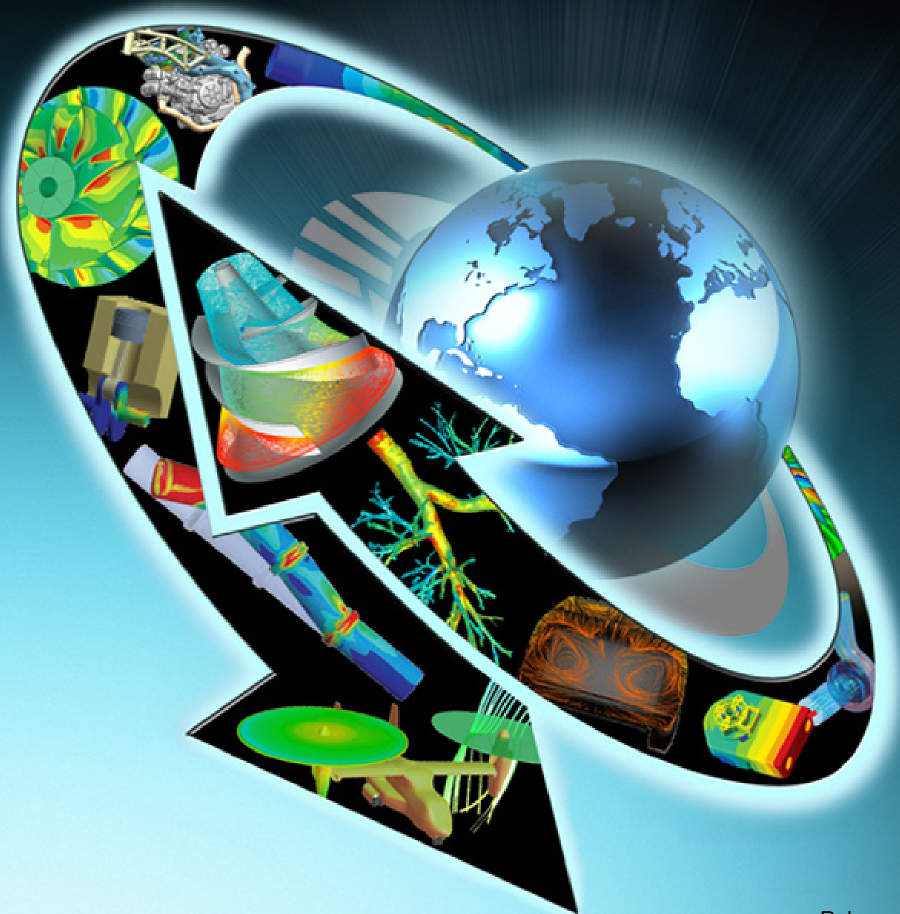
Customer Training Material

Lecture 6

Turbulence Modeling



Introduction to ANSYS FLUENT

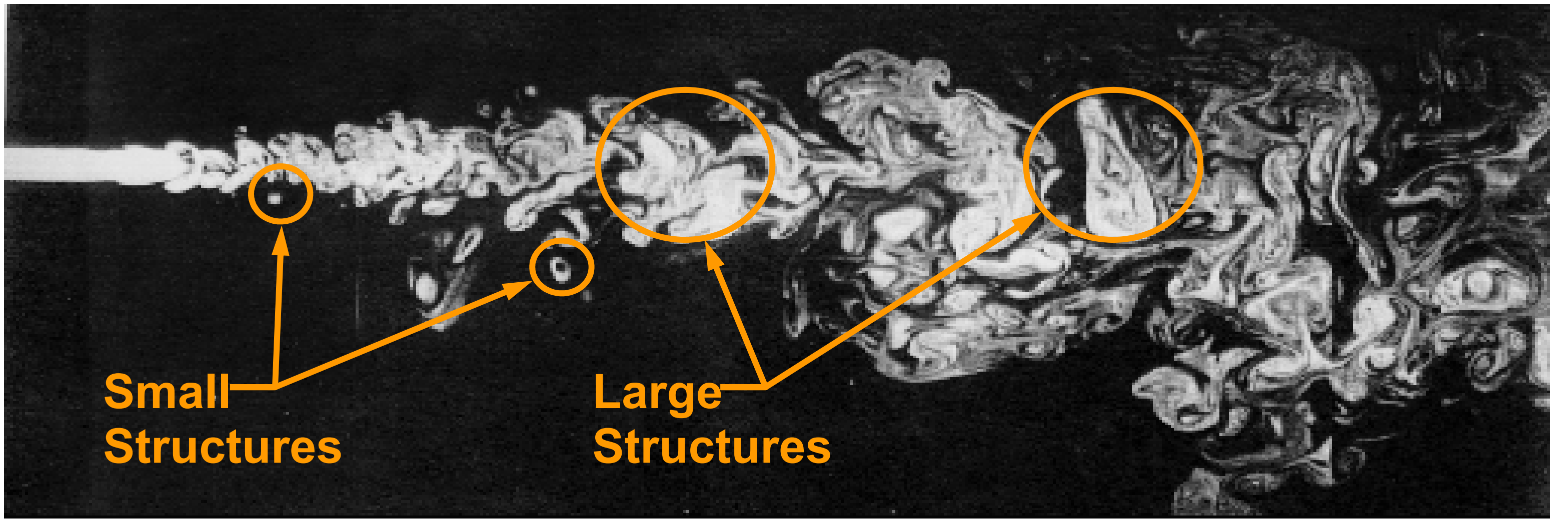
Release 13.0
December 2010

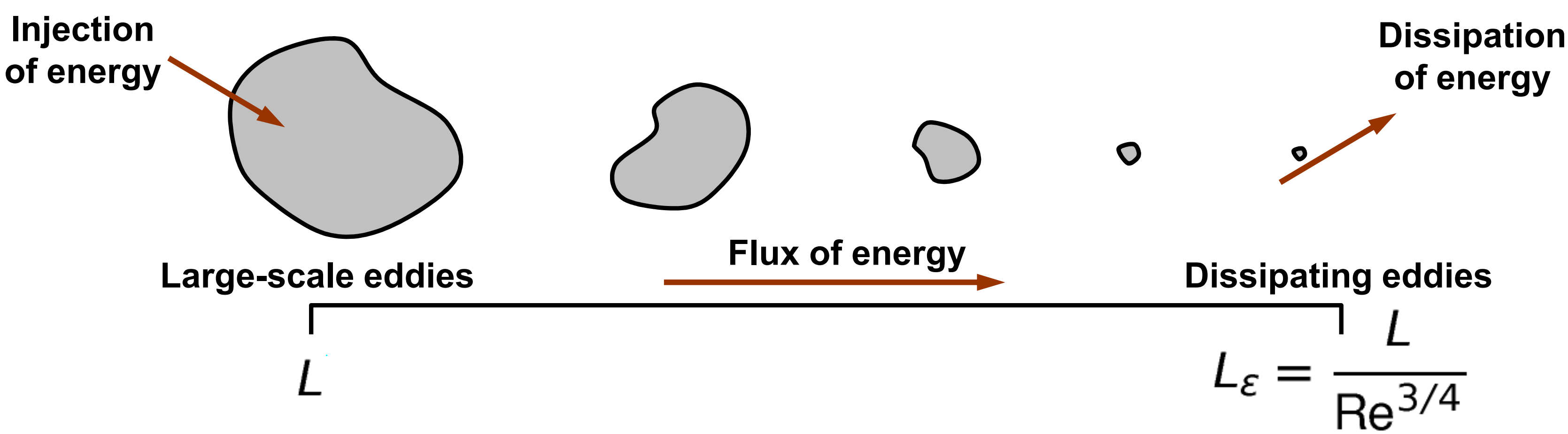
Introduction

- Most engineering flows are turbulent.
- Unlike everything else we have discussed on this course, turbulence is essentially a random process.
- Therefore we cannot ‘perfectly’ represent the effects of turbulence in the CFD simulation. Instead we need a **Turbulence Model.**
- The *turbulence model* is exactly that, a model. There is no “one size fits all” answer to turbulence modelling. You need to pick the most appropriate tool for your simulation.

