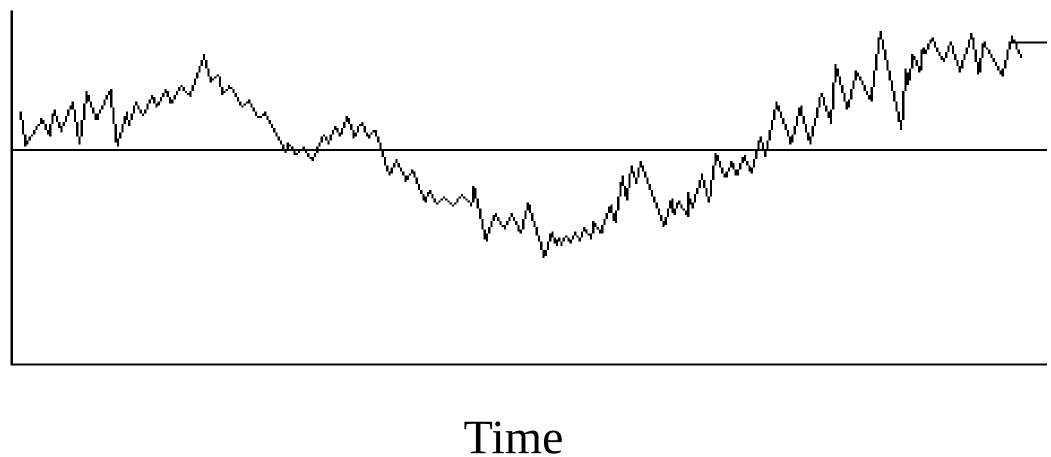
- Characterization of Turbulent Flows
- Computation of turbulent flows (DNS, LES and RANS approaches)
- Reynolds Averaged Navier-Stokes Equations (RANS)
- Reynolds Stress Tensor and the Closure Problem
- Turbulence Kinetic Energy (k) Equation
- Eddy Viscosity Models (EVM)
- Reynolds Stress Model
- Near-wall Treatments Options and Mesh Requirement
- Inlet Boundary Conditions
- Turbulence Modeling Guidelines

Characteristics of Turbulence

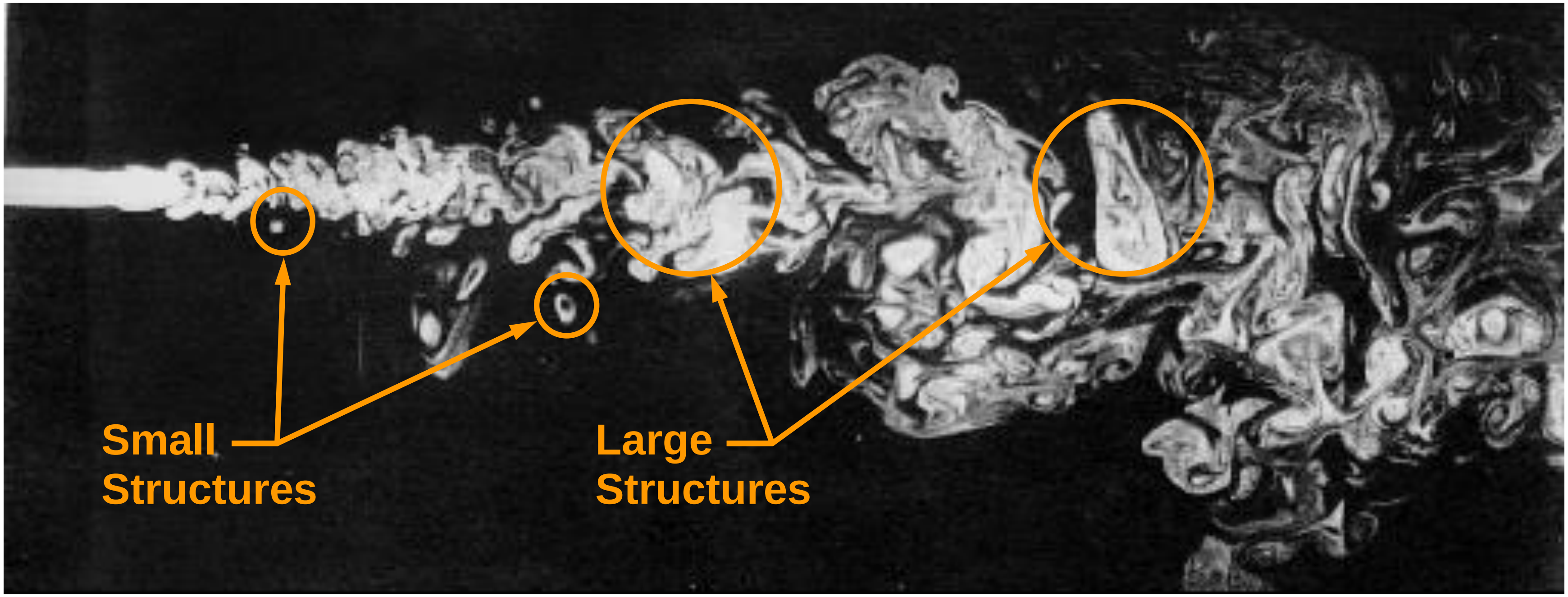
- Inherently unsteady, three dimensional and aperiodic swirling motions (fluctuations) resulting in enhancement of mixing, heat transfer and shear.

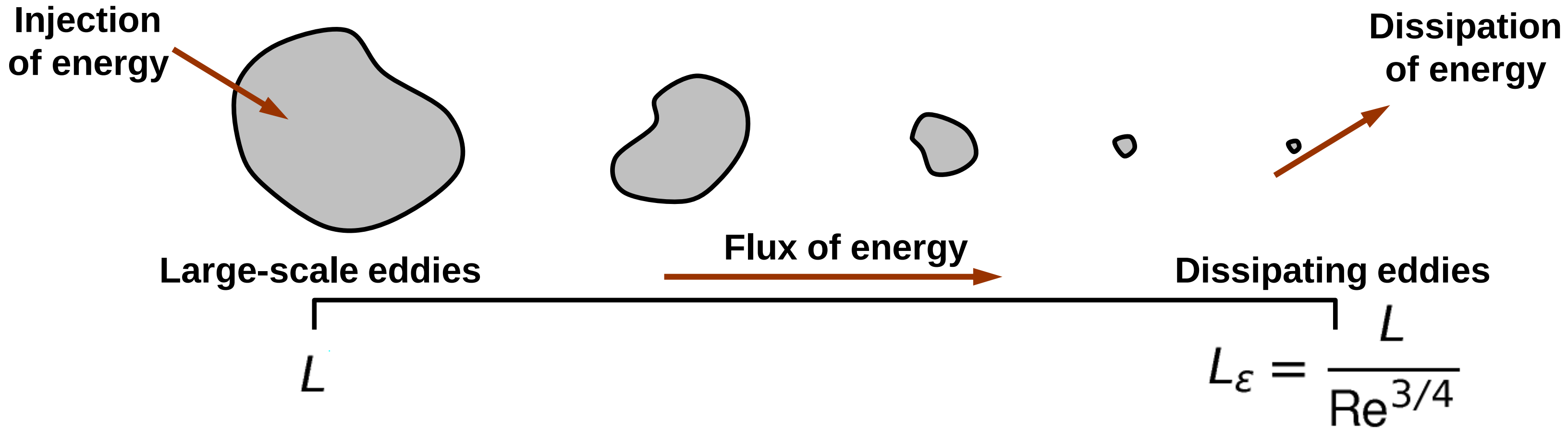
- Instantaneous fluctuations are random (unpredictable) both in space and in time. But statistical averaging of turbulence fluctuations results in accountable transport mechanisms

- Very sensitive to (or dependent on) initial conditions.



Turbulent Flow Structures





Energy Cascade (after Richardson, 1922)

Is the Flow Turbulent?

External Flows

$Re_x \geq 500{,}000$ along a surface

$Re_d \geq 20{,}000$ around an obstacle

Internal Flows

$Re_{d_h} \geq 2{,}300$

where $Re_L = \dfrac{\rho U L}{\mu}$

$L = x, d, d_h,$ etc.

Other factors such as free-stream turbulence, surface conditions, blowing, suction, and other disturbances etc. may cause transition to turbulence at lower Reynolds numbers

