


Lecture 7: Turbulence Modeling

15.0 Release

A visualization of fluid flow, showing blue, wavy, semi-transparent surfaces that represent the movement of a fluid over a surface, likely a wing or airfoil.


Fluid Dynamics

A 3D model of a purple gear with a glowing white center, surrounded by other faint gear shapes, representing structural analysis.

Structural Mechanics

A series of concentric green circles with a glowing center, representing electromagnetic field lines or wave propagation.

Electromagnetics

A 3D structure composed of several blue and black rectangular blocks, representing a complex system or multiphysics simulation.

Systems and Multiphysics

Introduction to ANSYS Fluent

Lecture Theme:

The majority of engineering flows are turbulent. Simulating turbulent flows in Fluent requires activating a turbulence model, selecting a near-wall modeling approach and providing inlet boundary conditions for the turbulence model.

Learning Aims:

You will learn:

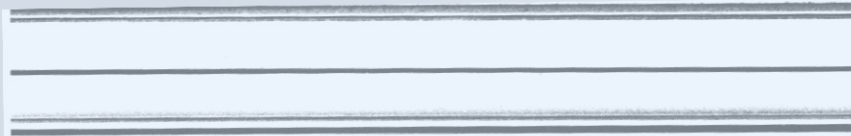
- How to use the Reynolds number to determine whether the flow is turbulent
- How to select the turbulence model
- How to choose which approach to use for modeling flow near walls
- How to specify turbulence boundary conditions at inlets

Learning Objectives:

You will be able to determine whether a flow is turbulent and be able to set up and solve turbulent flow problems

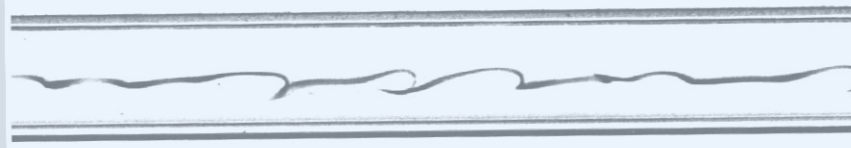
Observation by Osborne Reynolds

- Flows can be classified as either :



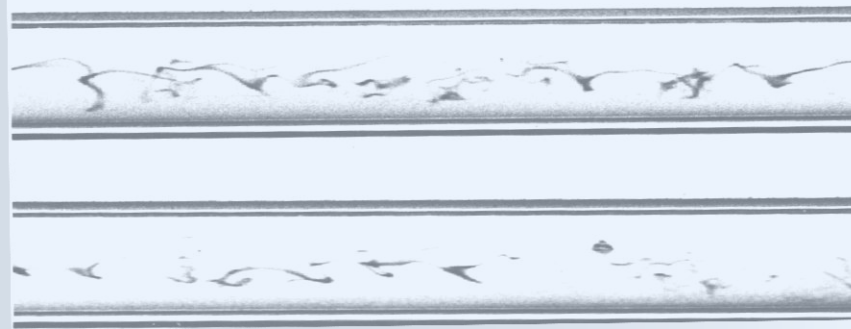
Laminar

(Low Reynolds Number)



Transitional

(Increasing Reynolds Number)



Turbulent

(Higher Reynolds Number)

Reynolds Number

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

$$\text{Re}_L = \frac{\rho \cdot U \cdot L}{\mu}$$

- The Reynolds number is based on the length scale of the flow:

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

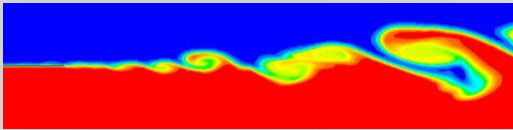
- Transition to turbulence varies depending on the type of flow:
 - External flow
 - along a surface : $\text{Re}_x > 500\,000$
 - around on obstacle : $\text{Re}_L > 20\,000$
 - Internal flow : $\text{Re}_d > 2\,300$

Overview of Computational Approaches

- Three basic approaches can be used to calculate a turbulent flow

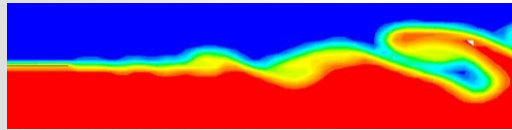
DNS

(Direct Numerical Simulation)



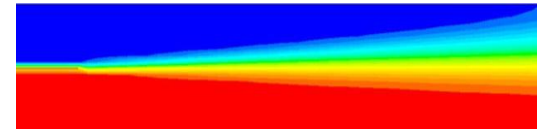
LES

(Large Eddy Simulation)



RANS

(Reynolds Averaged Navier-Stokes Simulation)



- Numerically solving the full unsteady Navier-Stokes equations
 - Resolves the whole spectrum of scales
 - No modeling is required
 - **But the cost is too prohibitive!**
- Not practical for industrial flows!**

- Solves the filtered N-S equations
- Some turbulence is directly resolved
- **Less expensive than DNS, but the efforts and computational resources needed are still too large for most practical applications**
- Available in Fluent but not discussed in Introductory Training

- Solve time-averaged N-S equations
- All turbulent motion is modeled
- For most problems the time-averaged flow (and level of turbulence) are all that is needed
- Many different models are available
- **This is the most widely used approach for industrial flows**

Turbulence Models Available in Fluent

RANS based
models

One-Equation Model

Spalart-Allmaras

Two-Equation Models

Standard $k-\epsilon$

RNG $k-\epsilon$

Realizable $k-\epsilon^*$

Standard $k-\omega$

SST $k-\omega^*$

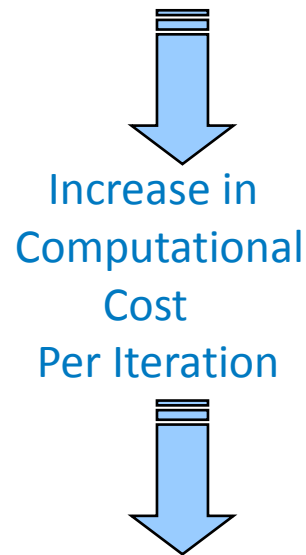
Reynolds Stress Model

$k-l-\omega$ Transition Model

SST Transition Model

Detached Eddy Simulation

Large Eddy Simulation



* SST $k-\omega$ and Realizable $k-\epsilon$ are recommended choices for standard cases

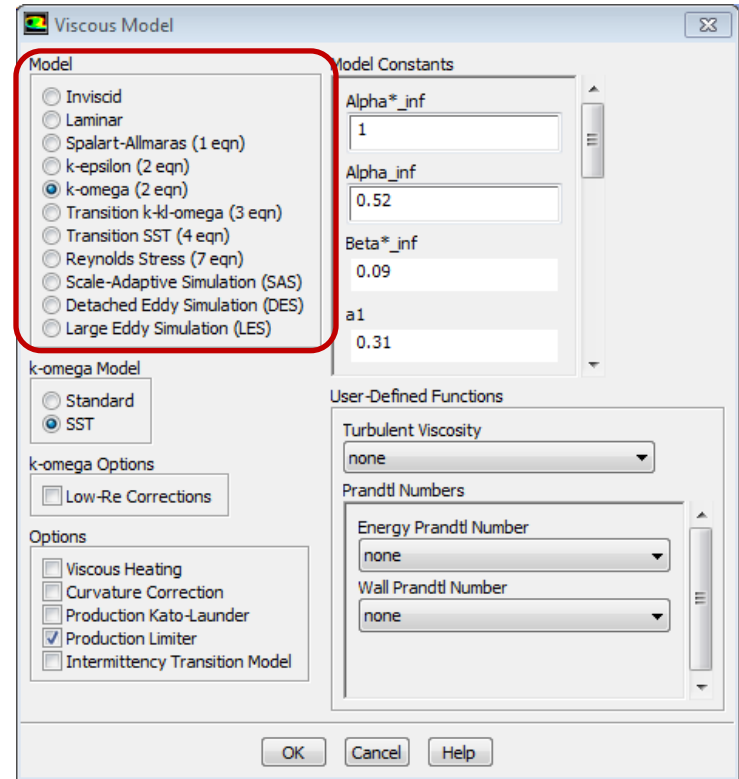
RANS Turbulence Model Usage

Model	Behavior and Usage
Spalart-Allmaras	Economical for large meshes. Good for mildly complex (quasi-2D) external/internal flows and boundary layer flows under pressure gradient (e.g. airfoils, wings, airplane fuselages, missiles, ship hulls). Performs poorly for 3D flows, free shear flows, flows with strong separation.
Standard $k-\epsilon$	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.
Realizable $k-\epsilon^*$	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).
RNG $k-\epsilon$	Offers largely the same benefits and has similar applications as Realizable. Possibly harder to converge than Realizable.
Standard $k-\omega$	Superior performance for wall-bounded boundary layer, free shear, and low Reynolds number flows compared to models from the $k-\epsilon$ family. Suitable for complex boundary layer flows under adverse pressure gradient and separation (external aerodynamics and turbomachinery). Separation can be predicted to be excessive and early.
SST $k-\omega^*$	Offers similar benefits as standard $k-\omega$. Not overly sensitive to inlet boundary conditions like the standard $k-\omega$. Provides more accurate prediction of flow separation than other RANS models.
RSM	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).