Turbulent Flow Structures

- Although “random”, there are patterns to the motion as eddies dissipate



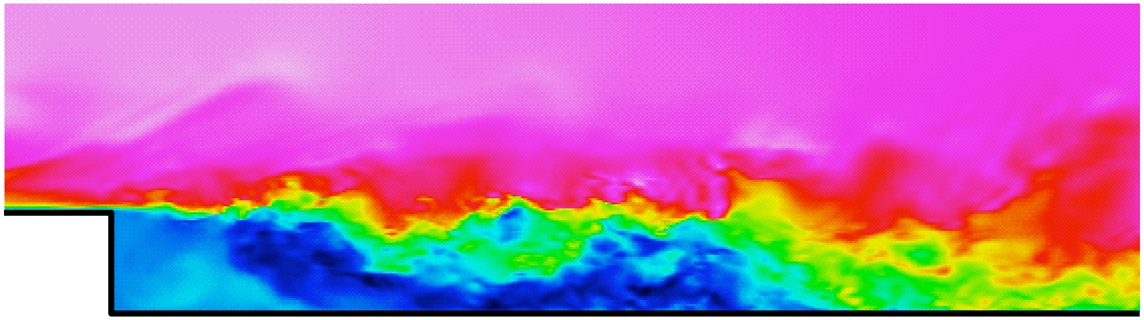


Energy Cascade (after Richardson, 1922)

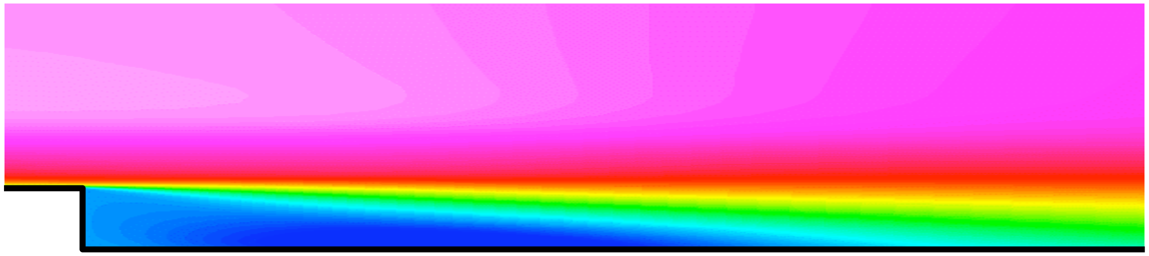
Backward Facing Step

- As engineers, in most cases we do not actually need to see an exact snapshot of the velocity at a particular instant.
- Instead for most problems, knowing the time-averaged velocity (and intensity of the turbulent fluctuations) is all we need to know. This gives us a useful way to approach modelling turbulence.

Instantaneous velocity contours



Time-averaged velocity contours



Is the Flow Turbulent?

External Flows

$\mathrm{Re}_x \geq 500{,}000$ along a surface

$\mathrm{Re}_d \geq 20{,}000$ around an obstacle

Internal Flows

$\mathrm{Re}_{d_h} \geq 2{,}300$

where $\mathrm{Re}_L = \dfrac{\rho U L}{\mu}$

$L = x, d, d_h, \text{etc.}$

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

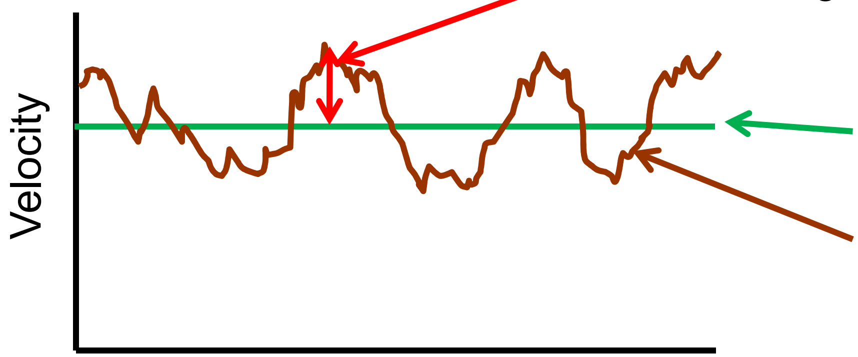
Natural Convection

$\dfrac{\mathrm{Ra}}{\mathrm{Pr}} \geq 10^9$ where $\mathrm{Ra} = \dfrac{\beta g L^3 \Delta T}{\nu\alpha} = \dfrac{\rho^2 c_p \beta g L^3 \Delta T}{\mu k}$ (Rayleigh number)

$\mathrm{Pr} = \dfrac{\nu}{\alpha} = \dfrac{\mu c_p}{k}$ (Prandtl number)

Mean and Instantaneous Velocities

- If we recorded the velocity at a particular point in the real (turbulent) fluid flow, the instantaneous velocity (U) would look like this:



- At any point in time: $U = \overline{U} + u'$
- The time average of the fluctuating velocity u' must be zero: $\overline{u'} = 0$
- BUT, the RMS of u' is not necessarily zero: $\overline{u'^2} \neq 0$
- Note you will hear reference to the turbulence energy, *k*. This is the sum of the 3 fluctuating velocity components: $k = 0.5 * \left(\overline{u'^2} + \overline{v'^2} + \overline{w'^2} \right)$

Approaches to Turbulence Modelling

- **Direct Numerical Simulation (DNS)**. **– Not In FLUENT**
  - It is technically possible to resolve every fluctuating motion in the flow.
  - However the grid must be very fine, and the timestep very small.
  - These demands increase with the Reynolds number.
  - The reality is that this is only a research tool for lower Reynolds-number flows restricted to supercomputer applications.

- **Reynolds-Averaged Navier Stokes (RANS) - In FLUENT**
  - This is the main tool used by engineers.
  - Equations are solved for time-averaged flow behaviour and the magnitude of turbulent fluctuations.

- **Large Eddy Simulation (LES) - In FLUENT**
  - In terms of computational demand LES lies between DNS and RANS.
  - Like DNS, a 3D simulation is performed over many timesteps.
  - However only the larger 'eddies' are resolved.
  - The grid can be coarser and timesteps larger than DNS because the smaller fluid motions are represented by a sub-grid-scale (SGS) model.

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}

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

RANS Models

- RANS Models fall into one of two categories. The difference in these is how the Reynolds Stress $\overline{u_i' u_j'}$ term on the previous slide is calculated:
  - **Eddy viscosity models** (EVM).
    - These assume the 'stress' is proportional to the 'strain' (strain being the gradients of velocity). The only new (unknown) quantity needed by EVMs is an effective turbulent viscosity μ_t.

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):
    - It is possible to derive a transport equation for the Reynolds Stress terms.
      - The model is more complex, since there are more equations to solve
      - Indeed there are further unknowns, which themselves require a model.
      - However they make the important step in allowing the Reynolds stresses to be anisotropic (the magnitude of u' can vary in the different directions). In some flows this is of crucial importance.

RANS Models - 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 which 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 in which the Reynolds stresses 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.

Turbulence Models Available in FLUENT

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

*A separate license is required

RANS : EVM : 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.

RANS : EVM : Standard k–ε Model

- Turbulence energy k has its own transport equation:

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

- This requires a dissipation rate, ε, which is entirely modeled phenomenologically (not derived) as follows:

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

- Dimensionally, the dissipation rate is related to *k* and a turbulence length scale:

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

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

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

RANS : EVM :Standard k–ε (SKE) Model [2]

- The **Standard K-Epsilon** 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.

RANS : EVM: RKE 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 compared with the standard k-epsilon model 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

RANS : EVM: Standard k–ω and SST k–ω

- **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.

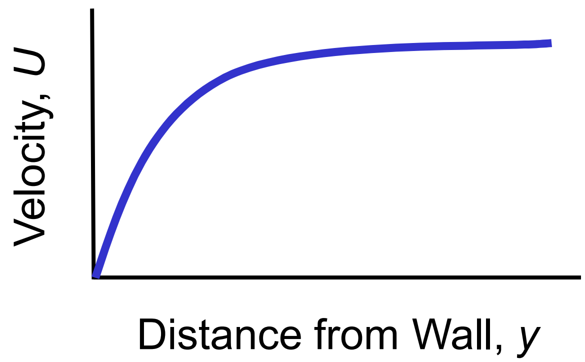
RANS : RSM: Reynolds Stress Model

- 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 where normal stresses play an important 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 do not always provide indisputably superior performance over EVMs.