Natural Convection

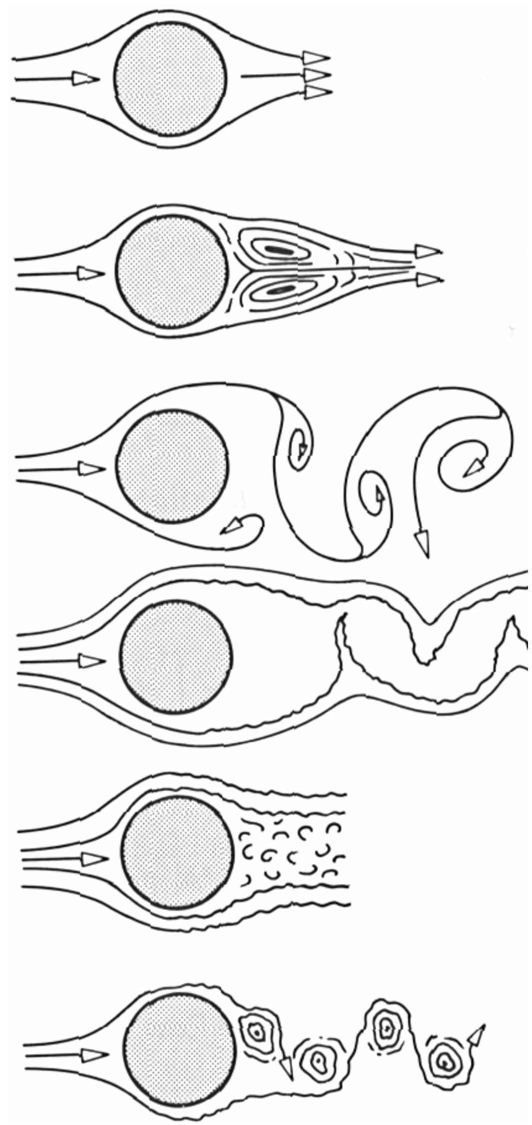
$$\frac{Ra}{Pr} \geq 10^9$$

where

$$Ra = \frac{\beta g L^3 \Delta T}{\nu \alpha} = \frac{\rho^2 c_p \beta g L^3 \Delta T}{\mu k} \quad \text{(Rayleigh number)}$$

$$Pr = \frac{\nu}{\alpha} = \frac{\mu c_p}{k} \quad \text{(Prandtl number)}$$

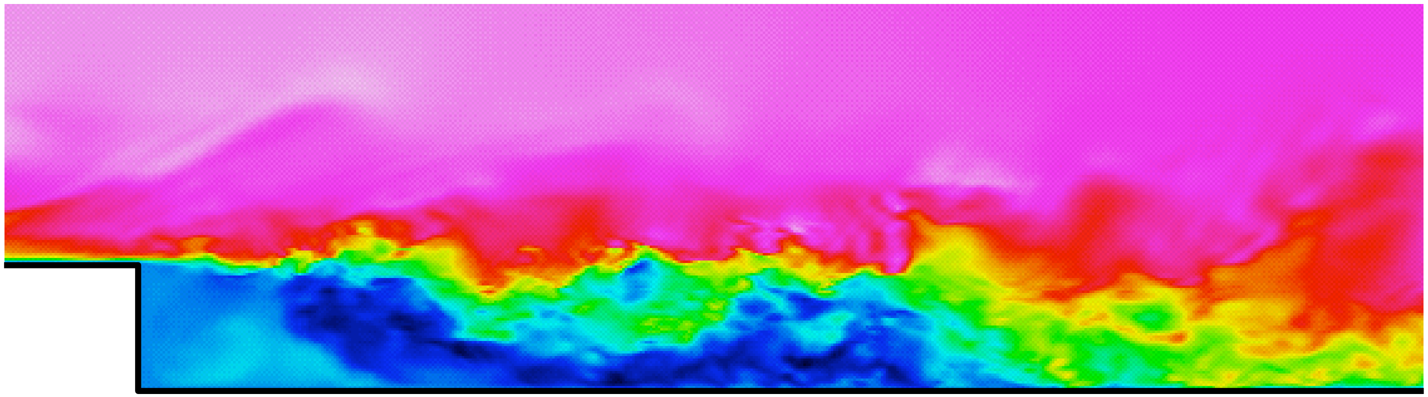
Reynolds Number Effects



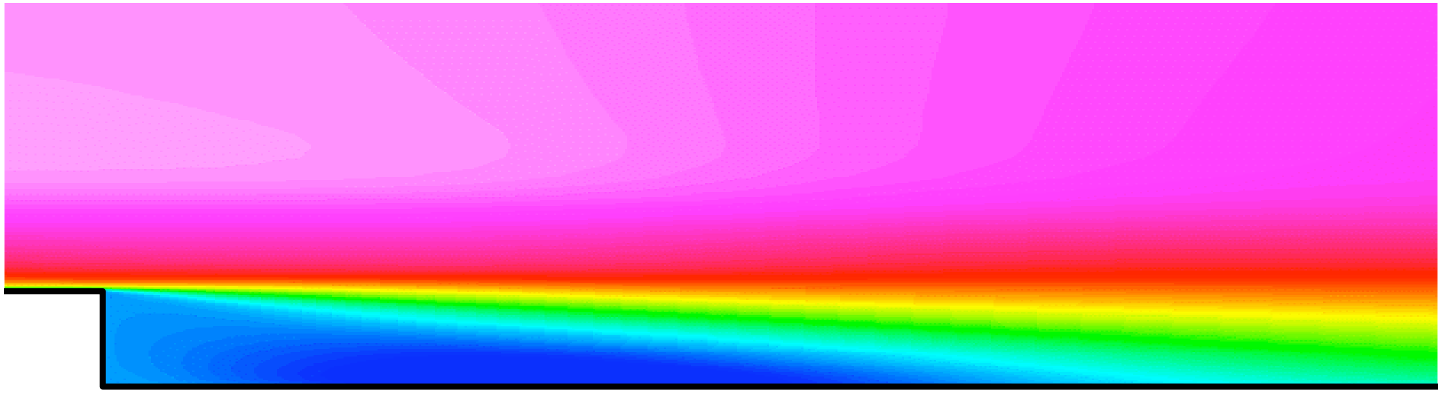
$Re < 5$	**Creeping flow (no separation)**
$5\text{-}15 < Re < 40$	**A pair of stable vortices in the wake**
$40 < Re < 150$	**Laminar vortex street**
$150 < Re < 3\times10^5$	**Laminar boundary layer up to the separation point, turbulent wake**
$3\times10^5 < Re < 3.5\times10^6$	**Boundary layer transition to turbulent**
$Re > 3.5\times10^6$	**Turbulent vortex street, but the separation is narrower than the laminar case**

Backward Facing Step

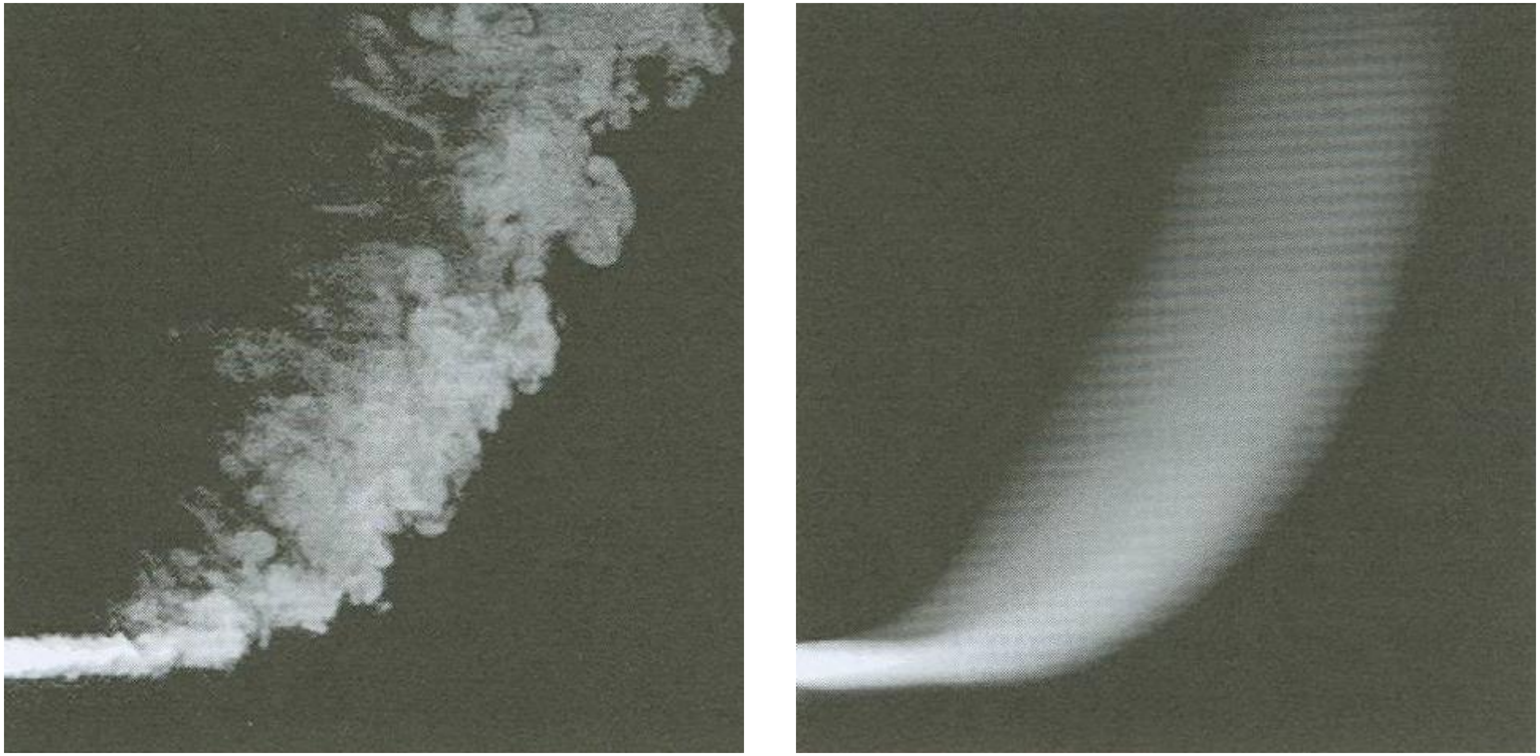
Instantaneous velocity contours



Time-averaged velocity contours



Plume in Cross Flow



From Su and Mungal in Durbin and Medic (2008)

- On the left is an instantaneous snap shot of a plume, on the right is a time-lapse picture which smoothes out the detailed structures (vortices) and shows only the averaged, diffused state of the same flow

Turbulence Modelling

- Most flows in practice are turbulent
- With increasing Re, smaller eddies
- Very fine grid necessary to describe all length scales
- Even the largest supercomputer does not have (yet) enough speed and memory to simulate turbulent flows of high Re.

