The Most Accurate and Advanced Turbulence Capabilities

Peyman Davoudabadi
Confidence by Design Workshop
Chicago, IL June 14, 2012
Agenda

- **Introduction to Turbulent Flows**
  - Characteristics of a turbulent flow
  - Challenges in simulation of turbulent flows

- **Unsteady Modeling with Scale-Resolving Simulations (SRS)**
  - Large Eddy Simulations
  - Detached Eddy Simulations

- **Steady State Turbulence Modeling**
  - Reynolds Averaged Navier-Stokes Simulations (RANS), e.g., $k-\varepsilon$, $k-\omega$, RSM
Agenda

• Introduction to Turbulent Flows
  ✓ Characteristics of a turbulent flow
  ✓ Challenges in simulation of turbulent flows

• Unsteady Modeling with Scale-Resolving Simulations (SRS)
  ✓ Large Eddy Simulations
  ✓ Detached Eddy Simulations

• Steady State Turbulence Modeling
  ✓ Reynolds Averaged Navier-Stokes Simulations (RANS), e.g.,
    \( k-\varepsilon, k-\omega, \) RSM
Characteristics of Turbulent Flows

- Unsteady, irregular (aperiodic) motion in which transported quantities (mass, momentum, scalar species) fluctuate in time and space
  - The fluctuations are responsible for enhanced mixing of transported quantities
- Instantaneous fluctuations are random (unpredictable, irregular) both in space and time
  - Statistical averaging of fluctuations results in accountable, turbulence related transport mechanisms
- Contains a wide range of eddy sizes (scales)
  - Large eddies ‘carry’ small eddies
  - The behavior of large eddies is different in each flow
    - Sensitive to upstream history
  - The behavior of small eddies is more universal in nature
Reynolds Number

• The Reynolds number is defined as

$$Re_L = \frac{\rho UL}{\mu}$$

where $U$ and $L$ are representative velocity and length scales for a given flow. $L = x, d, d_h, etc.$

• Turbulent flows occur at large Reynolds numbers

**External Flows**

- $Re_x \geq 500,000$ along a surface
- $Re_d \geq 20,000$ around an obstacle

**Internal Flows**

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

**Natural Convection**

$$Ra = \frac{\beta g L^3 \Delta T}{\nu \alpha} = \frac{\rho^2 c_p \beta g L^3 \Delta T}{\mu k}$$

(Rayleigh number)

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

where

$$Pr = \frac{\nu}{\alpha} = \frac{\mu c_p}{k}$$

(Prandtl number)
Two Examples of Turbulence

Homogeneous, decaying, grid-generated turbulence

Turbulent boundary layer on a flat plate
Energy Cascade

- **Injection of energy from mean flow**
- **Flux of Energy**
- **Dissipation of energy**

Larger scale eddies

\[ l \gg \eta \]

\[ \eta \approx l/\text{Re}_T^{3/4} \]

Resolved

**Direct Numerical Simulation (DNS)**

\[
\rho \left( \frac{\partial U_i}{\partial t} + U_k \frac{\partial U_i}{\partial x_k} \right) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_k} \left( \mu \frac{\partial U_i}{\partial x_j} \right)
\]
Smallest Scales of Turbulence

- Smallest eddy --- the Kolmogorov scales:

  - Large eddy energy supply rate is balanced by the small eddy energy dissipation rate $\varepsilon = -\frac{dk}{dt}$

  - $k \equiv \frac{1}{2}(u''^2 + v''^2 + w''^2)$ is (specific) turbulent kinetic energy $[L^2 / T^2]$

  - $\varepsilon$ is dissipation rate of $k$ $[L^2 / T^3]$

- Motion at smallest scales dependent upon dissipation rate, $\varepsilon$, and kinematic viscosity, $\nu [L^2 / T]$

- From dimensional analysis, the Kolmogorov scales can be estimated as follows:

  - Length scale: $\eta = (\nu^3 / \varepsilon)^{1/4}$
  - Time scale: $\tau = (\nu / \varepsilon)^{1/2}$
  - Velocity scale: $V = (\nu \varepsilon)^{1/4}$
Implication of Scales

• Consider a mesh fine enough to resolve smallest eddies and large enough to capture mean flow features

• Example: 2D channel flow

• \( N_{\text{cells}} \sim (4l/\eta)^3 \)
  
or
  \( N_{\text{cells}} \sim (3\text{Re}_\tau)^{9/4} \)

  where

  \( \text{Re}_\tau = u_\tau H / 2 \nu \)

• \( \text{Re}_H = 30,800 \rightarrow \text{Re}_\tau = 800 \rightarrow N_{\text{cells}} = 4 \times 10^7! \)

  Numerical time step size required, \( \Delta t \sim \tau \)

  \[
  \Delta t_{\text{2D Channel}} \approx \frac{0.003H}{\sqrt{\text{Re}_\tau u_\tau}}
  \]

  \textbf{Number of time steps} \sim 48,000!
Removing the Small Scales

• Two methods can be used to eliminate need to resolve small scales:
  – Reynolds Averaging
    • Transport equations for mean flow quantities are solved
    • All scales of turbulence are modeled
    • Transient solution ($\Delta t$) is set by global unsteadiness
  – Filtering (LES)
    • Transport equations for ‘resolvable scales’
    • Resolves larger eddies; models smaller ones
    • Inherently unsteady ($\Delta t$) dictated by smallest resolved eddies

• Both methods introduce additional terms that must be modeled for closure
Energy Cascade

Injection of energy from mean flow

Larges scale eddies

Flux of Energy

Dissipating eddies

\[ \eta \approx l / Re^{3/4} \]

Resolved

SRS (Scale Resolving Simulations), e.g., LES

Modeled

\[ \Delta_{LES} \]

Modeled

RANS (Reynolds Averaged Navier-Stokes Simulations)
# Turbulent Flow Simulation Methods

<table>
<thead>
<tr>
<th>Method</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>DNS</strong></td>
<td>Numerically solving the full unsteady Navier-Stokes equations</td>
</tr>
<tr>
<td><strong>SRS</strong></td>
<td>Includes Large Eddy Simulation (LES)</td>
</tr>
<tr>
<td><strong>RANS</strong></td>
<td>Solve Reynolds-averaged Navier-Stokes equations (time-average)</td>
</tr>
</tbody>
</table>

- **DNS** (Direct Numerical Simulation)
  - Numerically solving the full unsteady Navier-Stokes equations
  - No modeling is required
  - A useful research tool only
  - Feasible only for simple geometries and low turbulent Reynolds numbers
  - **DNS is not suitable for practical industrial CFD**

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

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

• In other words, \( U(\bar{x}, t) = \bar{U}(\bar{x}) + u'(\bar{x}) \)

• We can apply the same time averaging procedure to the governing equations, which gives us the Reynolds-Averaged Navier-Stokes (RANS) equations

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

• RANS turbulence models provide closure for the Reynolds stress tensor, which represents the effect of turbulent fluctuations on the mean flow. This allows us to perform steady state simulations of turbulent flow.
Comparison of SRS and RANS

**RANS (Reynolds Averaged Navier-Stokes Simulations)**

- **Advantages**: For many applications, steady state solutions are preferable because of
  - Smaller computational overhead
  - Simplified post-processing
  - In many cases, only time-averaged values are of interest and a good RANS model with a good quality grid will provide all the required accuracy
- **Disadvantages**: For some flows, challenges associated with RANS modeling can limit the level of accuracy that it is possible to attain

**SRS (Scale Resolving Simulations)**

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

• Example: Channel flow at Re = 114,000
  – 2D Periodic case; Boundary layer thickness, $\delta$, equal to channel half-width

• Top: Wall Modeled LES
  – 1.2 million cells, transient calculation, run time is order of days