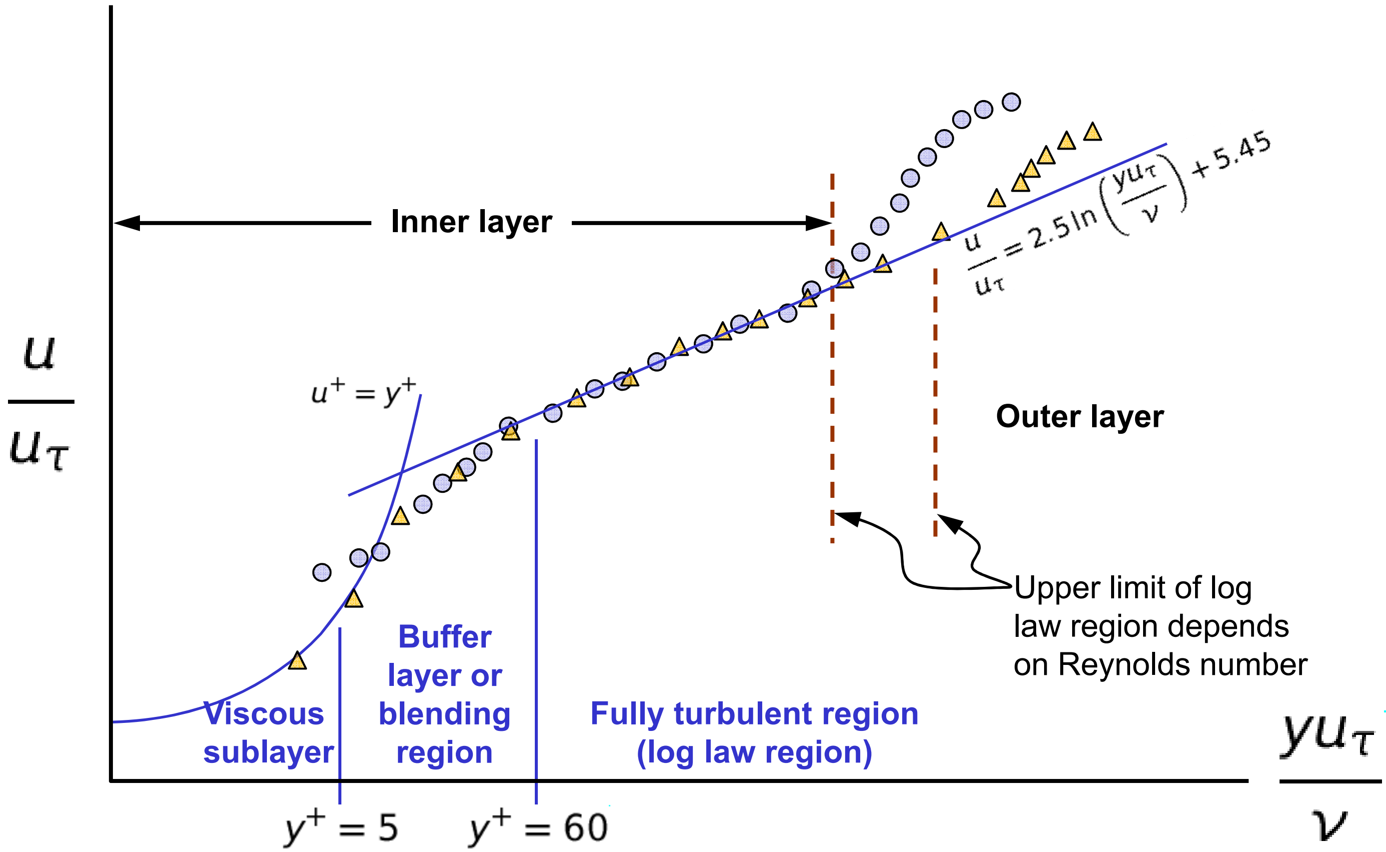
Turbulence near a Wall

- Near to a wall, the velocity changes rapidly.



- If we plot the same graph again, where:
  - Log scale axes are used
  - The velocity is made dimensionless, from $U/U\tau$ $\left(u_\tau = \sqrt{\frac{\tau_{wall}}{\rho}} \right)$
  - The wall distance vector is made dimensionless $y^+ = \frac{y u_\tau}{\nu}$
- *Then we arrive at the graph on the next page*. The shape of this is generally the same for all flows:

The Universal Law of The Wall



- Since this profile is common to many flows, we can use it for a 'Wall Model'
- The size of your grid cell nearest to the wall (value of **y^+**) is very important. The value you need depends on the modelling approach chosen.

Choice of Wall Modelling Strategy.

- 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:

Resolving the Viscous Sublayer

- First grid cell needs to be at about $y^+ = 1$
- This will add significantly to the mesh count
- Use a low-Reynolds number turbulence model (like k-omega)
- Generally speaking, if the forces on the wall are key to your simulation (aerodynamic drag, turbomachinery blade performance) this is the approach you will take

Using a Wall Function

- First grid cell needs to be $30 < y^+ < 300$

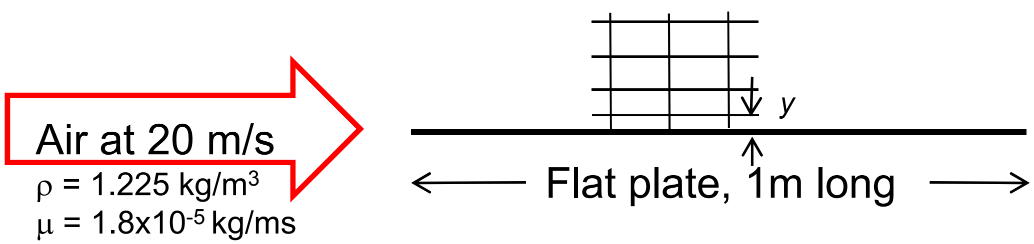
  (Too low, and model is invalid. Too high and the wall is not properly resolved.)
- Use a wall function, and a high Re turbulence model (SKE, RKE, RNG)
- Generally speaking, this is the approach if you are more interested in the mixing in the middle of the domain, rather than the forces on the wall.

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 Re_l = 1.4x10^6

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

Example in predicting near-wall cell size [2]

- A literature search suggests a formula for the skin friction on a plate[1] thus:

$$C_f = 0.058\,\mathrm{Re}_l^{-0.2} \qquad C_f = 0.0034$$

- Use this value to predict the wall shear stress τ_w

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

- From τ_w compute the velocity U_τ

$$U_\tau = \sqrt{\frac{\tau_w}{\rho}} \qquad U_\tau = 0.82 \text{ m/s}$$

- Rearranging the equation shown previously for y^+ gives a formula for the first cell height, *y,* in terms of $U\tau$

$$y = \frac{y^+ \mu}{U_\tau \rho} \qquad y = 9x10^{-4}\, m$$

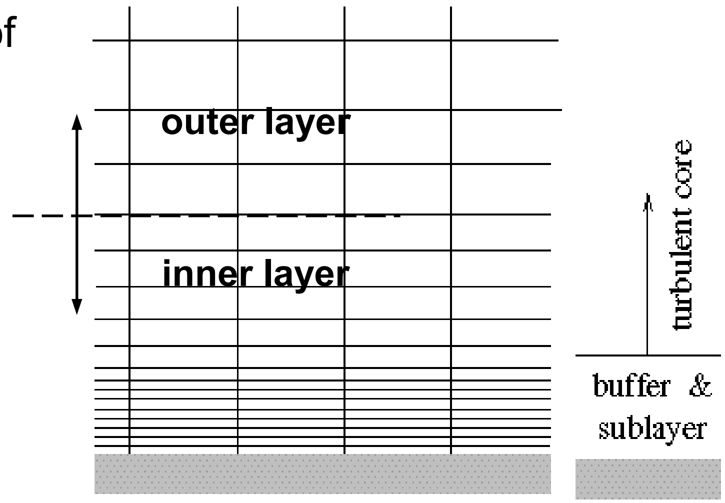
- We know we are aiming for y^+ of 50, hence:

our first cell height *y* should be approximately 1 mm.

[1] An equivalent formula for internal flows, based on the pipe-diameter Reynolds number is $C_f = 0.079\,\mathrm{Re}_d^{-0.25}$

Other options for Wall Modelling

- **Enhanced Wall Treatment Option (GUI)**
  - 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.
  - 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, buffer and log-law layers)



- **Scalable Wall Functions (TUI)**
  - In practice, many users often fail to maintain

    $30 < y^+ < 300$
  - Iteration by iteration, the first cell may change from being either inside / or outside of the viscous sublayer, which can lead to instabilities.
  - Scalable wall functions can be accessed by a TUI command
    `/define/models/viscous/near-wall-treatment/scalable-wall-functions`

Summary on Near-Wall Treatment

- Wall Functions are still the most affordable boundary treatment for many industrial CFD applications
- Standard wall functions work 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 $\overline{u_i' u_j'}$ must be specified, depending on which turbulence model has been selected

- Four methods for directly or indirectly specifying turbulence parameters:

  1) Explicitly input k, ε, ω, or Reynolds stress components (this is the only method that allows for profile definition)

  - Note by default, the FLUENT GUI enters k=1 m²/s² and ε=1m²/s³. **These values MUST be changed**, they are unlikely to be correct for your simulation.

  2) Turbulence intensity and length scale

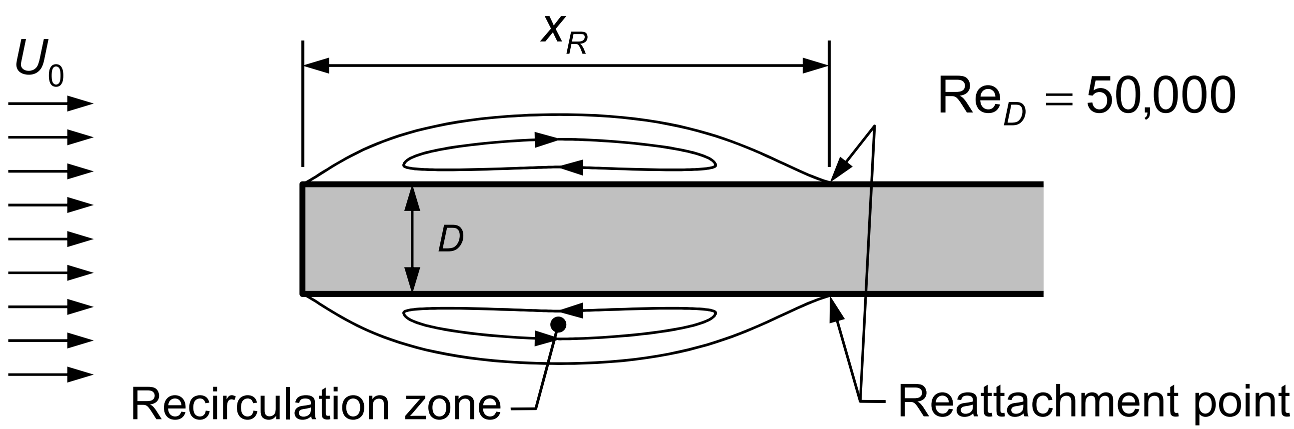
  - Length scale is related to size of large eddies that contain most of energy
    - For boundary layer flows: $l \approx 0.4\delta_{99}$
    - For flows downstream of grid: $l \approx$ opening size