Computational methods for turbulent flows:

- Direct Numerical Simulation (DNS)
- Large Eddy Simulation (LES)
- Reynolds-Averaged Navier-Stokes (RANS)

Direct Numerical Simulation (DNS)

- Discretize Navier-Stokes eq on a sufficiently fine grid for resolving all motions occurring in turbulent flow
- Does not use any models
- Equivalent to laboratory experiment

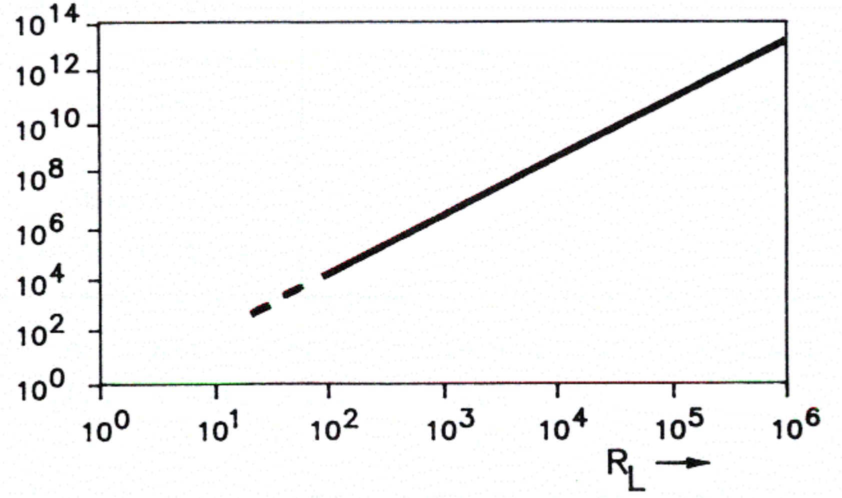
Relationship between length η of smallest eddies and the length L of largest eddies,

$$\frac{L}{\eta} \sim (\mathrm{Re}_L)^{\frac{3}{4}}$$

Direct Numerical Simulation (DNS)

Number of elements necessary to discretize the flow field

$$n_{elem} \sim (\mathrm{Re}_L)^{\frac{9}{4}}$$



In industrial applications, Re $> 10^6$ ⟹ $n_{elem} > 10^{13}$

RANS and LES Modelling

Large Eddy Simulation (LES)

- Only large eddies are computed
- Small eddies are modelled, subgrid-scale (SGS) models

Reynolds-Averaged Navier-Stokes (RANS)

- Variables decomposed in a mean part and a fluctuating part,
- Navier-Stokes equations averaged over time
- Turbulence models are necessary

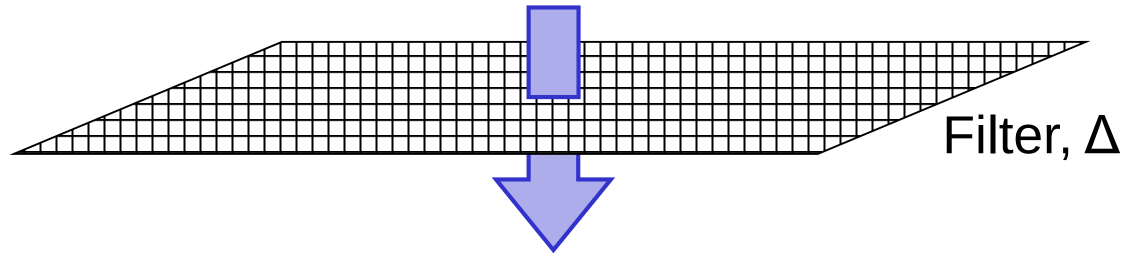
$$u = \bar{u} + u'$$

Large Eddy Simulation (LES)

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

Instantaneous component — Resolved Scale — Subgrid Scale

$$\frac{\partial u_i}{\partial t} + \frac{\partial \left(u_i u_j\right)}{\partial x_j} = -\frac{1}{\rho}\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\nu \frac{\partial u_i}{\partial x_j}\right)$$



Filtered N-S equation

$$\frac{\partial \overline{u}_i}{\partial t} + \frac{\partial \left(\overline{u}_i \overline{u}_j\right)}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \overline{p}}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\nu \frac{\partial \overline{u}_i}{\partial x_j}\right) - \frac{\partial \tau_{ij}}{\partial x_j}$$

$$\tau_{ij} = \rho\left(\overline{u_i u_j} - \overline{u}_i \overline{u}_j\right)$$

(Subgrid scale Turbulent stress)

- Spectrum of turbulent eddies in the Navier-Stokes equations is filtered:
  - The filter is a function of grid size
  - Eddies smaller than the grid size are removed and modeled by a subgrid scale (SGS) model.
  - Larger eddies are directly solved numerically by the filtered transient NS equation

Large Eddy Simulation (LES)

- Large Eddy Simulation (LES)
  - LES has been most successful for high-end applications where the RANS models fail to meet the needs. For example:
    - Combustion
    - Mixing
    - External Aerodynamics (flows around bluff bodies)
  - Subgrid scale (SGS) turbulent models:
    - Smagorinsky-Lilly model
    - Wall-Adapting Local Eddy-Viscosity (WALE)
    - Dynamic Smagorinsky-Lilly model
    - Dynamic Kinetic Energy Transport
  - Detached eddy simulation (DES) model
    - Choice of RANS in DES includes S-A, RKE, or SST

RANS Equations and the Closure Problem

- The time-averaging is defined as

$$\bar{f} = \lim_{T\to\infty} \frac{1}{T}\int_0^T f(x_i,t)\,dt$$

- The instantaneous field is defined as the sum of the mean and the fluctuating component, such as

$$p = \bar{p} + p' \qquad u_i = \bar{u}_i + u_i'$$

- By averaging the Navier-Stokes equations, we obtain the Reynolds averaged Navier-Stokes (RANS) equations:

$$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho \bar{u}_i)}{\partial x_i} = 0$$

Reynolds stress tensor, R_{ij}

$$\frac{\partial (\rho \bar{u}_i)}{\partial t} + \frac{\partial \left(\rho \bar{u}_i \bar{u}_j\right)}{\partial x_j} = -\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} - \frac{2}{3}\delta_{ij}\frac{\partial \bar{u}_m}{\partial x_m}\right)\right] + \frac{\partial}{\partial x_j}\left(-\rho\overline{u_i' u_j'}\right)$$

Reynolds Stress Tensor

$$R_{ij} = -\rho \begin{pmatrix} \overline{u'u'} & \overline{u'v'} & \overline{u'w'} \\ \overline{v'u'} & \overline{v'v'} & \overline{v'w'} \\ \overline{w'u'} & \overline{w'v'} & \overline{w'w'} \end{pmatrix}$$

- R_{ij} is a symmetric, second-order tensor; it comes from averaging the convective acceleration term in the momentum equation

- Reynolds stress thus provides the averaged effect of turbulent (randomly fluctuating) convection, which is highly diffusive

- Reynolds stress tensor in the RANS equations represents a combination of mixing due to turbulent fluctuation and smoothing by averaging.

The Closure Problem

- In order to close the RANS equations, the Reynolds stress tensor must be modeled.
  - Eddy Viscosity Models (EVM) – Based on the Boussinesq hypothesis that the Reynolds stress is proportional to the rate of strain of the time-averaged (mean) velocity. The proportionality constant is called Eddy Viscosity (or Turbulent Viscosity)

Eddy viscosity

$$-\rho\overline{u_i'u_j'} = \mu_t\left(\frac{\partial \overline{u}_i}{\partial x_j} + \frac{\partial \overline{u}_j}{\partial x_i}\right) - \frac{2}{3}\delta_{ij}\left(\rho k + \mu_t\frac{\partial \overline{u}_m}{\partial x_m}\right)$$

  - Reynolds Stress Models (RSM): By deriving and solving transport equations (PDEs) for each of the six distinct Reynolds stress terms (plus a dissipation-rate equation)

More on Eddy Viscosity

- Eddy viscosity is similar to molecular viscosity in its effect of diffusing momentum.

- Eddy viscosity is NOT a fluid property; it is a turbulent flow characteristic. Unlike an isothermal laminar flow in which viscosity is a constant, eddy viscosity varies with position throughout the flow field

- EVMs are the most widely used turbulence models for CFD.

- Some known limitations of the eddy viscosity concept:
  - Isotropy assumption is built in; however, there are many flows which are highly anisotropic (flows with large streamline curvature, impingement, and highly swirling flows, etc.).
  - Eddy viscosity models do not include dependence of the Reynolds stresses on the rate of rotation of the flow.
  - The assumption that Reynolds stress scales with the strain-rate tensor of the mean velocity is not always valid.

RANS Based Turbulence Models

RANS based models

One-Equation Model

Spalart-Allmaras

Two-Equation Models

Standard k–ε

RNG k–ε

Realizable k–ε

Standard k–ω

SST k–ω

4-Equation v2f

Reynolds Stress Model

***k–kl–ω* Transition Model**

SST Transition Model

Detached Eddy Simulation

Large Eddy Simulation

Increase in Computational Cost Per Iteration

The Spalart-Allmaras (S-A) Model

- Spalart-Allmaras is a low-cost RANS model solving a transport equation for a modified eddy viscosity
  - When in modified form, the eddy viscosity is easy to resolve near the wall
- Mainly intended for aerodynamic/turbomachinery applications with mild separation, such as supersonic/transonic flows over airfoils, boundary-layer flows, etc.
- Embodies a relatively new class of one-equation models where it is not necessary to calculate a length scale related to the local shear layer thickness
- Designed specifically for aerospace applications involving wall-bounded flows
  - Has been shown to give good results for boundary layers subjected to adverse pressure gradients.
  - Gaining popularity for turbomachinery applications.
- Limitations:
  - No claim is made regarding its applicability to all types of complex engineering flows.
  - Cannot be relied upon to predict the decay of homogeneous, isotropic turbulence.