* Realizable $k-\epsilon$ or SST $k-\omega$ are the recommended choice for standard cases

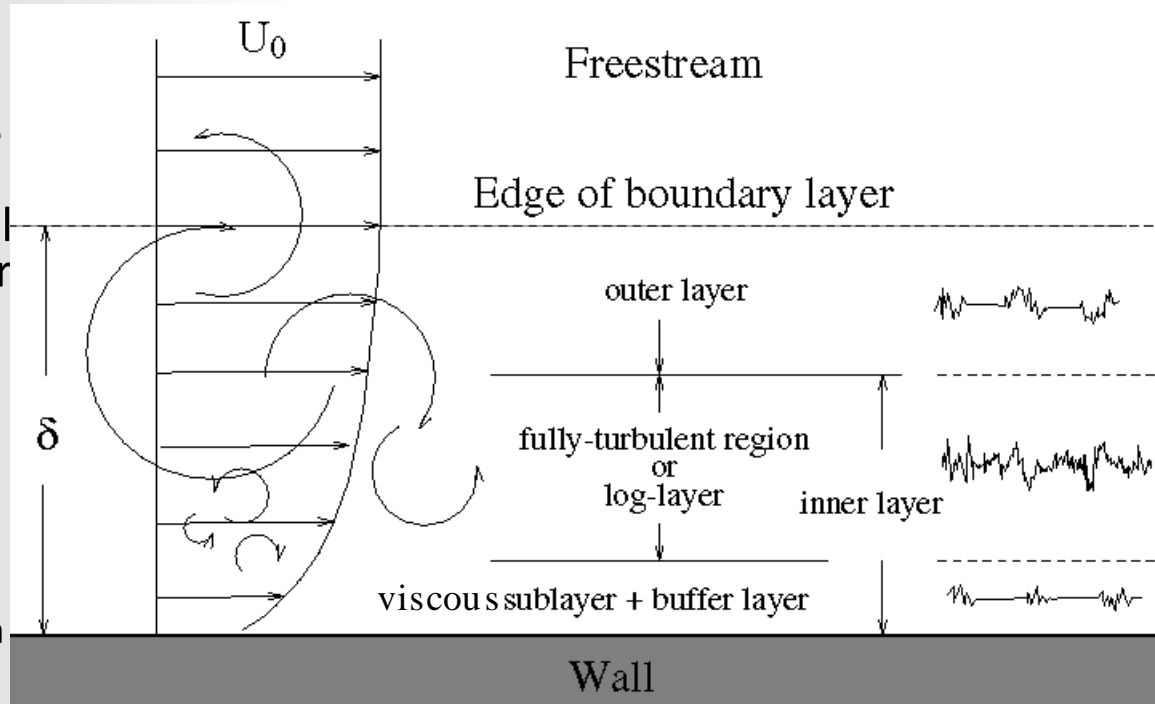
Turbulence Model Selection: A Practical Approach

- Fluent offers the choice of many turbulence models
 - Some are used only for very specific applications
 - Consider advanced training to learn more about the details of each model
- For getting started
 - The **Realizable k- ϵ** or **SST k- ω** models are recommended choices for standard cases
 - Where highly accurate resolution of boundary layers is critical, such as applications involving flow separation or finely resolved heat transfer profiles, SST k- ω is preferred
 - If only a crude estimate of turbulence is required, the standard k- ϵ model can be used
 - This might occur in problems where the solution depends more strongly on other physical models or modeling assumptions than on the turbulence model



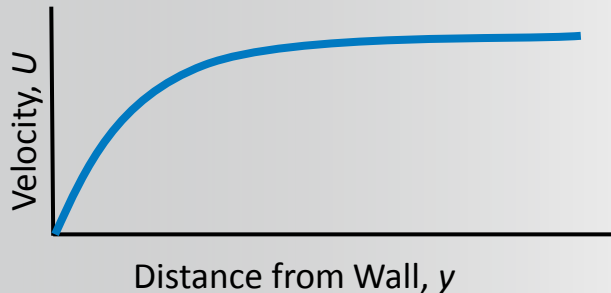
Turbulent Boundary Layers

- A turbulent boundary layer consists of distinct regions
- For CFD, the most important are the viscous sublayer, immediately adjacent to the wall and the log-layer, slightly further away from the wall
- Different turbulence models require different inputs depending on whether the simulation needs to resolve the viscous sublayer with the mesh
 - This is an important consideration in a turbulent flow simulation and will be described in the next few slides



Boundary Layer Profiles

- Near to a wall, the velocity changes rapidly.



- If we plot the same graph again, where:
 - Log scale axes are used

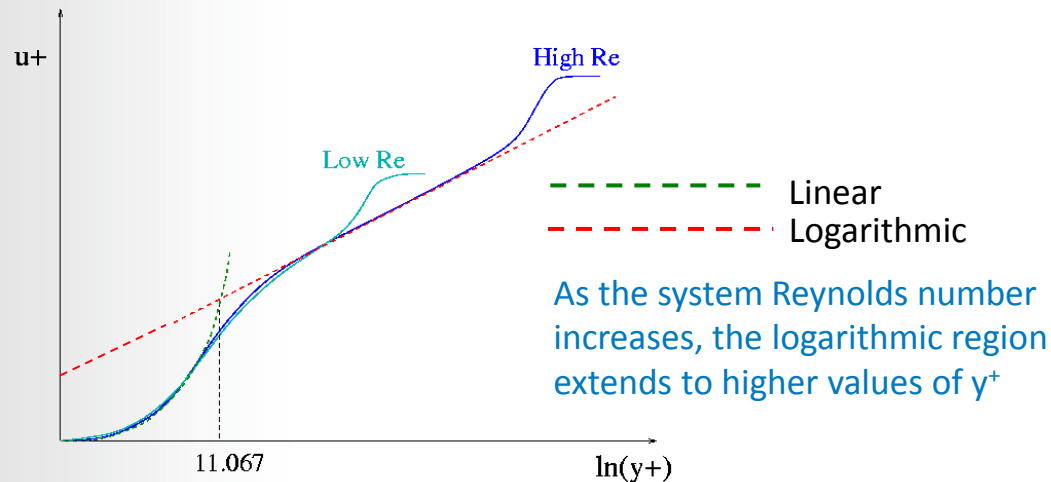
- The velocity is made dimensionless, from U/U_τ where $u_\tau = \sqrt{\frac{\tau_{\text{wall}}}{\rho}}$
- The wall distance is made dimensionless: $y^+ = \frac{yu_\tau}{\nu}$

- Then we arrive at the graph on the next page. The shape of this is generally the same for all flows:

Dimensionless Boundary Layer Profiles

- By scaling the variables near the wall the velocity profile data takes on a predictable form (transitioning from linear in the viscous sublayer to logarithmic behavior in the log-layer)

Using the non-dimensional velocity and non-dimensional distance from the wall results in a predictable boundary layer profile for a wide range of flows