  3) Turbulence intensity and hydraulic diameter (primarily for internal flows)

  4) Turbulence intensity and viscosity ratio (primarily for external flows)

Example #1 – Turbulent Flow Past a Blunt Flat Plate

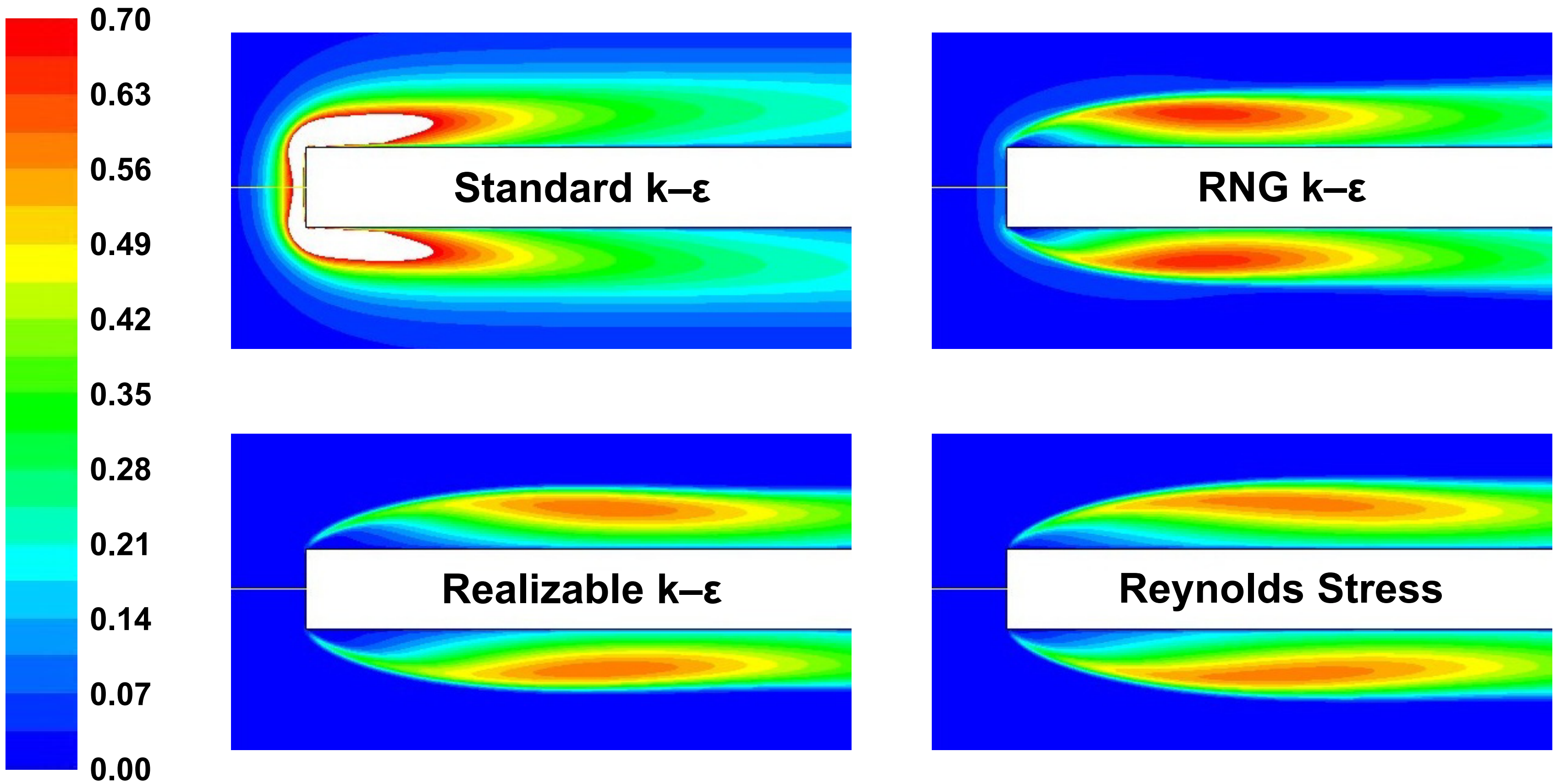
- Turbulent flow past a blunt flat plate was simulated using four different turbulence models.
  - 8,700 cell quad mesh, graded near leading edge and reattachment location.
  - Non-equilibrium boundary layer treatment



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

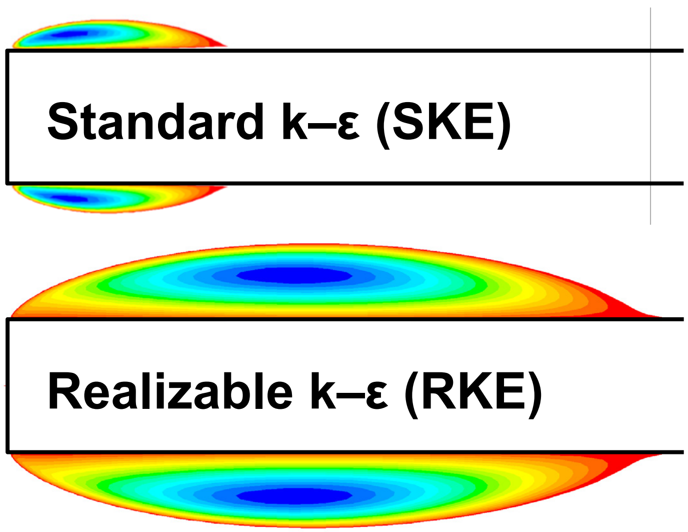
Example #1 – Turbulent Flow Past a Blunt Flat Plate

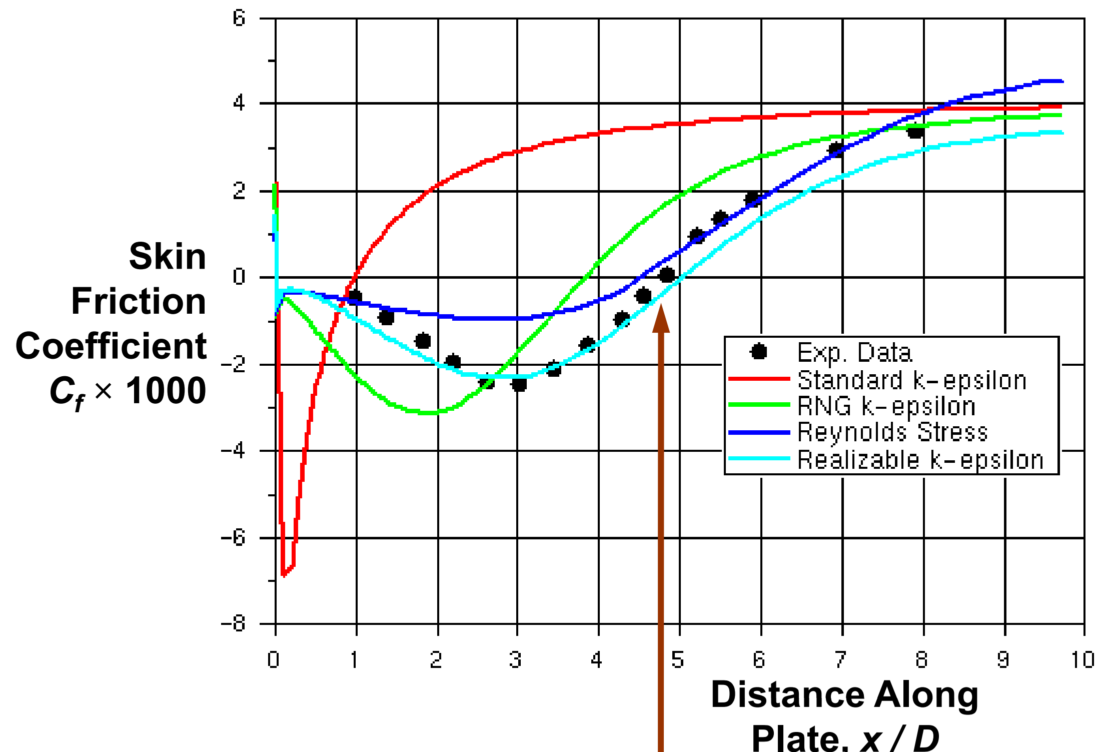
Contours of Turbulent Kinetic Energy (m^2/s^2)



Example #1 – Turbulent Flow Past a Blunt Flat Plate

Predicted separation bubble:



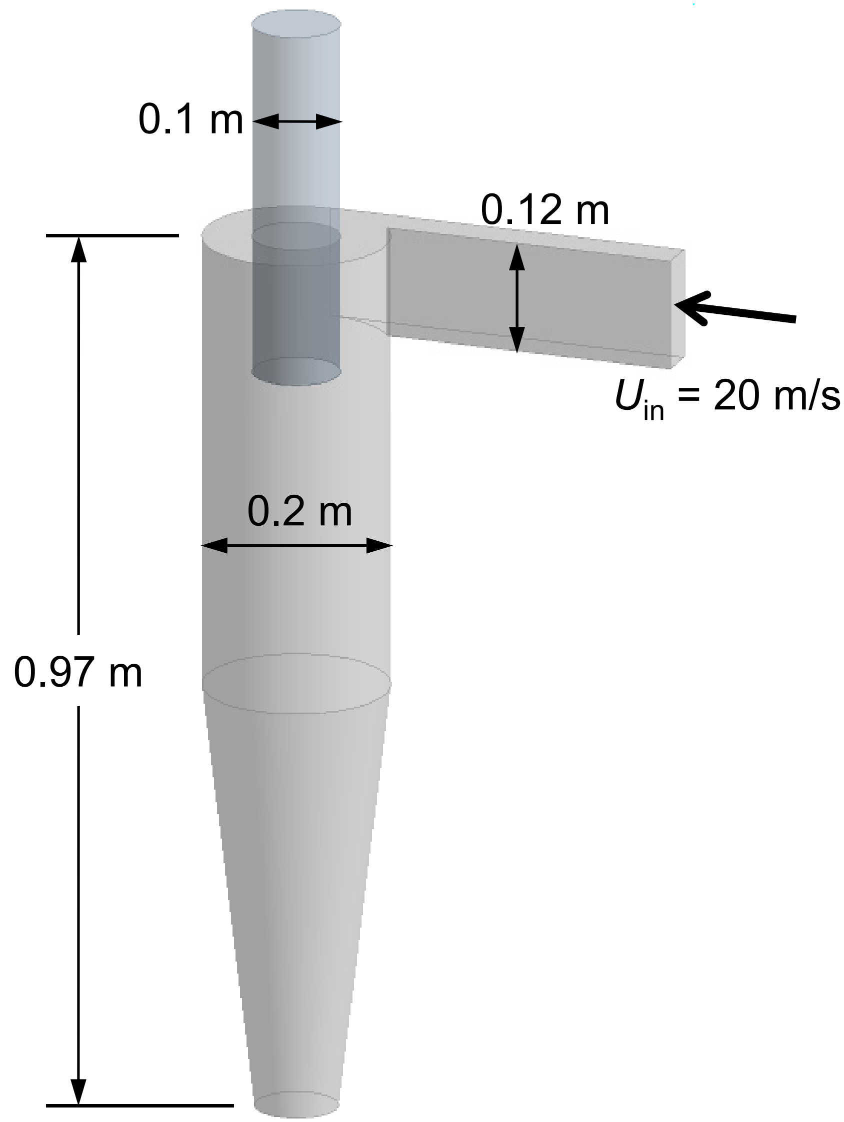


SKE severely underpredicts the size of the separation bubble, while RKE predicts the size exactly.

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

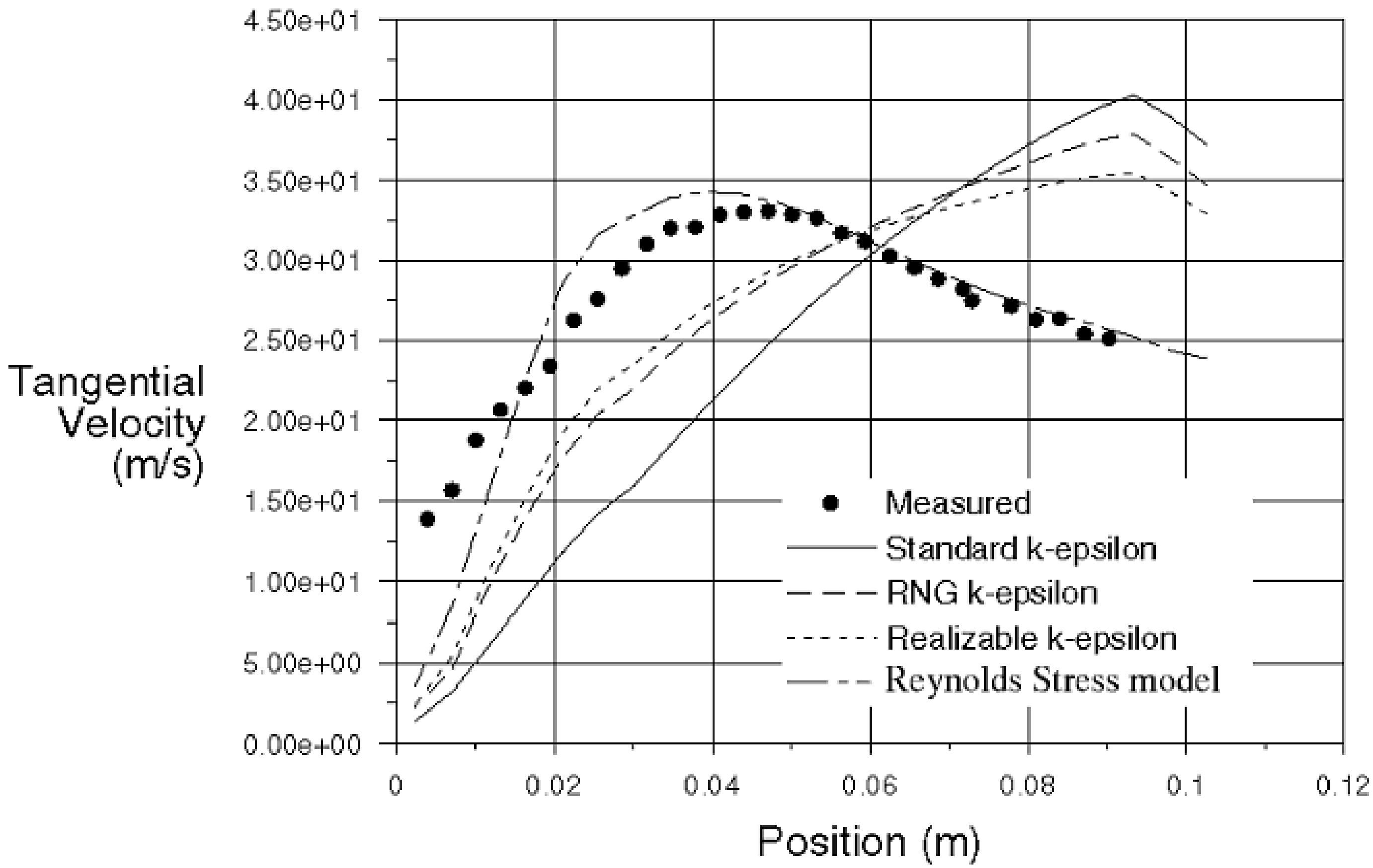
Example #2 – Turbulent Flow in a Cyclone

- 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



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. FLUENT's implementation allows the use of coarser meshes. 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.

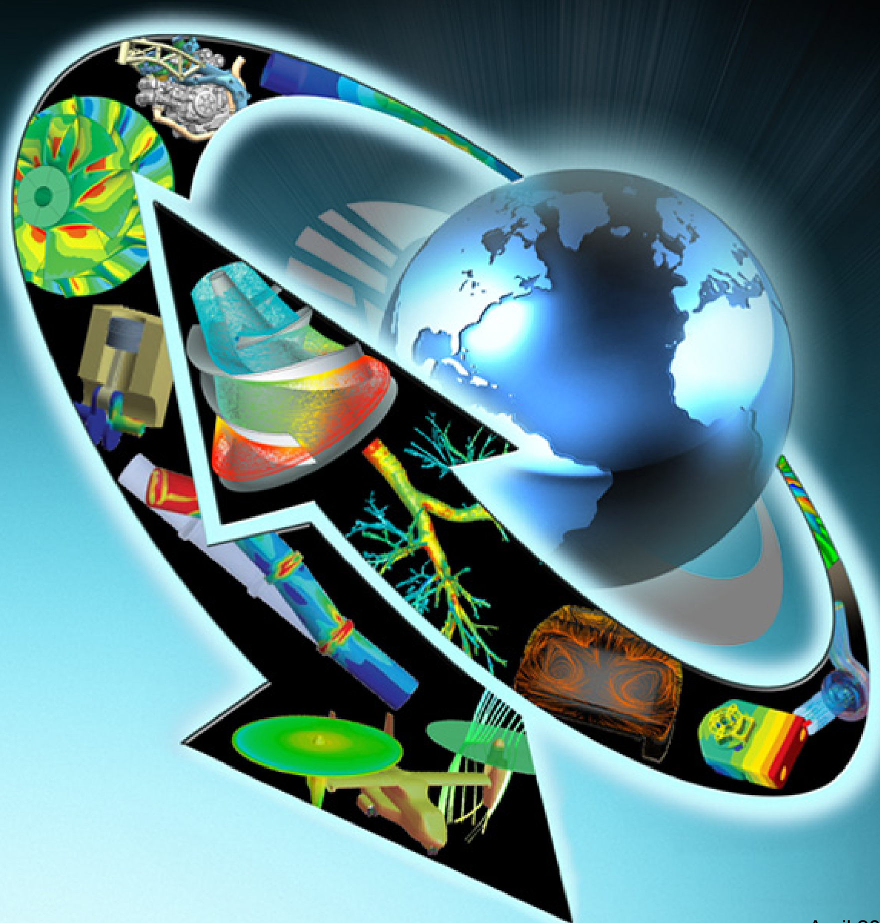
RANS Models Behavior Summary

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).