The *k* Equation

- Turbulence kinetic energy k equation is used to determine the turbulence velocity scale:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho \overline{u}_i k)}{\partial x_i} = \underbrace{-\rho \overline{u_i' u_j'} \frac{\partial \overline{u}_i}{\partial x_j}}_{\mathbf{P_k}} - \rho \varepsilon + \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]$$

where P_k is the rate of production and ε is the dissipation rate.

- Production actually refers to the rate at which kinetic energy is transferred from the mean flow to the turbulent fluctuations (remember the energy cascade). P_k is the turbulent stress times mean strain rate, so physically it is the rate of work sustained by the mean flow on turbulent eddies

- Obviously P_k needs to be modeled due to the presence of R_{ij} in the term

The Standard k–ε Model

- The choice of ε as the second model equation. The ε equation is entirely modeled phenomenologically (not derived) as follows:

$$\frac{\partial(\rho\varepsilon)}{\partial t}+\frac{\partial(\rho\overline{u}_i\varepsilon)}{\partial x_i}=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial\varepsilon}{\partial x_j}\right]+C_{1\varepsilon}P_k\frac{\varepsilon}{k}-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k}$$

- The dissipation rate is related to *k* and a turbulence length scale as follows:

$$\varepsilon\sim\frac{k^{3/2}}{L_t}$$

- Together with the *k* equation, eddy viscosity can be expressed as:

$$\mu_t=\rho C_\mu L_t\sqrt{k}=\rho C_\mu\frac{k^2}{\varepsilon}$$

The Standard k–ε (SKE) Model

- SKE is the most widely-used engineering turbulence model for industrial applications
  - Model parameters are calibrated by using data from a number of benchmark experiments such as pipe flow, flat plate, etc.
  - Robust and reasonably accurate for a wide range of applications
  - Contains submodels for compressibility, buoyancy, combustion, etc.

- Known limitations of the SKE model:
  - Performs poorly for flows with larger pressure gradient, strong separation, high swirling component and large streamline curvature.
  - Inaccurate prediction of the spreading rate of round jets.
  - Production of *k* is excessive (unphysical) in regions with large strain rate (for example, near a stagnation point), resulting in very inaccurate model predictions.

Realizable k–ε and RNG k–ε Models

- Realizable k–ε (RKE) model (Shih):
  - Dissipation rate (ε) equation is derived from the mean-square vorticity fluctuation, which is fundamentally different from the SKE.
  - Several realizability conditions are enforced for Reynolds stresses.
  - Benefits:
    - Accurately predicts the spreading rate of both planar and round jets
    - Also likely to provide superior performance for flows involving rotation, boundary layers under strong adverse pressure gradients, separation, and recirculation

- RNG k–ε (RNG) model (Yakhot and Orszag):
  - Constants in the k–ε equations are derived analytically using renormalization group theory, instead of empirically from benchmark experimental data. Dissipation rate equation is modified.
  - Performs better than SKE for more complex shear flows, and flows with high strain rates, swirl, and separation

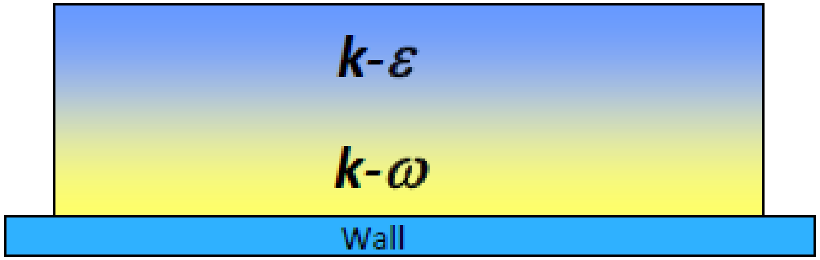
Standard k–ω and SST k–ω Models

- Standard k–ω (SKW) Model (Wilcox, 1998):
  - Robust low-Reynolds-number (LRN) formulation down to the viscous sublayer.
  - Several sub-models/options of k–ω: compressibility effects, transitional flows and shear-flow corrections.
  - Improved behavior under adverse pressure gradient.
  - SKW is more sensitive to free-stream conditions.
  - Most widely adopted in the aerospace and turbomachinery communities.

- Shear Stress Transport k–ω (SSTKW) model (Menter)
  - The SST k–ω model uses a blending function to gradually transition from the standard k–ω model near the wall to a high-Reynolds-number version of the k–ε model in the outer portion of the boundary layer.
  - Contains a modified turbulent viscosity formulation to account for the transport effects of the principal turbulent shear stress.
  - SST model generally gives accurate prediction of the onset and the size of separation under adverse pressure gradient.

SST Model

- **Shear Stress Transport (SST) Model**
  - The SST model is a hybrid two-equation model that combines the advantages of both k-ε and k-ω models
    - The k-ω model performs much better than k-ε models for boundary layer flows
    - Wilcox' original k-ω model is overly sensitive to the freestream value (BC) of ω, while the k-ε model is not prone to such problems



  - The k-ε and k-ω models are blended such that the SST model functions like the k-ω close to the wall and the k-ε model in the freestream

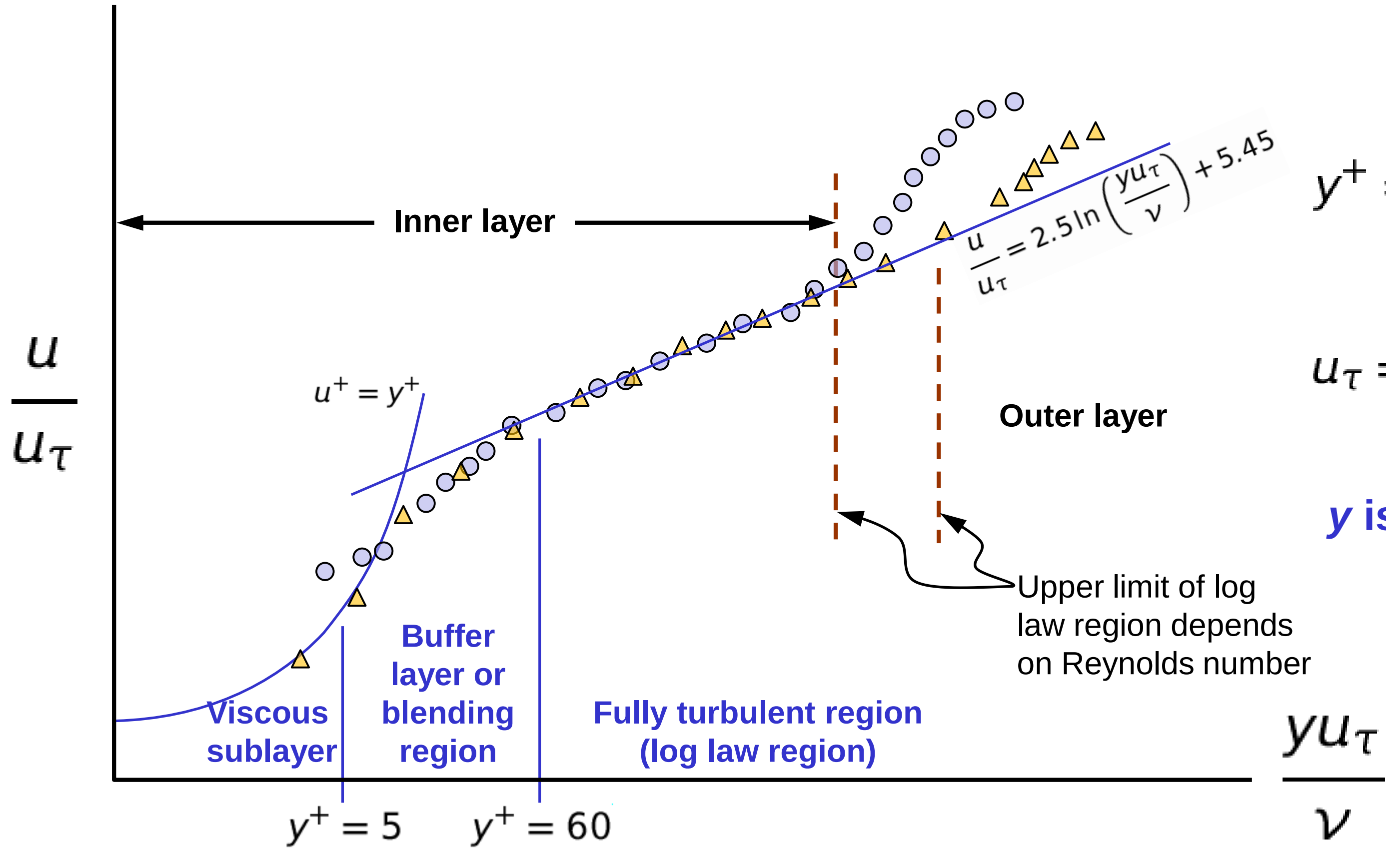
SST is a good compromise between k-ε and k-ω models

Reynolds Stress Model (RSM)

- Recall the limitations and weakness of eddy viscosity models:
  - Linear algebraic stress-strain relationship results in poor performance where stress transport is important, including non-equilibrium flows, separating and reattaching flows, etc.
  - Inability to account for extra strain due to streamline curvature, rotation, and highly skewed flows, etc.
  - Poor performance where turbulence is highly anisotropic (e.g., in flows normal stresses play important a role) and/or 3D effects are present.

- Attempting to avoid these shortcomings, transport equations for the six distinct Reynolds stress components are derived by averaging the products of velocity fluctuations and Navier-Stokes equations. A turbulent dissipation rate equation is also needed.
  - RSM is most suitable for highly anisotropic, three dimensional flows (where EVMs perform poorly). The computational cost is higher.
  - Currently RSMs still do not always provide indisputable superior performances over EVMs.