• Below: RANS
  – 140 cells, steady calculation, run time is order of minutes

• Important
  – For wall-bounded flows, in a more typical 3D industrial geometry, RANS would still be 2 orders of magnitude fewer cells and run times of hours versus days.
Agenda

• **Introduction to Turbulent Flows**
  ✓ Characteristics of a turbulent flow
  ✓ Challenges in simulation of turbulent flows

• **Unsteady Modeling with Scale-Resolving Simulations (SRS)**
  ✓ Large Eddy Simulations
  ✓ Detached Eddy Simulations

• **Steady State Turbulence Modeling**
  ✓ Reynolds Averaged Navier-Stokes Simulations (RANS), e.g.,
    $k-\varepsilon$, $k-\omega$, RSM
Motivation for Scale-Resolving Simulations (SRS)

• **Accuracy Improvements over RANS**
  - Flows with large separation zones (stalled airfoils/wings, flow past buildings, flows with swirl instabilities, etc.)

• **Additional information required**
  - **High-fidelity external aerodynamics**
  - **Acoustics** - Information on acoustic spectrum not reliable from RANS
  - **Vortex cavitation** – low pressure inside vortex causes cavitation – resolution of vortex required
  - **Fluid-Structure Interaction (FSI)** – unsteady forces determine frequency response of solid

Munch Pump Simulations with SAS-SST Model
Example: Embedded Large Eddy Simulation of Flow Around the Ahmed Body (challenging/popular test case)

- ANSYS-FLUENT R13.0 has been utilized
- RANS Model: Unsteady SST k–ω
- SAS Model: Zonal LES-RANS technique: ELES
- Detailed LDA Measurements from Becker and Lienhart

U velocity profiles

Scale Resolving Models in ANSYS FLUENT

- **Large Eddy Simulation (LES)**
  - Smagorinsky-Lilly (+ dynamic)
  - WALE model
  - Algebraic Wall Modeled LES (WMLES)
  - Dynamic kinetic energy subgrid model

- **Detached Eddy Simulation (DES) models**
  - DES-SST, DES-Spalart-Allmaras, and DES-Realizable k-ε

- **Scale-Adaptive Simulation (SAS) models**
  - SAS-SST model

- **Embedded LES (ELES) model**
  - Combination of all RANS modes with all non-dynamic LES models

- **Synthetic Turbulence Generator**
  - Vortex method
SRS Models for External Aerodynamics

- High fidelity unsteady turbulence modeling with DES, LES and SAS models
- Scale Adaptive Simulation (SAS) method provides “LES” level of flow-field capture in unsteady regions at less than half the cost of LES model
- Body fitted mesh allows large aspect ratio prism layers near the boundary without compromising with the accuracy of near wall solution
  - This helps to cut down the overall cell count drastically
- Delayed DES model to avoid grid induced separation
Example 1: DES/DDES of Separated Flow around a Realistic Car Model Exposed to Crosswind

<table>
<thead>
<tr>
<th>Model</th>
<th>Exp.</th>
<th>DDES</th>
<th>DES</th>
<th>LES</th>
</tr>
</thead>
<tbody>
<tr>
<td>Drag (SCx)</td>
<td>0.70</td>
<td>0.71</td>
<td>0.75</td>
<td>0.69</td>
</tr>
</tbody>
</table>

U=40 m/s
Yaw angle 20°
Re_H ~10^6

Grid Induced Separation with DES
Shielding with DDES

Courteys PSA Peugeot Citroën
Example 2: DES Modeling of NASA GCM 1/8th Scale Truck Model

- 6 deg Yaw case
- Velocity = 14 mph
- **Detached Eddy Simulation** model with Spalart-Allmaras option for turbulence
- Second order transient formulation
- P-V Coupling = SIMPLEC
Best Practices for Scale Resolving Simulations

- Best practices document written by Dr. Florian Menter
  - Everything you want to know about SRS in ANSYS
Agenda