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
  1. Calculate characteristic Reynolds number and determine whether flow is turbulent.
  2. If the flow is in the transition (from laminar to turbulent) range, consider the use of one of the turbulence transition models (not covered in this training).
  3. Estimate wall-adjacent cell centroid y^+ before generating the mesh.
  4. Prepare your mesh to use wall functions except for low-Re flows and/or flows with complex near-wall physics (non-equilibrium boundary layers).
  5. Begin with RKE (realizable k-ε) and change to S-A, RNG, SKW, SST or v2f if needed. Check the tables on previous slides as a guide for your choice.
  6. Use RSM for highly swirling, 3-D, rotating flows.
  7. ***Remember that there is no single, superior turbulence model for all flows!***

Appendix – Additional Notes on Turbulence Modelling

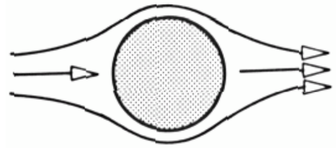


Characteristics of Turbulence

- 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
- Wide range of length scales (vortices or eddies) exist in all turbulent flows (from very small to very large).
- Very sensitive to (or dependent on) initial conditions.

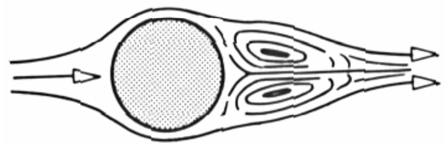
Reynolds Number Effects

$Re < 5$



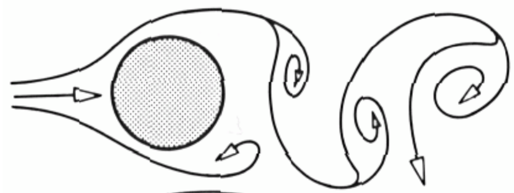
Creeping flow (no separation)

$5\text{-}15 < Re < 40$



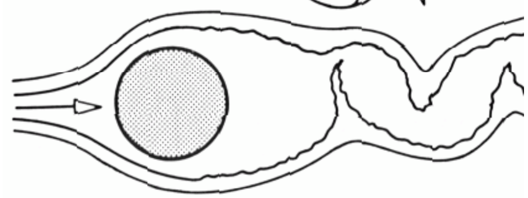
A pair of stable vortices in the wake

$40 < Re < 150$



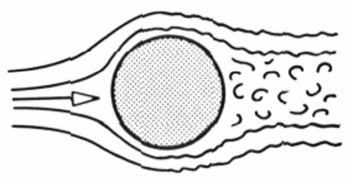
Laminar vortex street

$150 < Re < 3\times10^5$



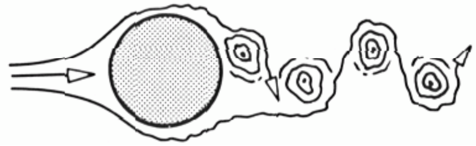
Laminar boundary layer up to the separation point, turbulent wake

$3\times10^5 < Re < 3.5\times10^6$



Boundary layer transition to turbulent

$Re > 3.5\times10^6$



Turbulent vortex street, but the separation is narrower than the laminar case

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.

Eddy Viscosity Models

- Dimensional analysis indicates that eddy viscosity can be determined if we have the necessary scales (velocity, length, etc.)

$$\frac{\mu_t}{\rho} = \nu_t \left[\frac{L^2}{t}\right] = \left[\frac{L^2}{t^2}\right] \times [t] = \left[\frac{L}{t}\right] \times [L]$$

  - For example, given the turbulence velocity scale and length scale, or velocity scale and time scale, eddy viscosity is determined and the RANS equations are closed
  - These scales can only be prescribed for very simple flows (like fully-developed turbulent pipe flow or Couette flow).

- For general applications, we need to derive transport equations (PDEs) of the chosen scales in order to compute eddy viscosity

- Turbulent kinetic energy *k* (per unit mass) provides useful physical insight into the EVMs

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 *k* Equation

- The dissipation rate ε refers to the viscous dissipation of kinetic energy into internal energy:

$$\varepsilon = \nu \overline{\frac{\partial u_j'}{\partial x_k} \frac{\partial u_j'}{\partial x_k}}$$

- Physically turbulence kinetic energy *k* is produced due to the mean flow gradients, and is dissipated by viscous effects. Imbalance between the production and the dissipation will cause *k* either to grow or to decay

- The last term in the *k* equation is a diffusion term. It is modeled by a gradient diffusion assumption or Reynolds analogy (hence the use of a turbulent Prandtl number in the diffusion term)

Direct Numerical Simulation (DNS)

- In DNS, the 3D unsteady Navier-Stokes equations are solved numerically by resolving all scales (both in space and in time)
- For simple geometries and at modest Reynolds numbers, DNS has been done successfully. For example, for a simple turbulent channel flow between two plates:
  $Re_\tau = 800$, $N = (Re_\tau)^{9/4} = 10{,}000{,}000$ (cells), $\Delta t = 10^{-5}$ sec.
- DNS is equivalent to a "numerical wind tunnel" for conducting more fundamental turbulence research
- For practical engineering purposes, DNS is not only too costly, but also the details of the simulation are usually not required.
- Two general engineering approaches to modeling turbulence: Large-Eddy Simulation (LES) and Reynolds Averaging Navier-Stokes (RANS) models

Turbulent Heat Transfer

- The Reynolds averaging of the energy equation produces a closure term and we call it the turbulent (or eddy) heat flux:
  - Analogous to the closure of Reynolds stress, a turbulent thermal diffusivity is assumed:

Turbulent thermal diffusivity

$$-\rho\overline{u_i' t'} = \rho\alpha_t \frac{\partial T}{\partial x_i}$$

  - Turbulent diffusivity is obtained from eddy viscosity via a turbulent Prandtl number (modifiable by the users) based on the Reynolds analogy:

$$\alpha_t = \frac{\nu_t}{\mathrm{Pr}_t} \qquad \mathrm{Pr}_t \approx 0.85 - 0.9$$

- Similar treatment is applicable to other turbulent scalar transport equations

The Spalart-Allmaras Turbulence Model

- A low-cost RANS model solving an equation for the modified eddy viscosity,

$$\frac{D\tilde{\nu}}{Dt} = G_\nu \left\{ \frac{\partial}{\partial x_j}\left[(\mu + \rho\tilde{\nu})\frac{\partial \tilde{\nu}}{\partial x_j}\right] + C_{b2}\rho\left(\frac{\partial \tilde{\nu}}{\partial x_j}\right)^2 \right\} - Y_\nu + S_{\tilde{\nu}}$$

- Eddy viscosity is obtained from

$$\mu_t = \rho\tilde{\nu}f_{\nu 1} \qquad f_{\nu 1} = \frac{(\tilde{\nu}/\nu)^3}{(\tilde{\nu}/\nu)^3 + C_{\nu 1}^3}$$

- The variation of $\tilde{\nu}$ very near the wall is easier to resolve than *k* and ε.

- Mainly intended for aerodynamic/turbomachinery applications with mild separation, such as supersonic/transonic flows over airfoils, boundary-layer flows, etc.

RANS Models – Standard k–ε (SKE) Model

- Transport equations for *k* and ε

$$\frac{D(\rho k)}{Dt} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k - \rho\varepsilon$$

$$\frac{D(\rho\varepsilon)}{Dt} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + C_{\varepsilon 1}\frac{\varepsilon}{k}G_k - \rho C_{\varepsilon 2}\frac{\varepsilon^2}{k}$$

$$C_\mu = 0.09 \quad C_{\varepsilon 1} = 1.44 \quad C_{\varepsilon 2} = 1.92 \quad \sigma_k = 1.0 \quad \sigma_\varepsilon = 1.3$$

- SKE is the most widely-used engineering turbulence model for industrial applications.
- Robust and reasonably accurate; it has many submodels for compressibility, buoyancy, and combustion, etc.
- Performs poorly for flows with strong separation, large streamline curvature, and high pressure gradient

RANS Models – k–ω Models

$$\rho\frac{Dk}{Dt} = \tau_{ij}\frac{\partial \overline{u}_i}{\partial x_j} - \rho\beta^* f_{\beta^*} k\omega + \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial k}{\partial x_j}\right]$$

$$\mu_t = \alpha^* \rho\frac{k}{\omega}$$

$$\rho\frac{D\omega}{Dt} = \alpha\frac{\omega}{k}\tau_{ij}\frac{\partial \overline{u}_i}{\partial x_j} - \rho\beta f_\beta \omega^2 + \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\omega}\right)\frac{\partial \omega}{\partial x_j}\right]$$