The Universal Law of The Wall



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

$$u_\tau = \sqrt{\frac{\tau_{\text{wall}}}{\rho}}$$

y **is the normal distance from the wall.**

- Dimensionless velocity profiles plotted in the near-wall coordinates
- The linear section in the semi-log plot is called the universal law of the wall layer, or log law layer, for equilibrium turbulent boundary layers (TBL)

Wall Modeling Strategies

- In the near-wall region, the solution gradients are very high, but accurate calculations in the near-wall region are paramount to the success of the simulation. The choice is between:

A) Using Wall Functions

B) Resolving the Viscous Sublayer

Using Wall Functions

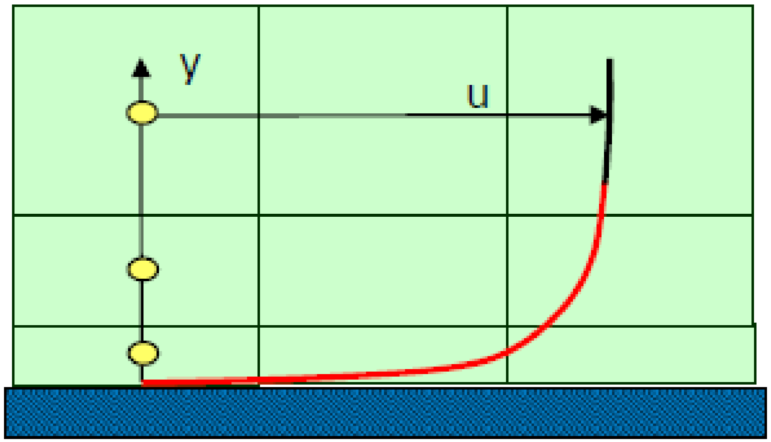
- Wall functions utilize the predictable dimensionless boundary layer profiles shown previously on slide 27 to determine the conditions at the centroid of the wall adjacent mesh cell.

Resolving the viscous sublayer

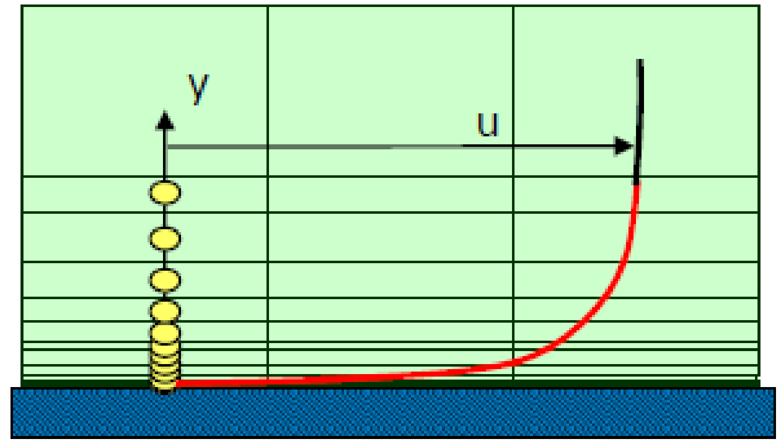
- First grid cell needs to be at about $y^+ \approx 1$ and a prism layer mesh with growth rate no higher than ≈ 1.2 should be used
  - These are not magic numbers – this guideline ensures the mesh will be able to adequately resolve gradients in the sublayer
- This will add significantly to the mesh count (see next slide)
- Generally speaking, if the forces or heat transfer on the wall are key to your simulation (aerodynamic drag, turbomachinery blade performance, heat transfer) this is the approach you will take and the recommended turbulence model for most cases is SST k-ω

Mesh Resolution Near the Wall

- Fewer nodes are needed normal to the wall when logarithmic-based wall functions are used (compared to resolving the viscous sublayer with the mesh)



Logarithmic-based Wall functions used to resolve boundary layer

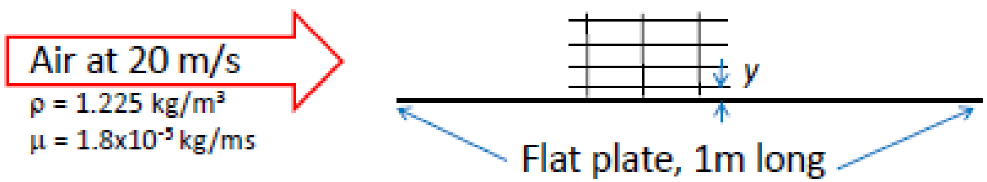


Viscous sublayer resolving approach used to resolve boundary layer

Boundary layer   First node wall distance is reflected by y^+ value

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:



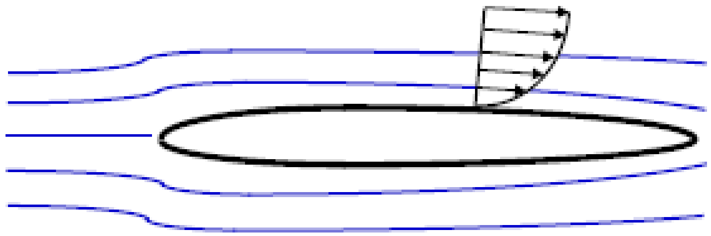
The question is what height (y) should the first row of grid cells be. We will use SWF, and are aiming for $Y^+ \approx 50$

- For a flat plate, Reynolds number ($\mathrm{Re}_l = \dfrac{\rho V L}{\mu}$) gives $\mathrm{Re}_l = 1.4\times10^6$

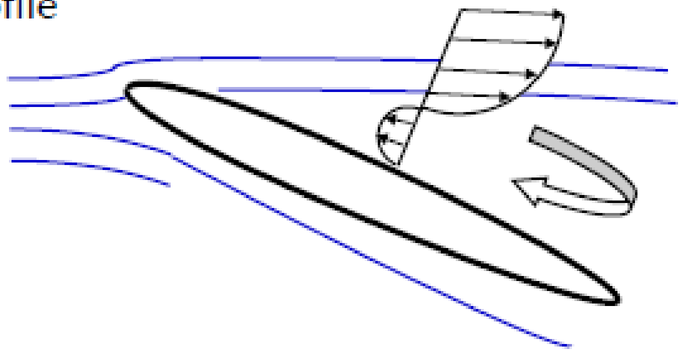
Recall from earlier slide, flow over a surface is turbulent when $\mathrm{Re}_L > 5\times10^5$

Limitations of Wall Functions

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



Wall functions applicable



Wall functions not applicable

Non-equilibrium wall functions have been developed in Fluent to address this situation but they are very empirical. Resolving the viscous sublayer with the mesh is recommended if affordable

- In these cases logarithmic-based wall functions should not be used
- Instead, directly resolving the viscous sublayer with the mesh can provide accurate results

The Need for Near-Wall Treatment

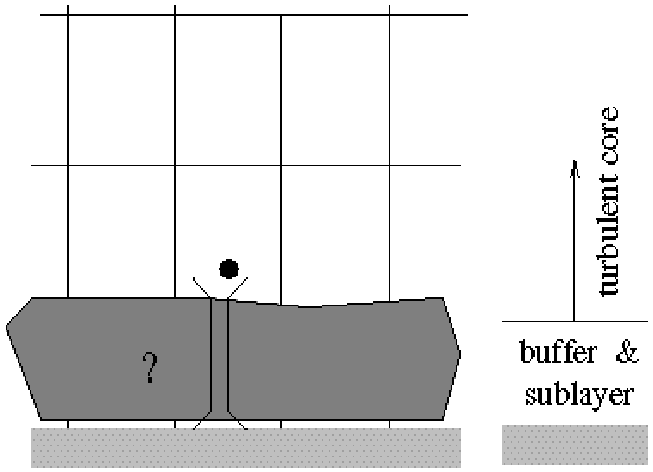
- In the near-wall region, the turbulent boundary layer is very thin and the solution gradients are very high, but accurate calculations in the near-wall region are paramount to the success of the simulation

- We can use a very fine mesh to resolve this region (viscous sublayer region), but it is very costly for industrial CFD applications

- For equilibrium turbulent boundary layers, the Universal Law of the Wall (or "log law") can be used in order to alleviate the problem:
  - Velocity profile and wall shear stress obtained from the log law are used to set the boundary values of stresses for the wall-adjacent cells.
  - The equilibrium assumption is used to set boundary conditions for turbulent kinetic energy (k), dissipation rate (ε) or specific dissipation rate (ω).
  - Non-equilibrium wall function method attempts to improve the results for flows with higher pressure gradients, separations, reattachment and stagnation
  - Similar log-laws are also constructed for the energy and species equations
  - Benefit: Wall functions allow the use of a relatively coarse mesh in the near-wall region thereby reduce the computational cost.