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

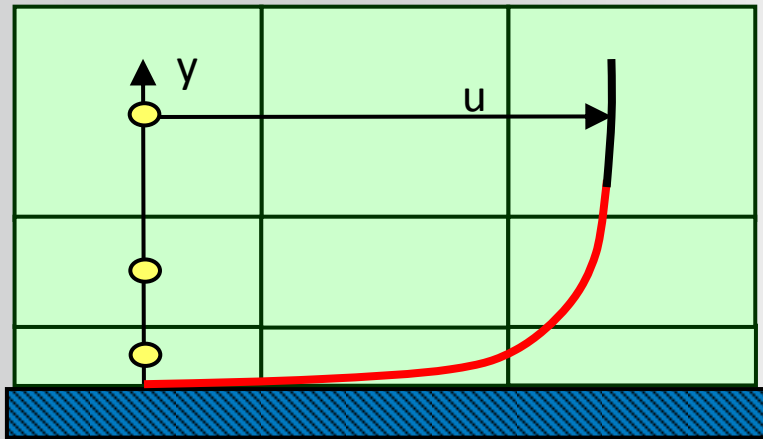
- Wall functions utilize the predictable dimensionless boundary layer profile shown on the previous slide to allow conditions at the wall (e.g. shear stress) to be determined by when the centroid of the wall adjacent mesh cell is located in the log-layer
 - To locate the first cell in the log-layer, it should typically have a y^+ value such that $30 < y^+ < 300$
 - This is a very general guideline - for very high Re, y^+ can be higher if still in log layer and for very low (but still turbulent) Re, the log-layer may not extend far enough away from the wall for the use of wall functions to be valid
 - Wall functions should never be used if $y^+ < 30$
- 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

B) Resolving the Viscous Sublayer

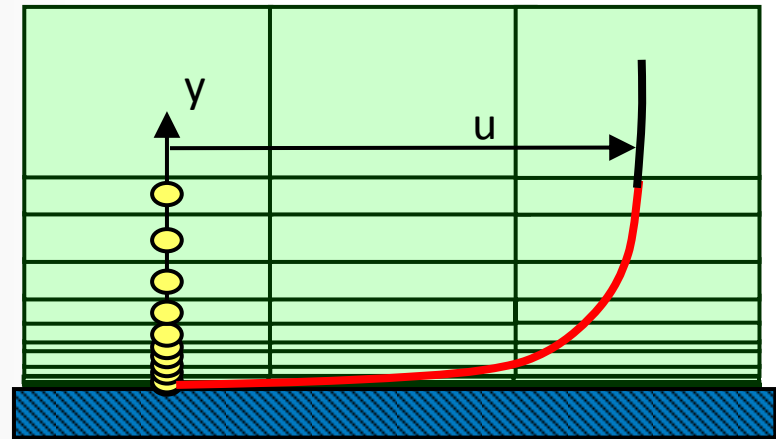
- First grid cell needs to be at about $y^+ \approx 1$ and a prism layer mesh with growth rate no higher than ≈ 1.2 should be used
 - These are not magic numbers – this guideline ensures the mesh will be able to adequately resolve gradients in the sublayer
- This will add significantly to the mesh count (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 $k-\omega$

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 ($Re_l = \frac{\rho V L}{\mu}$) gives $Re_l = 1.4 \times 10^6$

Recall from earlier slide, flow over a surface is turbulent when $Re_L > 5 \times 10^5$

Calculating Wall Distance for a Given y^+

- Begin with the definition of y^+ and rearrange:

$$y^+ = \frac{\rho U_\tau y}{\mu} \iff y = \frac{y^+ \mu}{U_\tau \rho}$$

- The target y^+ value and fluid properties are known, so we need U_τ , which is defined as:

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

- The wall shear stress, τ_w , can be found from the skin friction coefficient, C_f :

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

- A literature search suggests a formula for the skin friction on a plate¹ thus:

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

- Re is known, so use the definitions to calculate the first cell height

$$C_f = 0.058 \text{Re}_l^{-0.2} = .0034$$

$$\tau_w = \frac{1}{2} C_f \rho U_\infty^2 = 0.83 \text{ kg/(m} \cdot \text{s}^2)$$

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

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

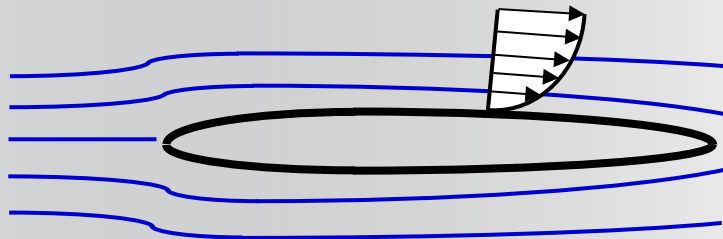
$$y = \frac{y^+ \mu}{U_\tau \rho} = 9 \times 10^{-4} \text{ m}$$

our first cell height y should be approximately 1 mm.

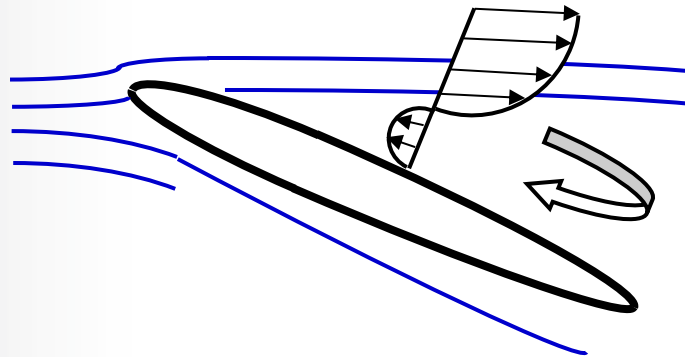
¹An equivalent formula for internal flows, with Reynolds number based on the pipe diameter is $C_f = 0.079 \text{Re}_d^{-0.25}$

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



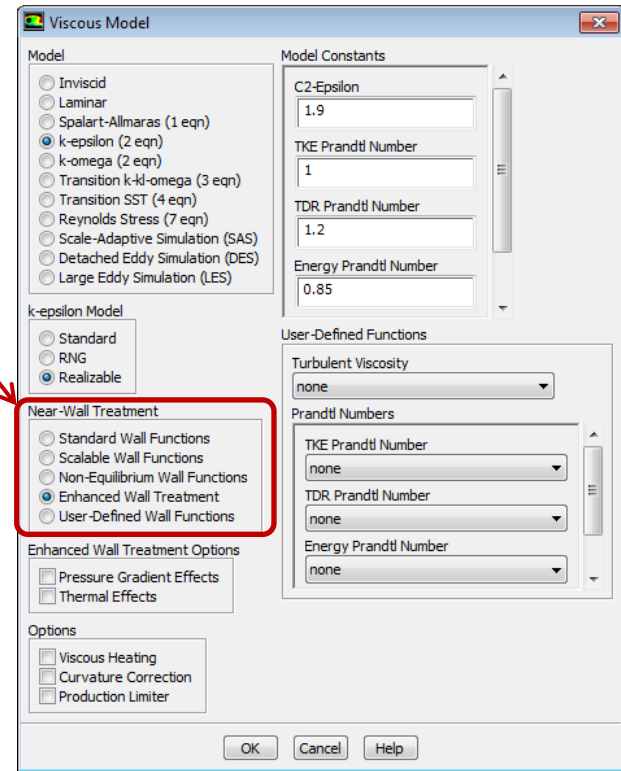
Wall functions not applicable

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

Turbulence Settings for Near Wall Modeling

- If the viscous sublayer is being resolved
 - Use $k-\omega$ models or $k-\epsilon$ models with Enhanced Wall Treatment (EWT)
 - No separate input is needed for $k-\omega$ models
- If wall functions are used
 - Use $k-\epsilon$ models with wall functions
 - EWT can also be used because it is a y^+ insensitive method and will act like a wall function if the first grid point is in the log-layer
 - For $k-\omega$ models
 - The $k-\omega$ models utilize a y^+ insensitive wall treatment and will act like a wall function if the first grid point is in the log layer
 - However, the advantages of these models may be lost when a coarse near-wall mesh is used