Specific dissipation rate, ω

$$\omega \approx \frac{\varepsilon}{k} \propto \frac{1}{\tau}$$

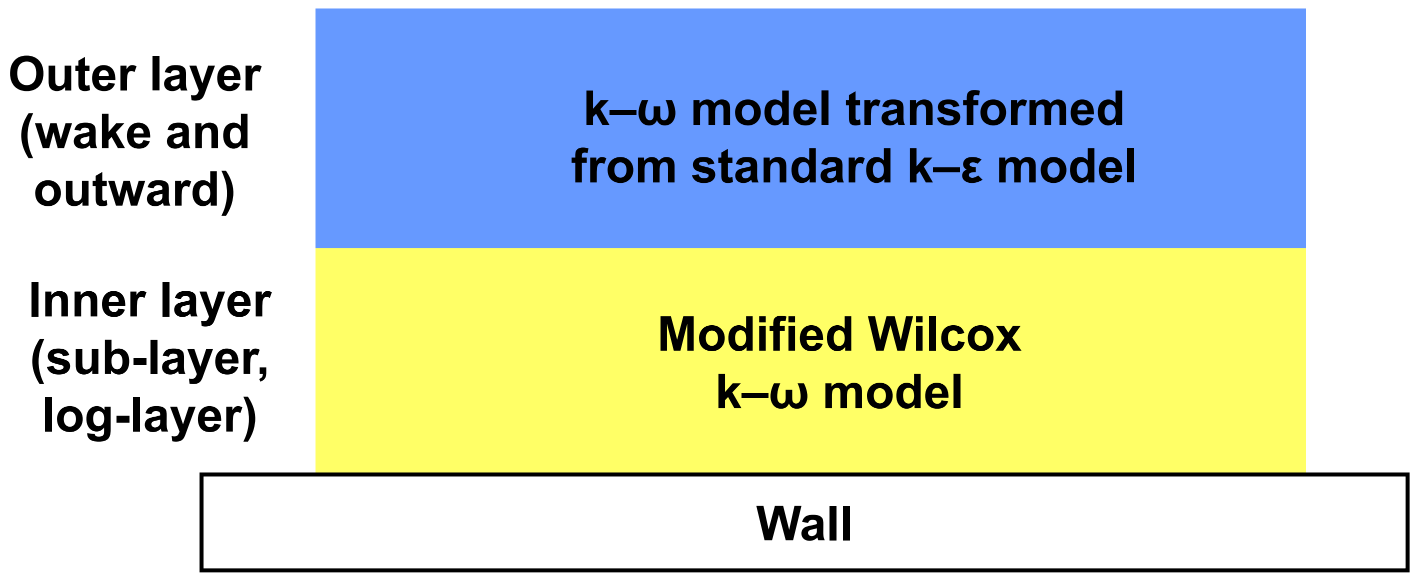
- Belongs to the general 2-equation EVM family. ANSYS FLUENT supports the standard k–ω model by Wilcox (1998) and Menter's SST k–ω model (1994).
- k–ω models have gained popularity for the following reasons:
  - Can be integrated to the wall without using any damping functions
  - Accurate and robust for a wide range of boundary layer flows with pressure gradient
- Most widely adopted in the aerospace and turbo-machinery communities.
- Several sub-models/options of k–ω: compressibility effects, transitional flows and shear-flow corrections.

Menter's SST k–ω Model Background

- Many people, including Menter (1994), have noted that:
  - The k–ω model has many good attributes and performs much better than k–ε models for boundary layer flows
  - Wilcox' original k–ω model is overly sensitive to the free stream value of ω, while the k–ε model is not prone to such problem
  - Most two-equation models, including k–ε models, over-predict turbulent stresses in the wake (velocity-defect) regions, which leads to poor performance in predicting boundary layers under adverse pressure gradient and separated flows
  - The basic idea of SST k–ω is to combine SKW in the near-wall region with SKE in the outer region

Menter's SST k–ω Model Main Components

- The SST k–ω model consists of
  - Zonal (blended) k–ω / k–ε equations (to address item 1 and 2 in the previous slide)
  - Clipping of turbulent viscosity so that turbulent stress stay within what is dictated by the structural similarity constant (Bradshaw, 1967) - addresses the overprediction problem



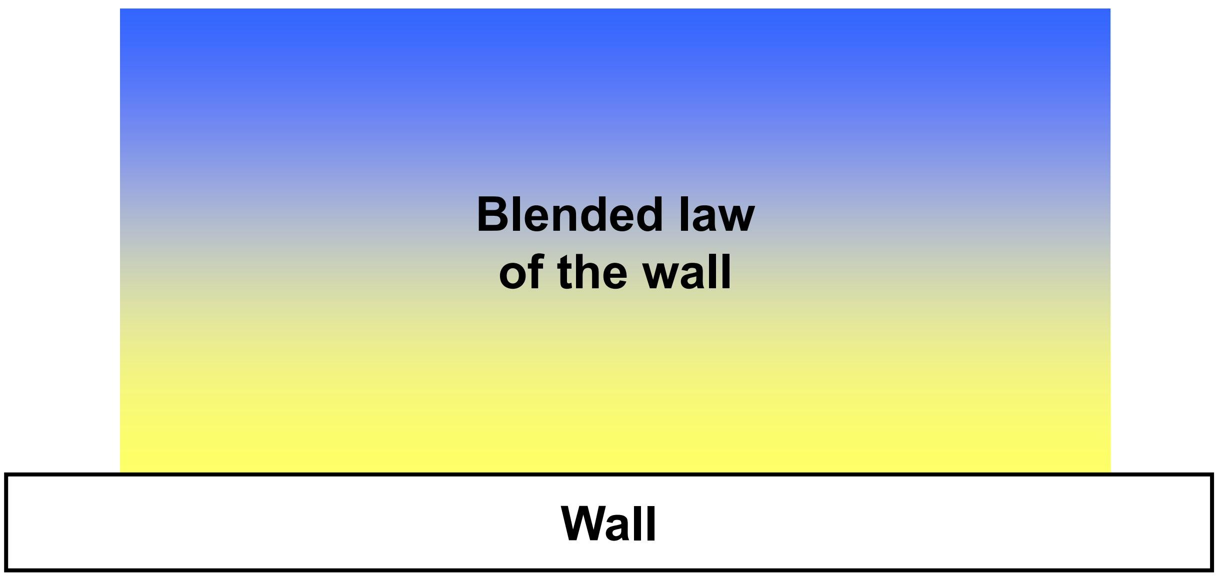
Menter's SST k–ω Model Blended equations

- The resulting blended equations are:

$$\rho\frac{Dk}{Dt} = \tau_{ij}\frac{\partial \overline{u}_i}{\partial x_j} - \rho\beta^* k\omega + \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]$$

$$\rho\frac{D\omega}{Dt} = \frac{\gamma}{\nu_t}\tau_{ij}\frac{\partial \overline{u}_i}{\partial x_j} - \beta\rho\omega^2 + \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\omega}\right)\frac{\partial \omega}{\partial x_j}\right] + 2\rho(1-F_1)\sigma_{\omega 2}\frac{1}{\omega}\frac{\partial k}{\partial x_j}\frac{\partial \omega}{\partial x_j}$$

$$\phi = F_1\phi_1 + (1-F_1)\phi_2 \qquad \phi = \beta, \sigma_k, \sigma_w, \gamma$$



Reynolds Stress Model (RSM)

$$\frac{\partial \rho \overline{u_i' u_j'}}{\partial t} + \frac{\partial}{\partial x_k}\left(\overline{u}_k \overline{u_i' u_j'}\right) = P_{ij} + \overbrace{F_{ij} + D_{ij}^T + \Phi_{ij} - \varepsilon_{ij}}^{\text{Modeling required for these terms}}$$

Stress production
Turbulent diffusion
Pressure strain
Dissipation

- Attempts to address the deficiencies of the EVM.
- RSM is the most 'physically sound' model: anisotropy, history effects and transport of Reynolds stresses are directly accounted for.
- RSM requires substantially more modeling for the governing equations (the pressure-strain is most critical and difficult one among them).
- But RSM is more costly and difficult to converge than the 2-equation models.
- Most suitable for complex 3-D flows with strong streamline curvature, swirl and rotation.