Near-Wall Mesh Requirement

- Standard and Non-Equilibrium Wall Functions:
  - Start calculation from log law religion so the Wall adjacent cells should have y^+ values between 30 and 300–500.
  - The mesh expansion ratio should be small (no larger than around 1.2).
  - Use the viscus sub layer law to get the value of the velocity that will be used as a boundary condition
  - y+ calculated after solution only not before and should be the minimum number of y+ higher than 30 to use this wall treatment function



Scalable Wall Functions

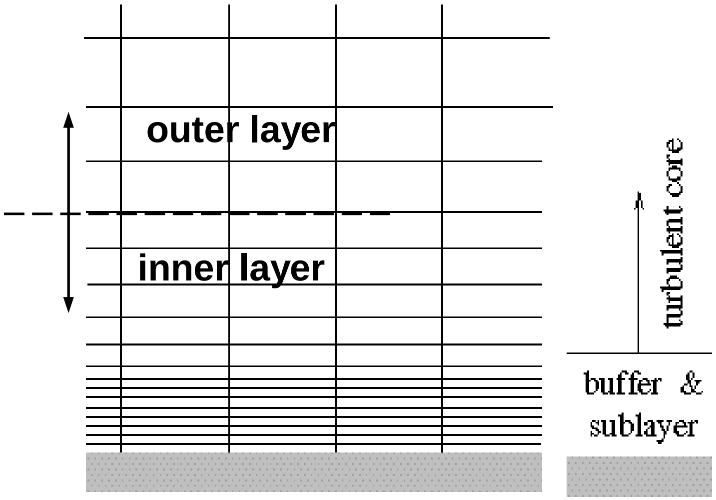
- In practice, many users often fail to maintain $30 < y^+ < 30–500$

- Conventional wall functions are a major limiting factor in accuracy. Wall functions are very sensitive to the first cell placement and the near-wall mesh; mesh refinement does not guarantee to deliver results with increasing accuracy, while EWT is still too costly.

- Scalable Wall Functions
  - For k–ε models, the scalable wall functions method assumes that the wall surface coincides with the edge of the viscous sublayer ($y^* = 11.26$). Hence fluid cells are always above the viscous sublayer, and inconsistency of predictions due to near-wall mesh refinement is avoided. (Note: in the k–ω, SST and S-A models, near-wall treatment is handled automatically by the solver; scalable wall functions are not available).

Enhanced Wall Treatment Option

- Combines a blended law-of-the wall and a two-layer zonal model.
- Suitable for low-Re flows or flows with complex near-wall phenomena.
- k–ε turbulence models are modified for the inner layer.
- Generally requires a fine near-wall mesh capable of resolving the viscous sublayer
  ($y^+ < 5$, and a minimum of 10–15 cells across the “inner layer”(viscous sublayer, the buffer layer and the log-law layer)



Summary on Near-Wall Treatment

- Wall Functions are still the most affordable boundary treatment for many industrial CFD applications
- In the k–ε family, scalable wall functions is the recommended setup option for models using standard wall functions
- Standard wall function works well with simple shear flows, and non-equilibrium wall function improves the results for flows with stronger pressure gradient and separation
- Enhanced wall treatment is used for more complex flows where log law may not apply (for example, non-equilibrium wall shear layers or the bulk Reynolds number is low)

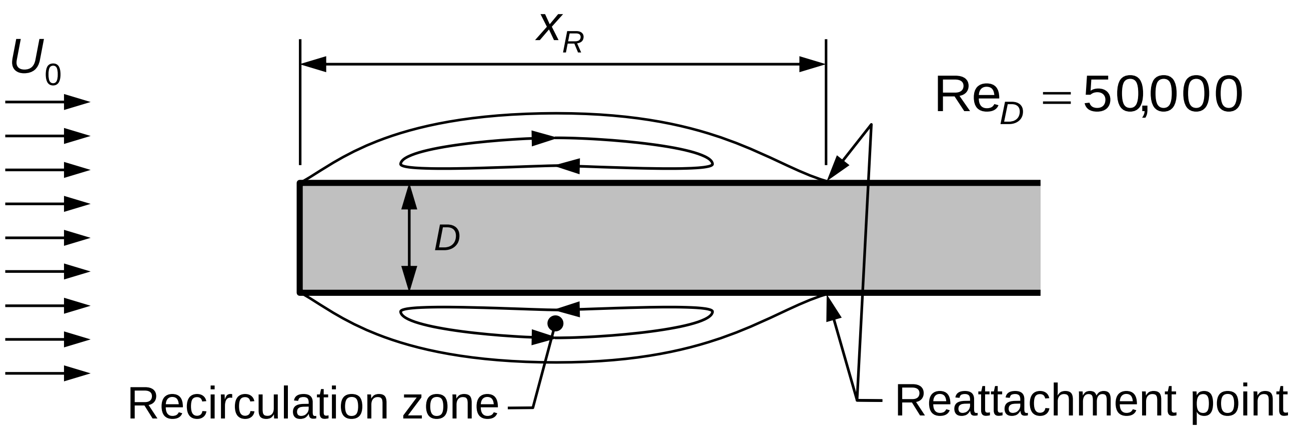
Inlet Boundary Conditions

- When turbulent flow enters a domain at inlets or outlets (backflow), boundary conditions for *k*, ε, ω and/or $-\rho \overline{u_i' u_j'}$ must be specified, depending on which turbulence model has been selected

- Four methods for directly or indirectly specifying turbulence parameters:
  - Explicitly input k, ε, ω, or Reynolds stress components (this is the only method that allows for profile definition)
  - Turbulence intensity and length scale
    - Length scale is related to size of large eddies that contain most of energy
      - For boundary layer flows: $l \approx 0.4\delta_{99}$
      - For flows downstream of grid: $l \approx$ opening size
  - Turbulence intensity and hydraulic diameter (primarily for internal flows)
  - Turbulence intensity and viscosity ratio (primarily for external flows)

Example #1 – Turbulent Flow Past a Blunt Flat Plate

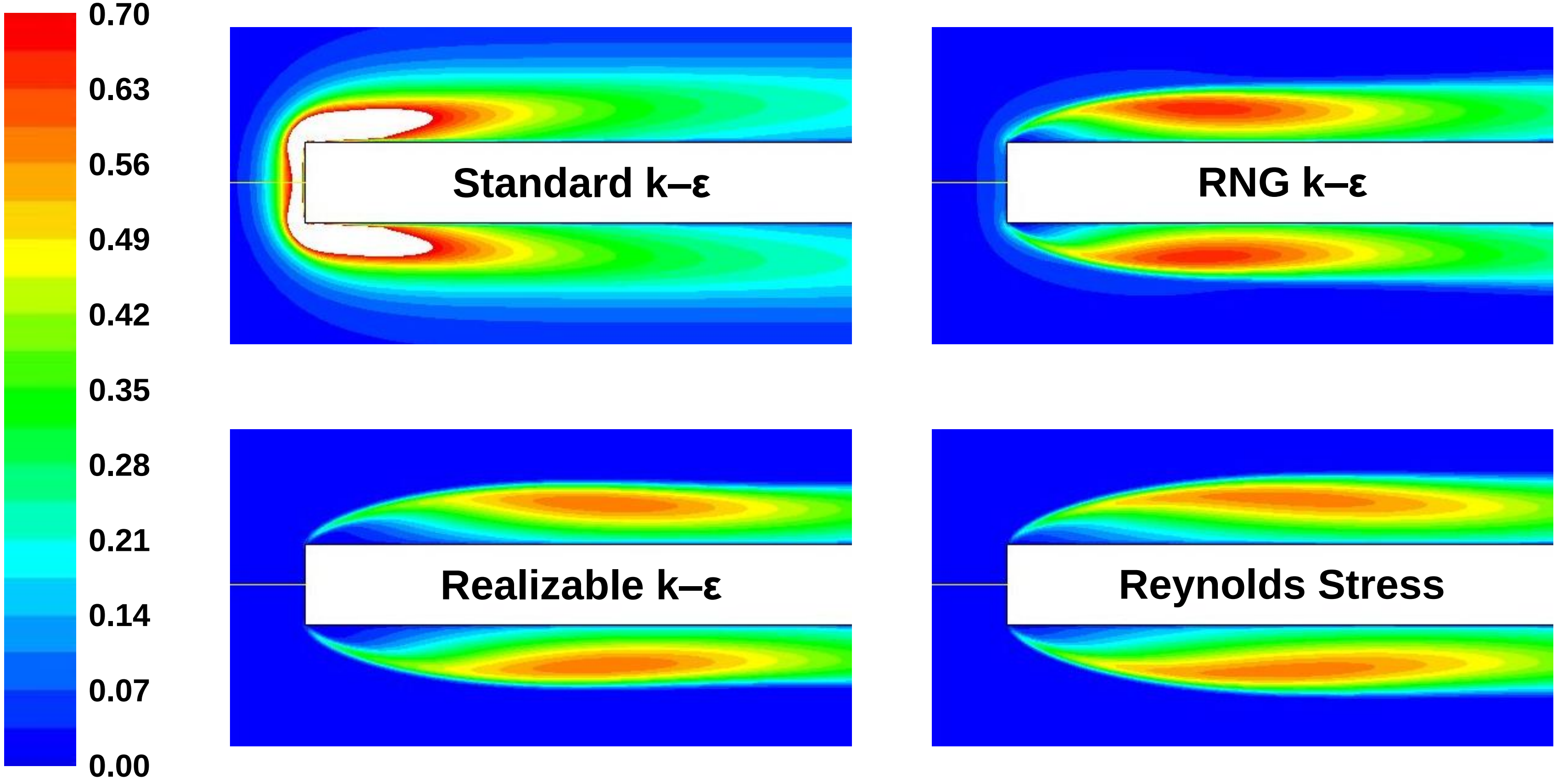
- Turbulent flow past a blunt flat plate was simulated using four different turbulence models.
  - 8,700 cell quad mesh, graded near leading edge and reattachment location.
  - Non-equilibrium boundary layer treatment



N. Djilali and I. S. Gartshore (1991), “Turbulent Flow Around a Bluff Rectangular Plate, Part I: Experimental Investigation,” *JFE*, Vol. 113, pp. 51–59.