Inlet Boundary Conditions

- When turbulent flow enters a domain at inlets or outlets (backflow), boundary conditions must be given for the turbulence model variables
- Four methods for specifying turbulence boundary conditions:

1) Turbulent intensity and viscosity ratio (default)

Default values of turbulent intensity = 5% and turbulent viscosity ratio = 10 are reasonable for cases where you have no information about turbulence at an inlet

2) Turbulent 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) Turbulent intensity and hydraulic diameter (primarily for internal flows)

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

Guidelines for 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 viscosity ratios:
 - Normal turbulent intensities range from 1% to 5%
 - The default turbulent intensity value 5% is sufficient for nominal turbulence through a circular inlet, and is a good estimate in the absence of experimental data
 - For external flows, turbulent viscosity ratio of 1-10 is typically a good value
 - For internal flows, turbulent viscosity ratio of 10-100 it typically a good value
 - For fully developed pipe flow at $Re = 50,000$, the turbulent viscosity ratio is around 100

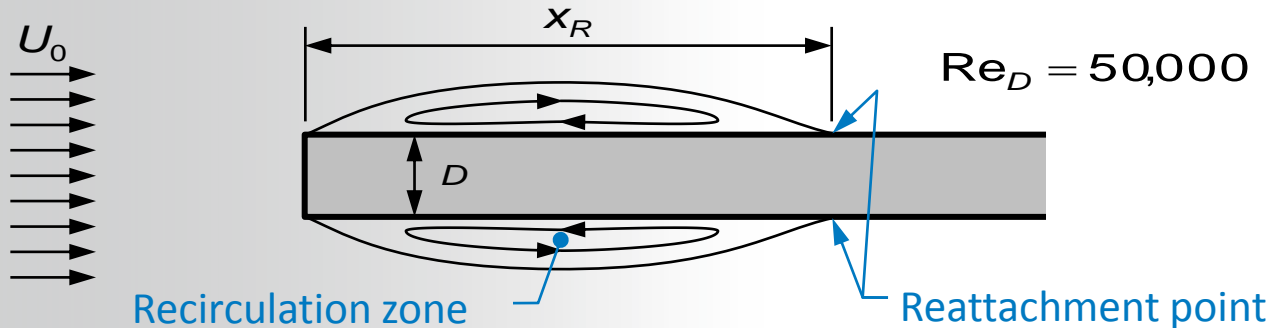
Summary – Turbulence Modeling Guidelines

- To perform a turbulent flow calculation in Fluent
 - Calculate the Reynolds number and determine whether flow is turbulent.
 - Decide on a near-wall modeling strategy
 - The choices are A) Resolve the viscous sublayer or B) Use wall functions
 - Create the mesh with y^+ suitable for the selected approach
 - Choose turbulence model and near wall treatment (if necessary) in the Viscous Models panel
 - Realizable k - ϵ or SST k - ω are recommended choices for standard cases
 - SST k - ω is preferred for cases where the viscous sublayer needs to be resolved (flow separation, detailed heat transfer)
 - Set reasonable boundary conditions for the turbulence model variables

Appendix

Example #1 – Turbulent Flow Past a Blunt Flat Plate

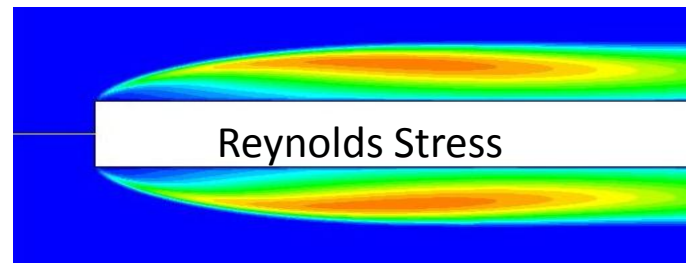
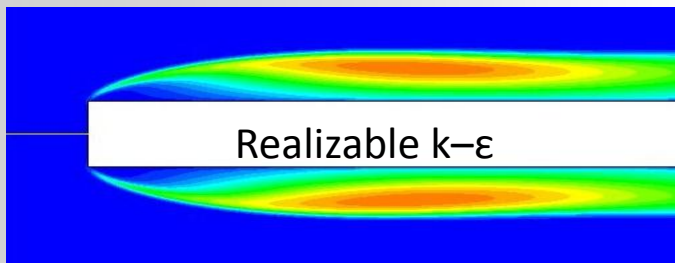
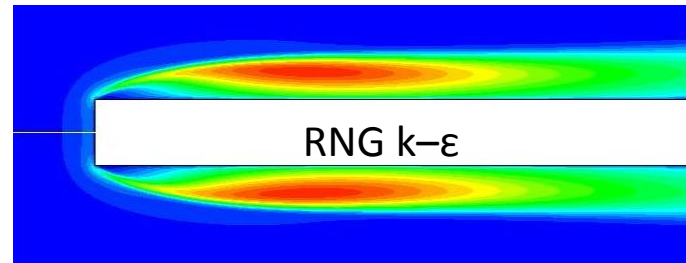
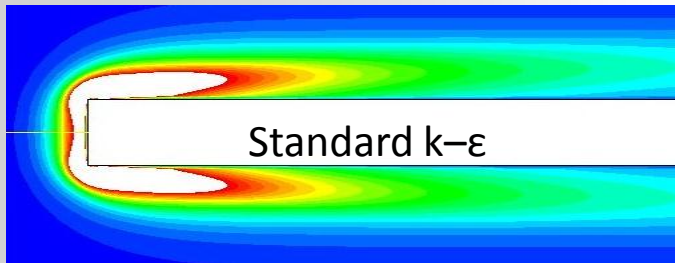
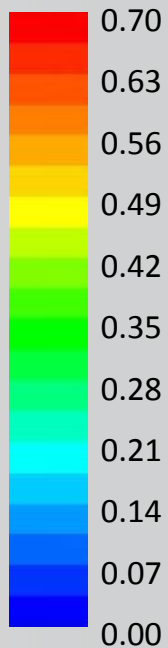
- Turbulent flow past a blunt flat plate was simulated using four different turbulence models.
 - 8,700 cell quad mesh, graded near leading edge and reattachment location.
 - Non-equilibrium boundary layer treatment



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

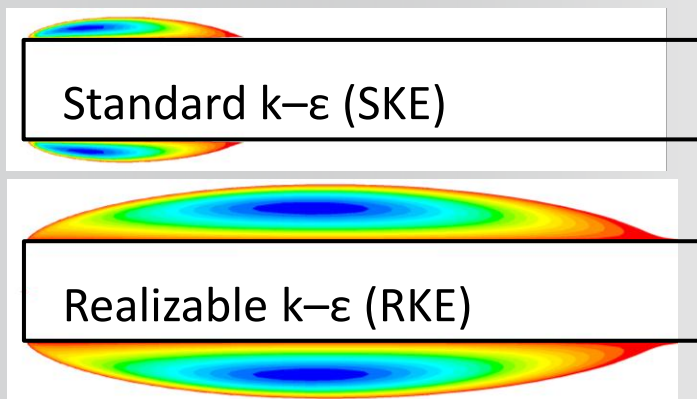
Example #1 Turbulent Flow Past a Blunt Flat Plate