The Wall-Adjacent Cell Size Estimation

- For a flat plate, a good power-law correlation for turbulent skin-friction coefficient is

$$C_f = \frac{\tau_w}{\frac{1}{2}\rho U_\infty^2} \approx 0.027\,\mathrm{Re}_x^{1/7}$$

- The distance from the wall to the centroid of the first fluid cell (Δy) can be estimated by choosing the desired y^+ with the estimated bulk Reynolds number for the wall shear layer:

$$\boxed{\frac{\Delta y}{y^+} \approx 8.6\,L\,\mathrm{Re}_L^{-13/14}} \qquad Re_L = \frac{\rho U_\infty L}{\mu} \quad \text{(Bulk Reynolds number)}$$

- For duct flow, one can similarly estimate Δy as follows:

$$C_f = \frac{0.078}{\mathrm{Re}_{d_h}^{1/4}} \qquad \boxed{\frac{\Delta y}{y^+} \approx 5.06\,d_h\,\mathrm{Re}_{d_h}^{-7/8}} \qquad d_h = \frac{4A}{P_{\mathrm{wet}}}$$

(Hydraulic diameter)

Standard Wall Functions

- Standard Wall Functions
  - Momentum boundary condition based on Launder-Spaulding law-of-the-wall:

$$u^* = \begin{cases} y^* & (y^* < y_\nu^*) \\ \dfrac{\ln(Ey^*)}{\kappa} & (y^* > y_\nu^*) \end{cases}$$

$$u^* = \frac{u_P C_\mu^{1/4} k_P^{1/2}}{u_\tau^2}$$

$$y^* = \frac{\rho C_\mu^{1/4} k_P^{1/2} y_P}{\mu}$$

  - Similar wall functions apply for energy and species.
  - Additional formulas account for k, ε, and $\rho \overline{u_i' u_j'}$
  - Less reliable when flow departs from conditions assumed in their derivation.
    - Severe pressure gradient or highly non-equilibrium near-wall flows, high transpiration or body forces, low Re or highly 3D flows

Standard Wall Functions

- Energy

$$T^* = \frac{(T_w - T_P)\,\rho c_p k_P^{1/2}}{\dot{q}} = \begin{cases} Pr y^* + \frac{1}{2} Pr \frac{C_\mu^{1/4} k_P^{1/2} U_P^2}{\dot{q}} U_P^2 & (y^* < y_T^*) \\ Pr_t \left[\frac{1}{\kappa} \ln(E y^*) + P \right] & \\ \quad + \frac{1}{2} \frac{\rho C_\mu^{1/4} k_P^{1/2}}{\dot{q}} \left\{ Pr_t U_P^2 + (Pr - Pr_t) U_c^2 \right\} & (y^* > y_T^*) \end{cases}$$

$$P = 9.24 \left[\left(\frac{Pr}{Pr_t} \right)^{3/4} - 1 \right] \left[1 + 0.28 \exp\left(-0.007 \frac{Pr}{Pr_t} \right) \right]$$

- Species

$$Y^* = \begin{cases} Sc\, y^* & (y^* < y_c^*) \\ Sc_t \left[\frac{1}{\kappa} \ln(E y^*) + P_c \right] & (y^* > y_c^*) \end{cases}$$

Enhanced Wall Treatment

- Enhanced wall functions
  - Momentum boundary condition based on a blended law of the wall.

$$u^+ = e^{\Gamma} u^+_{\text{lam}} + e^{1/\Gamma} u^+_{\text{turb}}$$

  - Similar blended wall functions apply for energy, species, and ω.
  - Kader's form for blending allows for incorporation of additional physics.
    - Pressure gradient effects
    - Thermal (including compressibility) effects
- Two-layer zonal model
  - A blended two-layer model is used to determine near-wall ε field.
    - Domain is divided into viscosity-affected (near-wall) region and turbulent core region.
      - Based on the wall-distance-based turbulent Reynolds number: $\text{Re}_y = \dfrac{\rho y \sqrt{k}}{\mu}$
      - Zoning is dynamic and solution adaptive
    - High Re turbulence model used in outer layer
    - Simple turbulence model used in inner layer

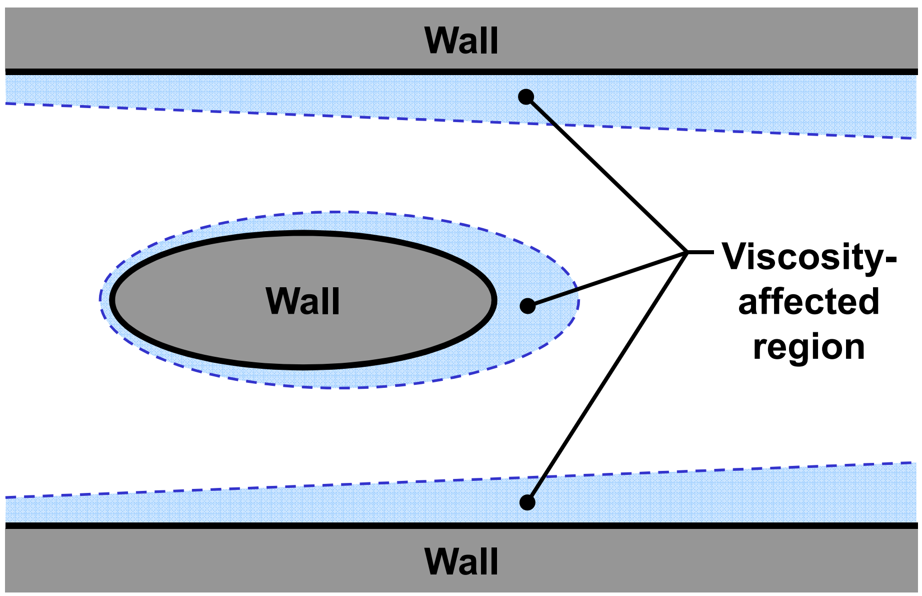
$$\lambda_\varepsilon (u_t)_{\text{outer}} + (1 - \lambda_\varepsilon)(u_t)_{\text{inner}}$$

  - Solutions for ε and μT in each region are blended:
- The Enhanced Wall Treatment option is available for the k–ε and RSM models (EWT is the sole treatment for Spalart Allmaras and k–ω models)

Two-Layer Zonal Model

- The two regions are demarcated on a cell-by-cell basis:
  - Turbulent core region (where $Re_y > 200$)
  - Viscosity affected region (where $Re_y < 200$)
  - y is the distance to the nearest wall.
  - Zoning is dynamic and solution adaptive.

$$Re_y = \frac{\rho y \sqrt{k}}{\mu}$$

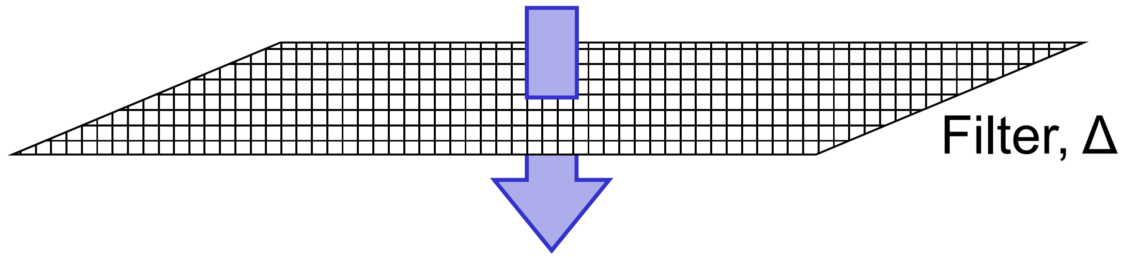


Large Eddy Simulation (LES)

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

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

- 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)
- Implementations in FLUENT:
  - 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
- LES is compatible with all combustion models in FLUENT
- Basic statistical tools are available: Time averaged and RMS values of solution variables, built-in fast Fourier transform (FFT).
- Before running LES, consult guidelines in the “Best Practices For LES” (containing advice for meshing, subgrid model, numerics, BCs, and more)

Stochastic Inlet Velocity BC

- It is often important to specify realistic turbulent inflow velocity BC for accurate prediction of the downstream flow:

$$\underbrace{u_i(\mathbf{x},t)}_{\text{Instantaneous component}} = \underbrace{\overline{u}_i(\mathbf{x},t)}_{\text{Time-averaged}} + \underbrace{u_i'(\mathbf{x},t)}_{\text{Coherent + random}}$$

- Different types of inlet boundary conditions for LES
  - No perturbations – Turbulent fluctuations are not present at the inlet.
  - Vortex method – Turbulence is mimicked by using the velocity field induced by many quasi-random point-vortices on the inlet surface. The vortex method uses turbulence quantities as input values (similar to those used for RANS-based models).
  - Spectral synthesizer
    - Able to synthesize anisotropic, inhomogeneous turbulence from RANS results (k–ε, k–ω, and RSM fields).
- Can be used for RANS/LES zonal hybrid approach.

Initial Velocity Field for LES/DES

- Initial condition for velocity field does not affect statistically stationary solutions

- However, starting LES with a realistic turbulent velocity field can substantially shorten the simulation time to get to statistically stationary state

- The spectral synthesizer can be used to superimpose turbulent velocity on top of the mean velocity field
  - Uses steady-state RANS (k–ε, k–ω, RSM, etc.) solutions as inputs to the spectral synthesizer
  - Accessible via a TUI command **`/solve/initialize/init-instantaneous-vel`**

v^2f Turbulence Model

- A model developed by Paul Durbin's group at Stanford University.
  - Durbin suggests that the wall-normal fluctuations $\overline{v'v'}$ are responsible for the near-wall damping of the eddy viscosity
  - Requires two additional transport equations: one for $\overline{v'v'}$ and one fo a relaxation function f to be solved together with k and ε.
  - Eddy viscosity model is $\nu_t \sim \overline{v'v'}T$ instead of $\nu_t \sim kT$

- v^2f shows promising results for many 3D, low Re, boundary layer flows. For example, improved predictions for heat transfer in jet impingement and separated flows, where k–ε models behave poorly

- But v^2f is still an eddy viscosity model, thus the same limitations still apply

- v^2f is an embedded add-on functionality in FLUENT which requires a separate license from Cascade Technologies (www.turbulentflow.com)