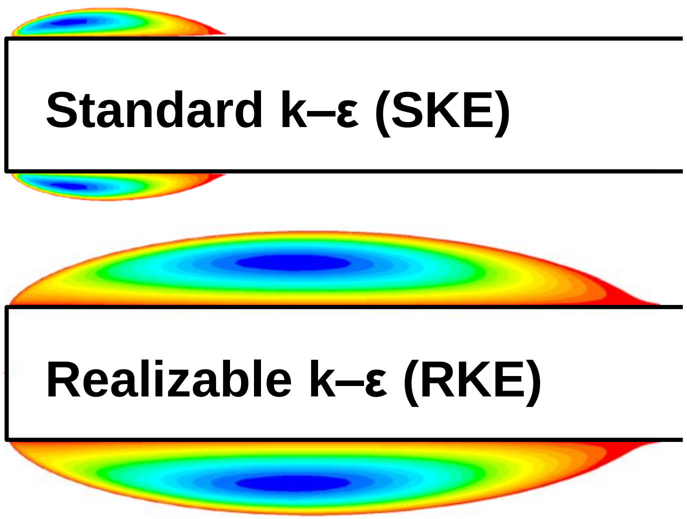
Example #1 – Turbulent Flow Past a Blunt Flat Plate

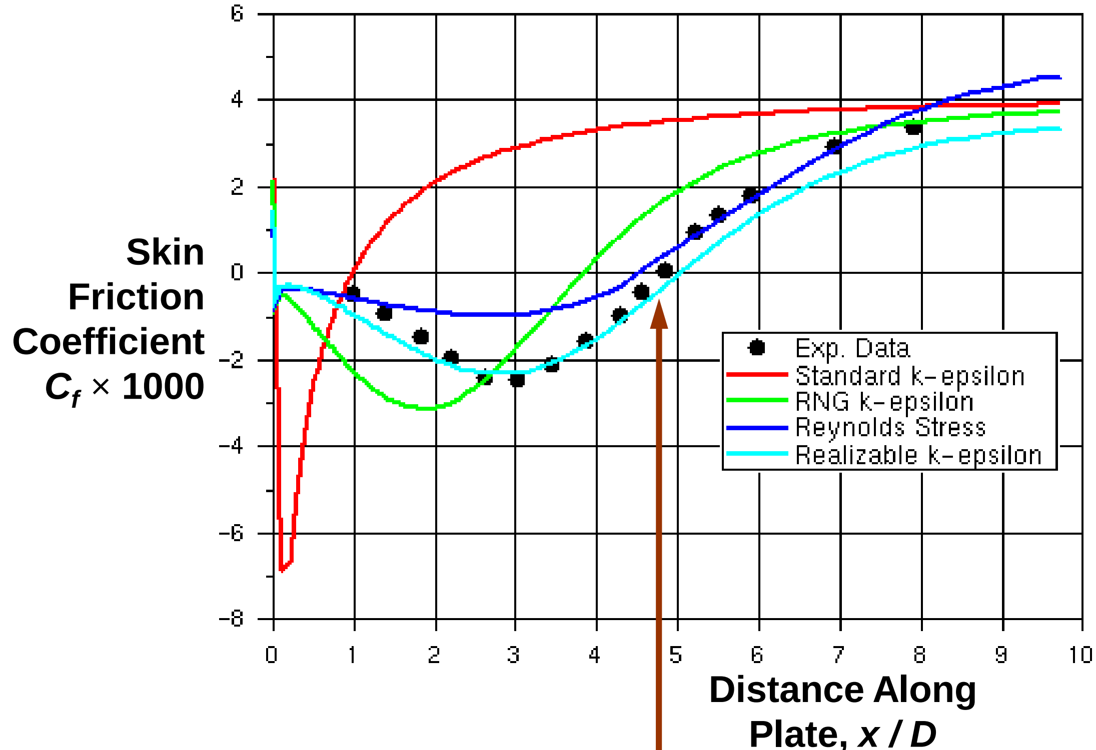
Contours of Turbulent Kinetic Energy (m^2/s^2)



Example #1 – Turbulent Flow Past a Blunt Flat Plate

Predicted separation bubble:



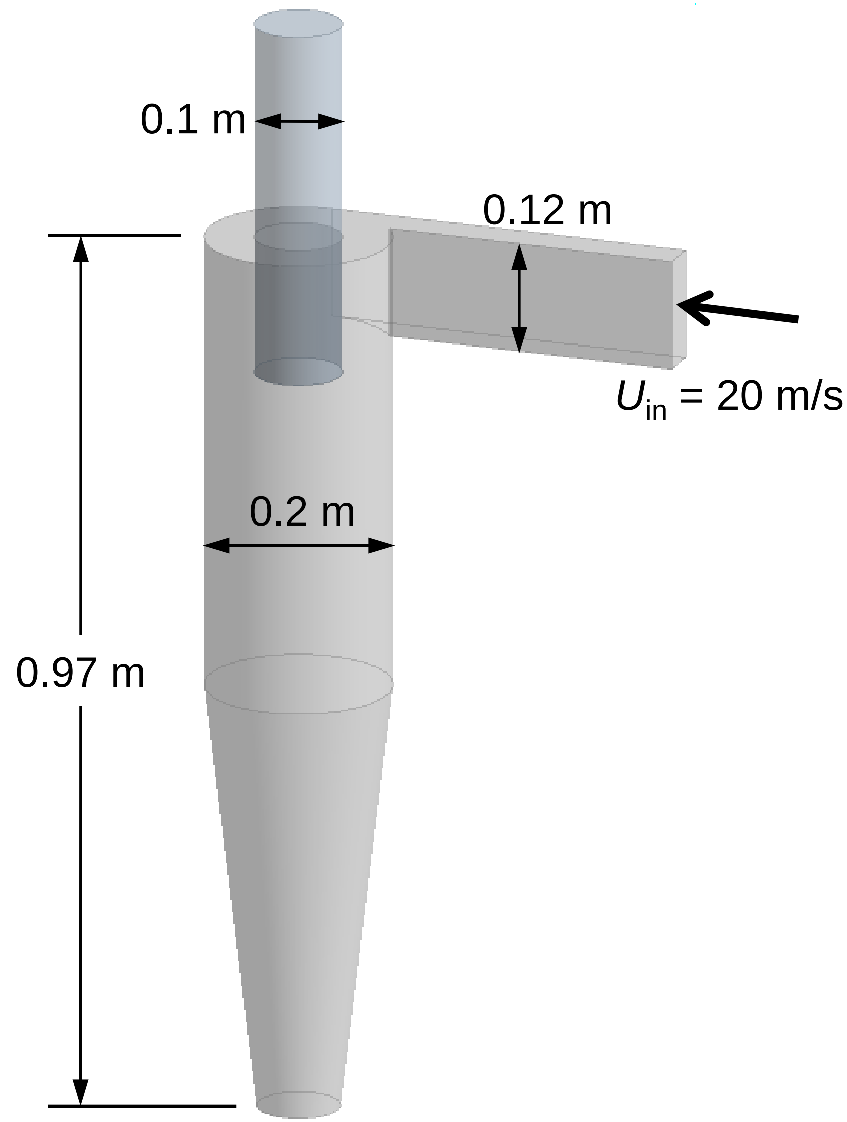


SKE severely underpredicts the size of the separation bubble, while RKE predicts the size exactly.

Experimentally observed reattachment point is at *x* / *D* = 4.7

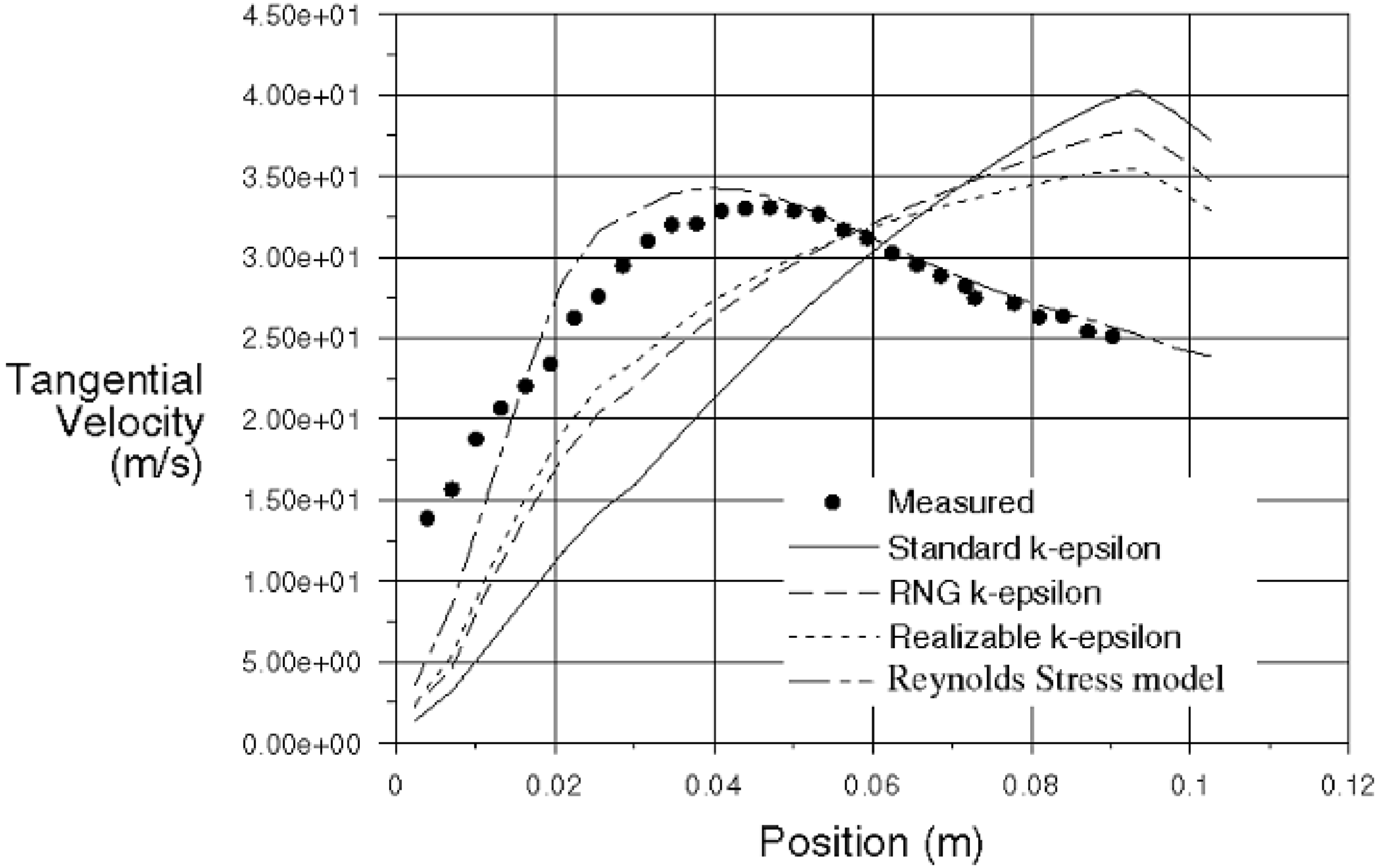
Example #2 – Turbulent Flow in a Cyclone