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

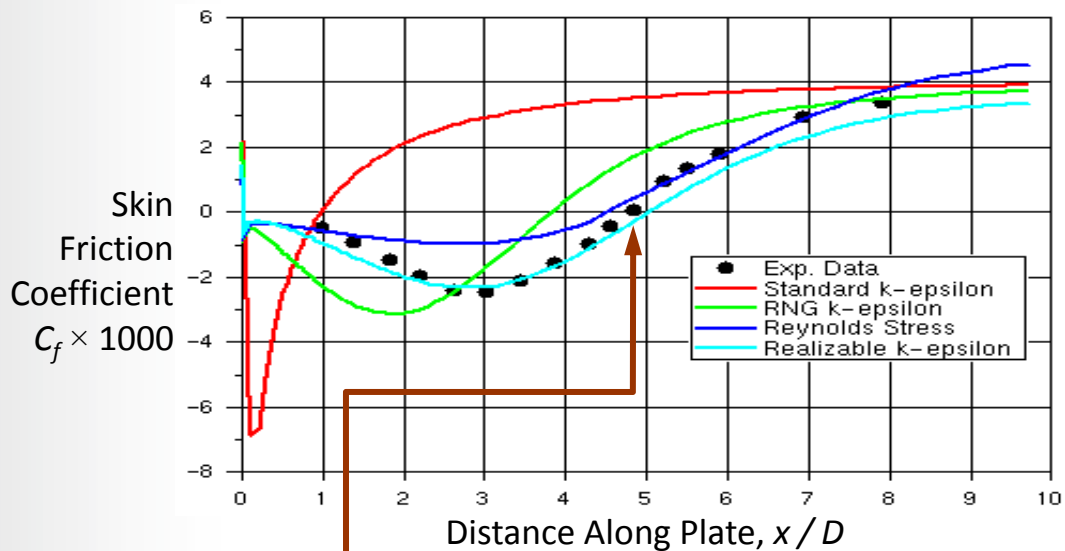


Example #1 Turbulent Flow Past a Blunt Flat Plate

Predicted separation bubble:



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



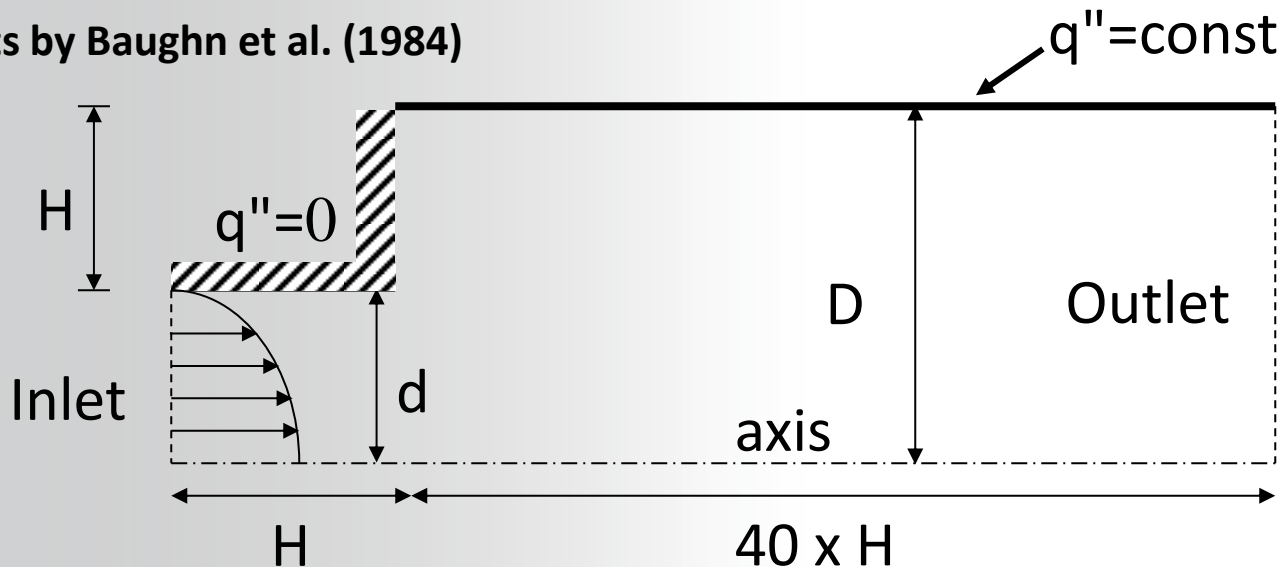
Experimentally observed reattachment point is at $x/D = 4.7$

Example #2 : Pipe Expansion with Heat Transfer

Reynolds Number $Re_D = 40750$

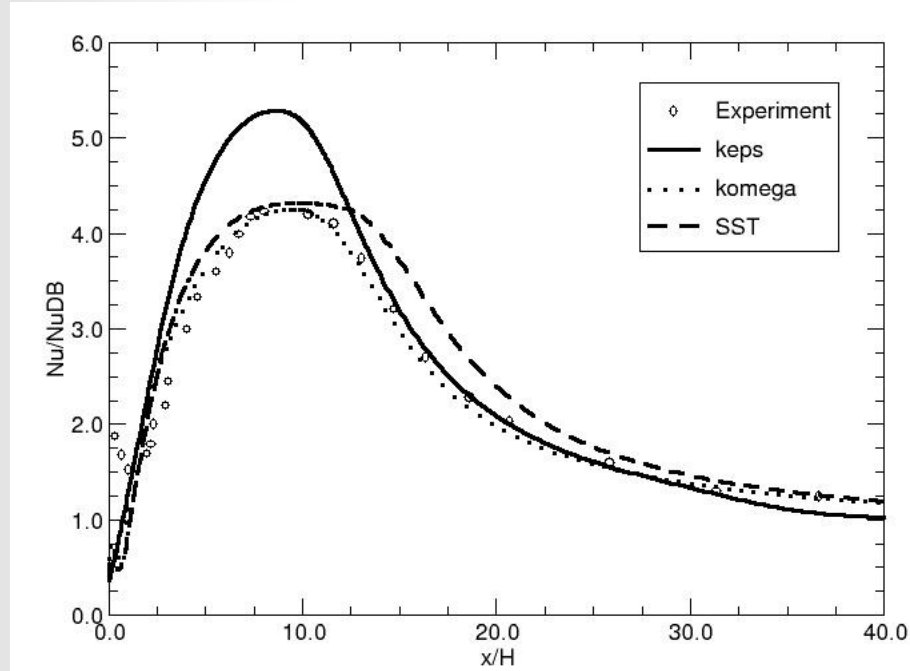
Fully Developed Turbulent Flow at Inlet

Experiments by Baughn et al. (1984)



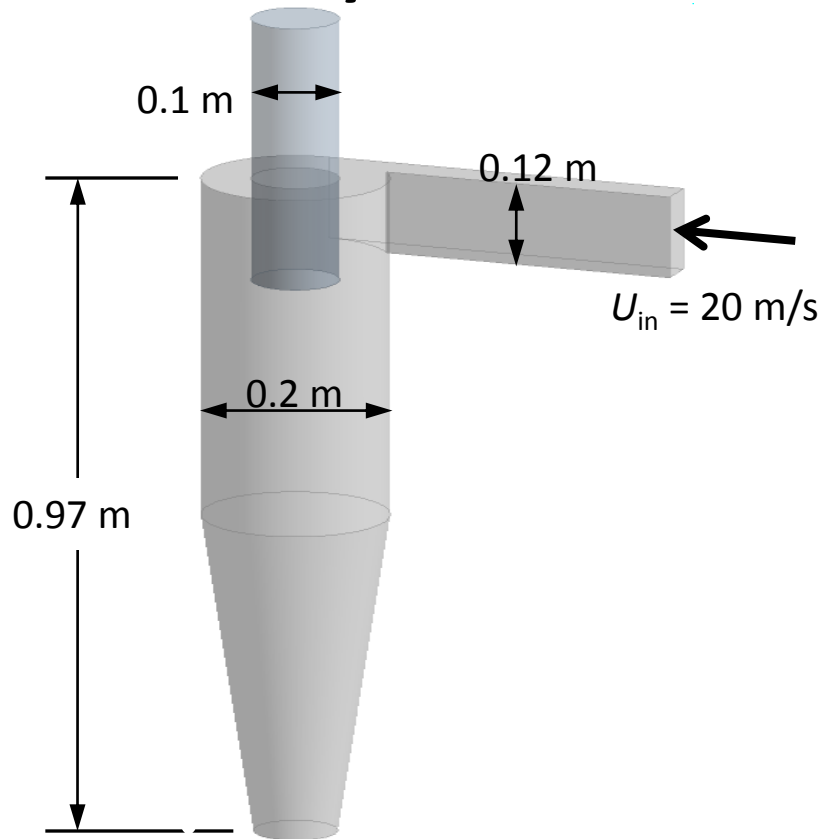
Example #2 : Pipe Expansion with Heat Transfer