• Introduction to Turbulent Flows
  ✓ Characteristics of a turbulent flow
  ✓ Challenges in simulation of turbulent flows

• Unsteady Modeling with Scale-Resolving Simulations (SRS)
  ✓ Large Eddy Simulations
  ✓ Detached Eddy Simulations

• Steady State Turbulence Modeling
  ✓ Reynolds Averaged Navier-Stokes Simulations (RANS), e.g.,
    \( k-\varepsilon \), \( k-\omega \), RSM
Overview

• The Role of Steady State (RANS) Turbulence Modeling

• Overview of Reynolds-Averaged Navier Stokes (RANS) Modeling Capabilities in ANSYS CFD
  – Overview of models
  – Wall treatment
  – Model extensions and other interesting new features in R14.0 and R14.5
Steady-State RANS Modeling Capabilities

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

Example: Optimization study achieves 1/3 reduction in pressure drop in U-bend over 30 different design iterations

The CFD user community requires a broad range of models to choose from in order to meet its needs

- Over 2/3 of all simulations reported using some variation of 1 or 2 equation model (S-A, k-ε family, k-ω family)
- In some applications, one model may be more dominant than others (example: aerodynamics & SST, cyclones & RSM), but for a broad range of applications, a variety of models is needed to match the appropriate model to the appropriate application
A wide array of models is available for steady state calculations

- Includes all commonly used models in the CFD World
- Includes useful extensions to the models such as Curvature Correction and EARSM
- Important to be able to ensure whatever the application, you can choose the most suitable model

<table>
<thead>
<tr>
<th>Steady RANS Turbulence Models in ANSYS</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>One-Equation Models</strong></td>
</tr>
<tr>
<td><strong>Spalart-Allmaras</strong></td>
</tr>
<tr>
<td><strong>Two-Equation Models</strong></td>
</tr>
<tr>
<td><strong>k–ε (Standard, Realizable, RNG)</strong></td>
</tr>
<tr>
<td><strong>k–ω (Standard, SST)</strong></td>
</tr>
<tr>
<td><strong>Curvature Correction (all 1 &amp; 2 eqn. models)</strong></td>
</tr>
<tr>
<td>*<em>V2F (4 eqn.)</em></td>
</tr>
<tr>
<td><strong>Explicit Algebraic Reynolds Stress Model (EARSM)</strong></td>
</tr>
<tr>
<td><strong>Reynolds Stress Models</strong></td>
</tr>
<tr>
<td><strong>Lauder-Reece-Rodi, Speziale-Sarkar-Gatski</strong></td>
</tr>
<tr>
<td><strong>Stress-ω</strong></td>
</tr>
<tr>
<td><strong>k–kl–ω Transition Model</strong></td>
</tr>
<tr>
<td><strong>SST Transition Model</strong></td>
</tr>
</tbody>
</table>

* additional license required
• It is not enough just to provide many choices

• More importantly, for the models that are available, emphasis is placed on
  – Correct implementation
    • Models should be well understood and tested
  – Accurate and validated for some class(es) of applications
  – Robust performance on all mesh topologies
  – Interoperability with other physical models, e.g. multiphase, dynamic mesh, combustion, etc.
  – Wall treatment

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

*Courtesy of the University of Bologna*
Near Wall Turbulence and the Law of the Wall

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

\[ \frac{U}{U_\tau} = 2.5 \ln\left( \frac{U_\tau y}{\nu} \right) + 5.45 \]

\[ U_\tau = \sqrt{\frac{\tau_w}{\rho}} \]

\[ y^+ = \frac{y U_\tau}{\nu} \]

where \( y \) is the normal distance from the wall.

\[ u^+ = \frac{u}{U_\tau} \]
Viscous Sub-layer Modeling Approach

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

1st cell centroid at $y^+ \sim 1$
Cases where high near-wall resolution is unaffordable. Wall functions bridge the gap between the wall and the log region where the first cell centroid is located.