- 40,000-cell hexahedral mesh
- High-order upwind scheme was used.
- Computed using SKE, RNG, RKE and RSM (second moment closure) models with the standard wall functions
- Represents highly swirling flows ($W_{max} = 1.8\ U_{in}$)



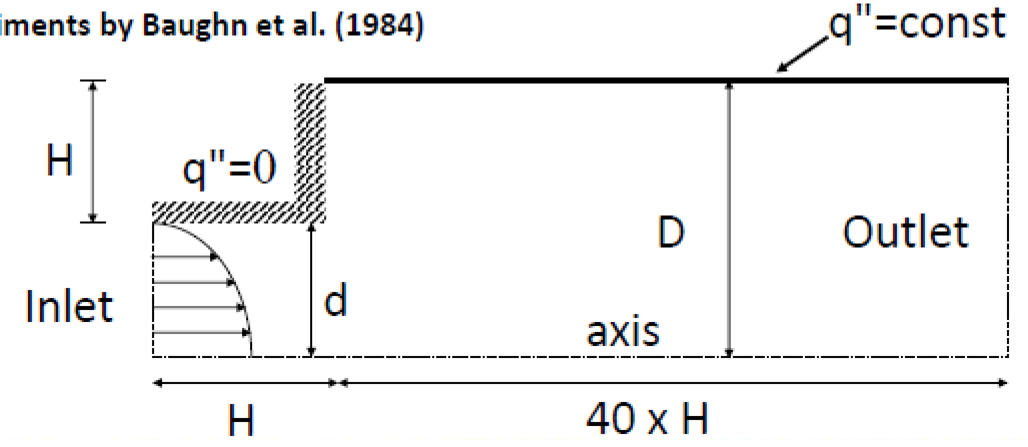
Example #2 – Turbulent Flow in a Cyclone

- Tangential velocity profile predictions at 0.41 m below the vortex finder



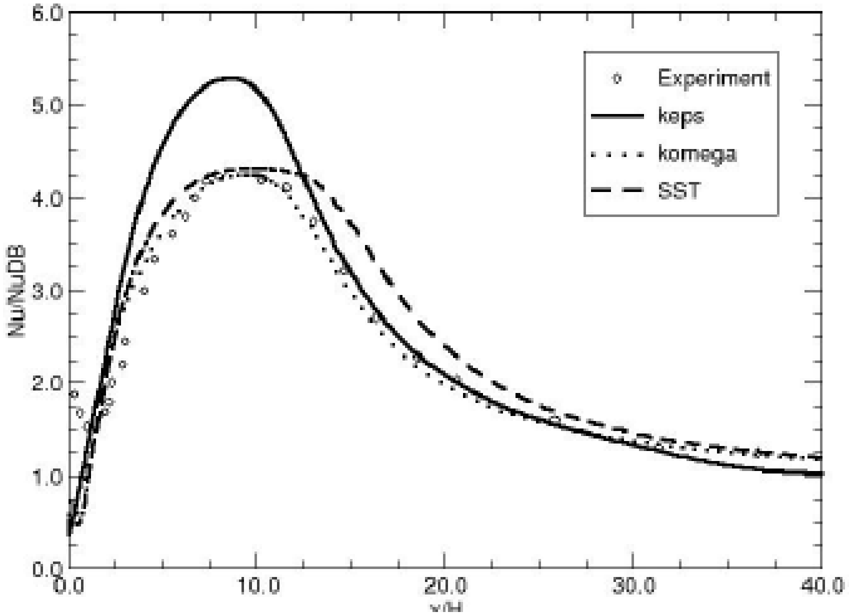
Example #3 – Pipe Expansion with Heat Transfer

- **Reynolds Number Re_D= 40750**
- **Fully Developed Turbulent Flow at Inlet**
- **Experiments by Baughn et al. (1984)**



Example #3 – Pipe Expansion with Heat Transfer

- Plot shows dimensionless distance versus Nusselt Number
- Best agreement is with SST and k-omega models which do a better job of capturing flow recirculation zones accurately



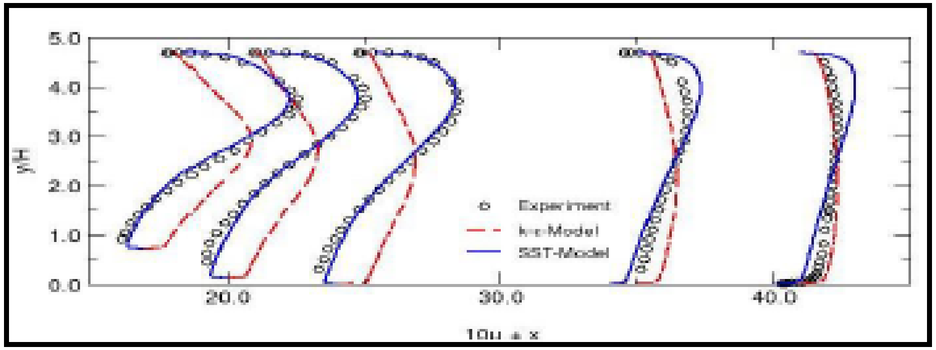
Example #4 – Diffuser

Shear Stress Transport (SST) Model

- It accounts more accurately for the transport of the turbulent shear stress, which improves predictions of the onset and the amount of flow separation compared to k-ε models

Standard k-ε fails to predict separation

SST result and experiment



Experiment Gersten et al.

Summary – Turbulence Modeling Guidelines

- Successful turbulence modeling requires engineering judgment of:
  - Flow physics
  - Computer resources available
  - Project requirements
    - Accuracy
    - Turnaround time
  - Choice of Near-wall treatment
- Modeling procedure
  - Calculate characteristic Reynolds number and determine whether or not the flow is turbulent.
  - If the flow is in the transition (from laminar to turbulent) range, consider the use of one of the turbulence transition models.
  - Estimate wall-adjacent cell centroid y^+ before generating the mesh.
  - Prepare your mesh to use wall functions except for low-Re flows and/or flows with complex near-wall physics (non-equilibrium boundary layers).
  - Begin with RKE (realizable k-ε) and change to S-A, RNG, SKW or SST if needed. Check the following tables as a guide for your choice.
  - Use RSM for highly swirling, 3-D, rotating flows.
  - Remember that there is no single, superior turbulence model for all flows!

RANS Models Descriptions

Model	**Description**
Spalart – Allmaras	A single transport equation model solving directly for a modified turbulent viscosity. Designed specifically for aerospace applications involving wall-bounded flows on a fine near-wall mesh. Option to include strain rate in k production term improves predictions of vortical flows.
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.
Realizable k–ε	A variant of the standard k–ε model. Its “realizability” stems from changes that allow certain mathematical constraints to be obeyed which ultimately improves the performance of this model.
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.
Reynolds Stress	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.

RANS Models Behavior Summary

Model	Behavior and Usage
Spalart – Allmaras	Economical for large meshes. Performs poorly for 3D flows, free shear flows, flows with strong separation. Suitable for mildly complex (quasi-2D) external/internal flows and boundary layer flows under pressure gradient (e.g. airfoils, wings, airplane fuselages, missiles, ship hulls).
Standard k–ε	Robust. Widely used despite the known limitations of the model. Performs poorly for complex flows involving severe pressure gradient, separation, strong streamline curvature. Suitable for initial iterations, initial screening of alternative designs, and parametric studies.
RNG k–ε	Suitable for complex shear flows involving rapid strain, moderate swirl, vortices, and locally transitional flows (e.g. boundary layer separation, massive separation, and vortex shedding behind bluff bodies, stall in wide-angle diffusers, room ventilation).
Realizable k–ε	Offers largely the same benefits and has similar applications as RNG. Possibly more accurate and easier to converge than RNG.
Standard k–ω	Superior performance for wall-bounded boundary layer, free shear, and low Reynolds number flows. Suitable for complex boundary layer flows under adverse pressure gradient and separation (external aerodynamics and turbomachinery). Can be used for transitional flows (though tends to predict early transition). Separation is typically predicted to be excessive and early.
SST k–ω	Offers similar benefits as standard k–ω. Dependency on wall distance makes this less suitable for free shear flows.
Reynolds Stress	Physically the most sound RANS model. Avoids isotropic eddy viscosity assumption. More CPU time and memory required. Tougher to converge due to close coupling of equations. Suitable for complex 3D flows with strong streamline curvature, strong swirl/rotation (e.g. curved duct, rotating flow passages, swirl combustors with very large inlet swirl, cyclones).