- 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



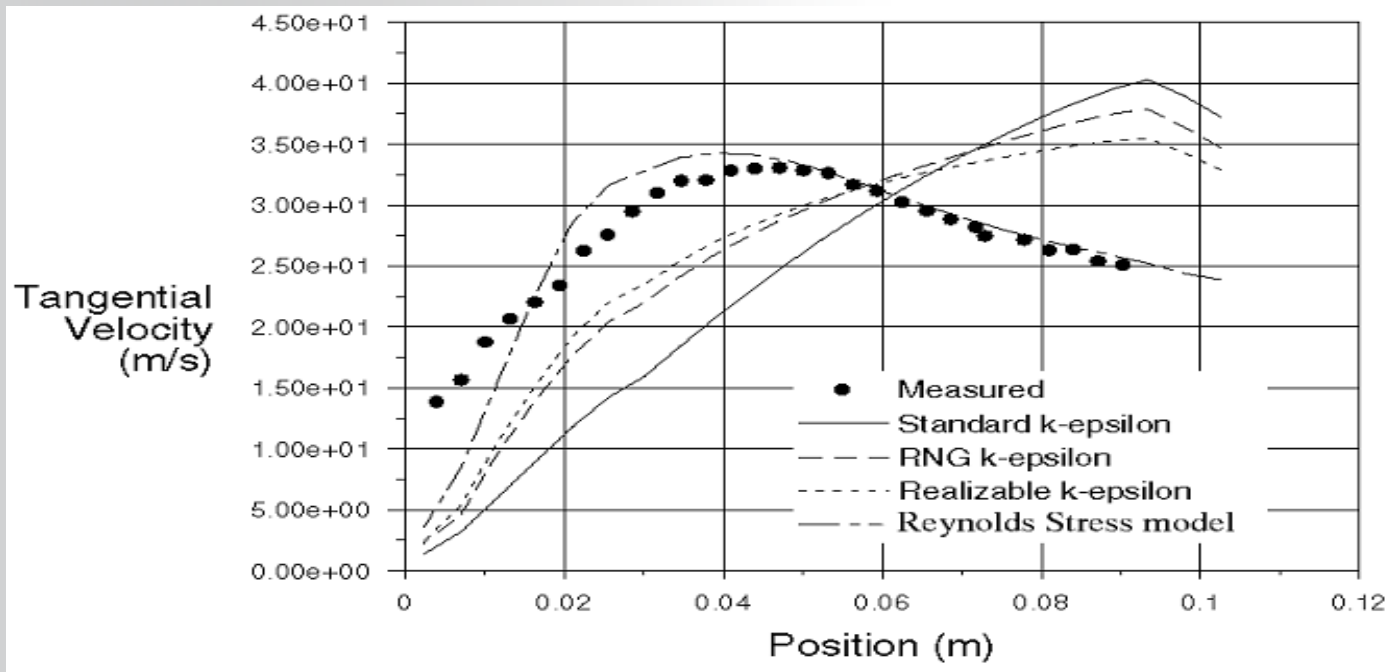
Example #3 Turbulent Flow in a Cyclone

- 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 ($W_{\max} = 1.8 U_{\text{in}}$)



Example #3 Turbulent Flow in a Cyclone

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

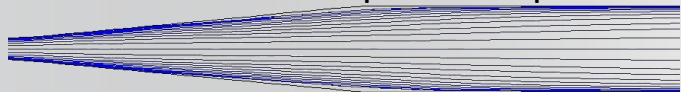


Example 4: Diffuser

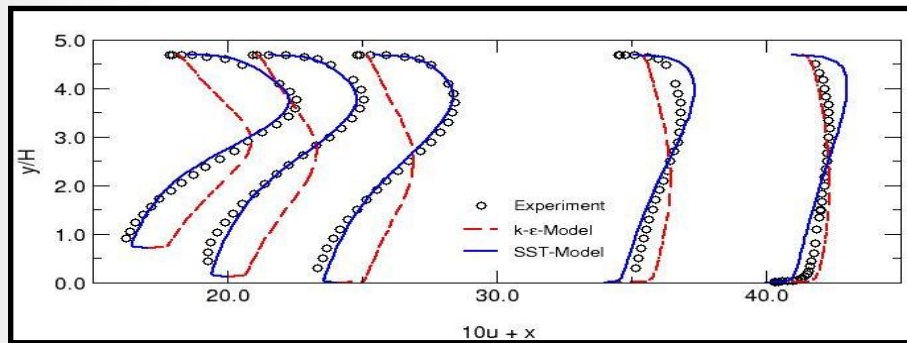
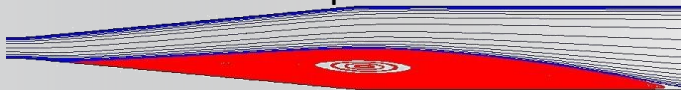
Shear Stress Transport (SST) Model

- 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



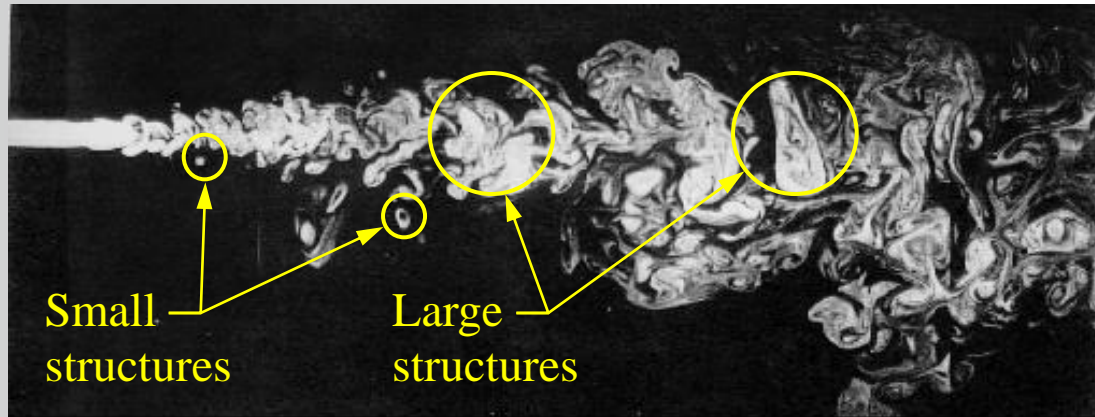
SST result and experiment



Experiment Gersten et al.

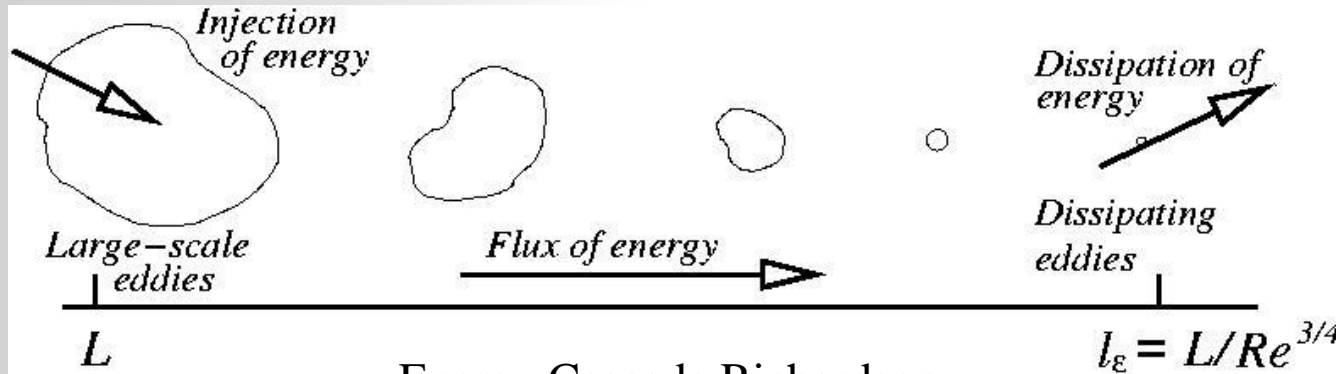
Turbulent Flow Structures

- Turbulent flow characteristics :
 - Unsteady, three-dimensional, irregular, stochastic motion in which transported quantities (mass, momentum, scalar species) fluctuate in time and space
 - Enhanced mixing of these quantities results from the fluctuations
 - Unpredictability in detail
 - Turbulent flows contain a wide range of eddy sizes
 - Large scale coherent structures are different in each flow, whereas small eddies are more universal



Energy Cascade

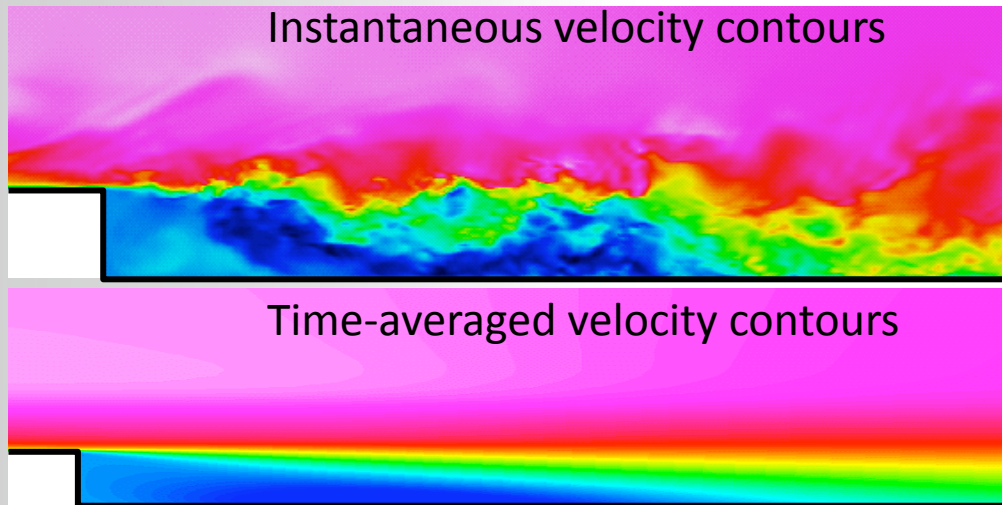
- Energy is transferred from larger eddies to smaller eddies
 - Larger eddies contain most of the energy
 - In the smallest eddies, turbulent energy is converted to internal energy by viscous dissipation



Energy Cascade Richardson
(1922), Kolmogorov (1941)

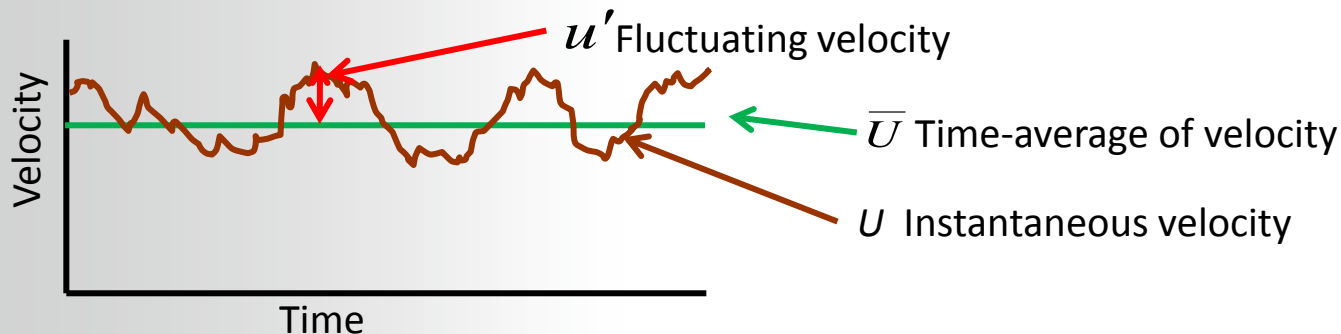
Backward Facing Step

- 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.



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: $U = \bar{U} + u'$
- The time average of the fluctuating velocity must be zero: $\overline{u'} = 0$
- BUT, the RMS of u' is not necessarily zero: $\overline{u'^2} \neq 0$
- The turbulent energy, k , is given by the fluctuating velocity components as: $k = \frac{1}{2} (\overline{u'^2} + \overline{v'^2} + \overline{w'^2})$

RANS Modeling : Averaging

- After decomposing the velocity into mean and instantaneous parts and time-averaging, the instantaneous Navier-Stokes equations may be rewritten as the Reynolds-averaged Navier-Stokes (RANS) equations:

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

$R_{ij} = -\overline{\rho u'_i u'_j}$
 (Reynolds stress tensor)

- The Reynolds stresses are additional unknowns introduced by the averaging procedure, hence they must be modeled (related to the averaged flow quantities) in order to close the system of governing equations

→ 6 unknowns ...

$$R_{ij} = -\overline{\rho u'_i u'_j}$$

$$\begin{pmatrix} -\overline{\rho u'^2} & -\overline{\rho u'v'} & -\overline{\rho u'w'} \\ -\overline{\rho u'v'} & -\overline{\rho v'^2} & -\overline{\rho v'w'} \\ -\overline{\rho u'w'} & -\overline{\rho v'w'} & -\overline{\rho w'^2} \end{pmatrix}$$

RANS Modeling : The Closure Problem

- The Reynolds Stress tensor $R_{ij} = -\rho \overline{u'_i u'_j}$ must be solved
- The RANS equations can be closed in two ways:

Reynolds-Stress Models (RSM)

- R_{ij} is directly solved via transport equations (modeling is still required for many terms in the transport equations)

$$\frac{\partial}{\partial t} (\rho \overline{u'_i u'_j}) + \frac{\partial}{\partial x_k} (\rho \bar{u}_k \overline{u'_i u'_j}) = P_{ij} + F_{ij} + D_{ij}^T + \Phi_{ij} - \epsilon_{ij}$$

- RSM is advantageous in complex 3D turbulent flows with large streamline curvature and swirl, but the model is more complex, computationally intensive, more difficult to converge than eddy viscosity models

- Note: All turbulence models contain empiricism
 - Equations cannot be derived from fundamental principles
 - Some calibrating to observed solutions and “intelligent guessing” is contained in the models

Eddy Viscosity Models

- Boussinesq hypothesis
→ Reynolds stresses are modeled using an eddy (or turbulent) viscosity, μ_T

$$R_{ij} = -\rho \overline{u'_i u'_j} = \mu_T \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \frac{2}{3} \mu_T \frac{\partial \bar{u}_k}{\partial x_k} \delta_{ij} - \frac{2}{3} \rho k \delta_{ij}$$

- The hypothesis is reasonable for simple turbulent shear flows: boundary layers, round jets, mixing layers, channel flows, etc.

Two-Equation Models

- Two transport equations are solved, giving two independent scales for calculating μ_t

- Virtually all use the transport equation for the turbulent kinetic energy, k

$$\rho \frac{Dk}{Dt} = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + \underset{\substack{\uparrow \\ \text{production}}}{P} - \underset{\substack{\leftarrow \\ \text{dissipation}}}{\rho \varepsilon}; \quad P = \mu_t S^2 (ske) \quad S = \sqrt{2S_{ij}S_{ij}}$$

- Several transport variables have been proposed, based on dimensional arguments, and used for second equation. The eddy viscosity μ_t is then formulated from the two transport variables.

- Kolmogorov, ω : $\mu_t \propto \rho k / \omega$, $l \propto k^{1/2} / \omega$, $k \propto \varepsilon / \omega$
 - ω is specific dissipation rate
 - defined in terms of large eddy scales that define supply rate of k
- Chou, ε : $\mu_t \propto \rho k^2 / \varepsilon$, $l \propto k^{3/2} / \varepsilon$
- Rotta, l : $\mu_t \propto \rho k^{1/2} l$, $\varepsilon \propto k^{3/2} / l$

Standard k - ε Model Equations

k -transport equation

$$\rho \frac{Dk}{Dt} = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + \underbrace{\mu_t S^2}_{\text{production}} - \underbrace{\rho \varepsilon}_{\text{dissipation}}; \quad S = \sqrt{2S_{ij}S_{ij}}$$

ε -transport equation

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

inverse time scale

coefficients

$\sigma_k, \sigma_\varepsilon, C_{1\varepsilon}, C_{2\varepsilon}$ ← Empirical constants determined from benchmark experiments of simple flows using air and water.

turbulent viscosity $\mu_t = \rho C_\mu \frac{k^2}{\varepsilon}$

RANS : EVM :Standard k - ϵ (SKE) Model

- 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.

- **Realizable $k-\epsilon$ (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

OFTEN PREFERRED TO STANDARD K-EPSILON.

RANS : EVM : Spalart-Allmaras (S-A) Model

- **Spalart-Allmaras** is a low-cost RANS model solving a single transport equation for a modified eddy viscosity
- Designed specifically for aerospace applications involving wall-bounded flows
 - Has been shown to give good results for boundary layers subjected to adverse pressure gradients.
 - Used mainly for aerospace and turbomachinery applications
- Limitations:
 - The model was designed for wall bounded flows and flows with mild separation and recirculation.
 - No claim is made regarding its applicability to all types of complex engineering flows.

k-omega Models

- In k- ω models, the transport equation for the turbulent dissipation rate, ε , is replaced with an equation for the specific dissipation rate, ω
 - The turbulent kinetic energy transport equation is still solved
- k- ω models have gained popularity in recent years mainly because:
 - Much better performance than k- ε models for boundary layer flows
 - For separation, transition, low Re effects, and impingement, k- ω models are more accurate than k- ε models
 - Accurate and robust for a wide range of boundary layer flows with pressure gradient
- **Two variations of the k- ω model are available in Fluent**
 - Standard k- ω model (Wilcox, 1998)
 - SST k- ω model (Menter)

- **k - ω models are RANS two-equations based models**

$$R_{ij} = -\rho \overline{u'_i u'_j} = \mu_T \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \frac{2}{3} \mu_T \frac{\partial \bar{u}_k}{\partial x_k} \delta_{ij} - \frac{2}{3} \rho k \delta_{ij}$$

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

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

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

ω = specific dissipation rate

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

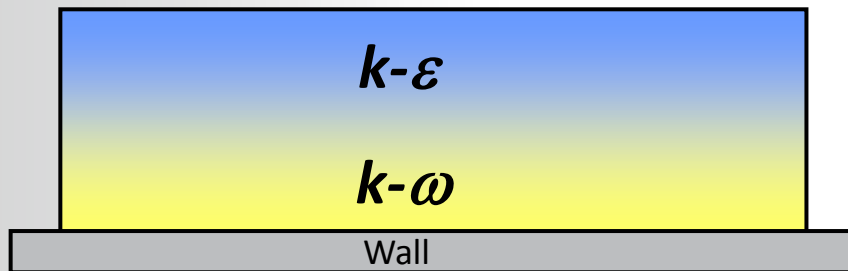
- **One of the advantages of the k - ω formulation is the near wall treatment for low-Reynolds number computations**

- designed to predict correct behavior when integrated to the wall
 - the k - ω models switches between a viscous sublayer formulation (i.e. direct resolution of the boundary layer) at low y^+ values and a wall function approach at higher y^+ values
- while k - ε model variations require Enhanced Wall Treatment to capture correct viscous sublayer behavior

SST Model

- Shear Stress Transport (SST) Model

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



- The $k-\varepsilon$ and $k-\omega$ models are blended such that the SST model functions like the $k-\omega$ close to the wall and the $k-\varepsilon$ model in the freestream

SST is a good compromise between $k-\varepsilon$ and $k-\omega$ models

RANS: Other Models in Fluent

- **RNG k- ϵ model**

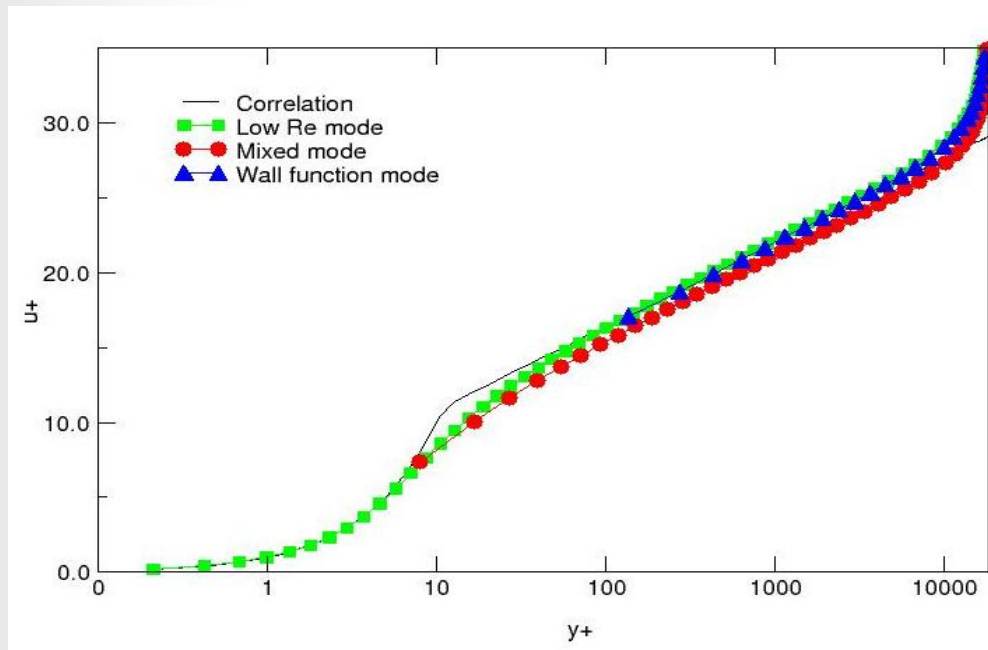
- Model constants are derived from renormalization group (RNG) theory instead of empiricism
- Advantages over the standard k- ϵ model are very similar to those of the RKE model

- **Reynolds Stress model (RSM)**

- Instead of using eddy viscosity to close the RANS equations, RSM solves transport equations for the individual Reynolds stresses
 - 7 additional equations in 3D, compared to 2 additional equations with EVM.
- Much more computationally expensive than EVM and generally very difficult to converge
 - As a result, RSM is used primarily in flows where eddy viscosity models are known to fail
 - These are mainly flows where strong swirl is the predominant flow feature, for instance a cyclone

Enhanced Wall Treatment (EWT)

- Need for y^+ insensitive wall treatment
- EWT smoothly varies from low-Re to wall function with mesh resolution
- EWT available for $k-\epsilon$ and RSM models
- Similar approach implemented for $k-\omega$ equation based models, and for the Spalart-Allmaras model



y^+ for the SST and k- ω Models

- **The SST and k- ω models were formulated to be near-wall resolving models where the viscous sublayer is resolved by the mesh**
 - To take full advantage of this formulation, y^+ should be ≈ 1
 - This is necessary for accurate prediction of flow separation

- **These models can still be used with a coarser near-wall mesh and produce valid results, within the limitations of logarithmic wall functions**
 - The first grid point should still be in the logarithmic layer ($y^+ < 300$ for most flows)
 - Many advantages of these models may be lost when a coarse near-wall mesh is used

RANS Turbulence Model 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 the strain rate in the 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 to k–ε models for wall-bounded and low Reynolds number flows. Options account for low Reynolds number effects, 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 $\tau_T \sim k$.
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.

* Realizable k-ε or SST k-ω are the recommended choice for standard cases

Turbulent Scales Related to k and ε

- Characteristics of the Turbulent Structures:

- Length scale : l [m]
- Velocity scale : \sqrt{k} [m/s]
- Time scale : $\frac{l}{\sqrt{k}}$ [s]

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

- Turbulent kinetic energy : $k = \frac{1}{2} \left(\overline{u'^2} + \overline{v'^2} + \overline{w'^2} \right)$ [m²/s²]

- Turbulent kinetic energy dissipation : ε [m²/s³] $\sim k^{3/2}/l$ (dimensional analysis)

- Turbulent Reynolds : $Re_t = k^{1/2}l/\nu \sim k^2/\nu\varepsilon$ [-]

- Turbulent Intensity : $I = \frac{u'}{U} \approx \frac{1}{U} \sqrt{\frac{2k}{3}}$ [-]