1st cell centroid at $30 < y^+ < 300$
The Importance of Mesh Insensitive Wall Treatment

- In practice, maintaining a prescribed value of $y^+$ in wall-adjacent cells throughout the domain for industrial cases is challenging.

- Maintaining a value of $y^+ > 30$ when using wall functions under cases of grid refinement can be especially problematic.

- Grid refinement can be a critical component of achieving a grid-independent solution, which is a universally accepted CFD best practice.

- **Wall treatments that are insensitive to $y^+$ values as the mesh is refined are critical for RANS models in industrial CFD**.
Y⁺ Insensitive Treatments in ANSYS CFD

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

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

Boundary layer velocity profile modeled with Standard k-ε for three different mesh densities using Enhanced Wall Treatment
RANS Model Extensions

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

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

**Case 1**
- Air: 5 m/s
- Water: 1 m/s

**Case 2**
- Air: 5 m/s
- (Single phase case with only air flowing over moving wall)
- Wall Velocity = 1 m/s

**Velocity profile in air region**

**Fine mesh vs Coarse mesh with turbulence damping**
- Fine mesh: 77520 cells
- Coarse mesh: 19380 cells

**Graphs:**
- Single phase
- Multiphase + damping
- Multiphase + No damping
Curvature Correction for One and Two Equation Models

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

Example: Prediction of the vortex free surface in an unbaffled mixing tank
Explicit Algebraic Reynolds Stress Model (EARSM)

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

**Left:** In-plane component of velocity vectors for Periodic flow in a square duct. EARSM (above) predicts secondary flow patterns with velocity ~2.4 percent of bulk velocity. SST (below) predicts no secondary flow

**Above and Right:** Flow in a rectangular, asymmetric diffuser. EARSM correctly predicts pressure coefficient on bottom surface
Turbulent Near Wall Treatment at Porous Medium Interface

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

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

Without Near Wall Treatment

With Near Wall Treatment
Summary and Conclusions – RANS

• Steady state RANS simulations will remain the dominant simulation method for turbulent flows for many years
  – While increasing use of LES and other scale resolving simulation methods for engineering applications is predicted, RANS will still maintain important advantages in some areas

• ANSYS strives to provide RANS models for use which are
  – Accurate
  – Robust
  – Y+ insensitive wall treatment
  – Interoperable with other physical models

• Developments in recent ANSYS releases extend the range of capabilities of the core turbulence models
  – Curvature correction, EARSM, free surface turbulence damping, porous media near wall treatment
Summary and Conclusions – Scale Resolving

• Industrial CFD:
  – A wide range of LES models required
  – Synthetic turbulence generator needed for ELES
  – A large number of validation cases needed to develop/optimize such models

• SRS/LES is a non-trivial subject:
  – Optimal model selection is problem dependent
  – Users need to understand models, application, grid and time step requirements
  – Optimal numerical settings important

• ANSYS Hybrid Models are a substantial advancement in providing SRS capability to industry on today's computers
Backup Slides
Scale Resolving Models in ANSYS CFD

- **Detached Eddy Simulation (DES) models**
  - DES-SST (Fluent, CFX), DES-Spalart-Allmaras and DES-Realizable k-ε (Fluent)

- **Large Eddy Simulation (LES)**
  - Smagorinsky-Lilly (+ dynamic) (Fluent, CFX)
  - WALE model (Fluent, CFX)
  - Algebraic Wall Modeled LES (WMLES) (Fluent, CFX)
  - Dynamic kinetic energy subgrid model (Fluent)

- **Embedded LES (ELES) model**
  - Combination of all RANS modes with all non-dynamic LES models (Fluent)
  - Zonal forcing model (CFX)

- **Synthetic turbulence generator**
  - Harmonic Turbulence Generator (HTG) (CFX)
  - Vortex method (Fluent)

- **Scale-Adaptive Simulation (SAS) models**
  - SAS-SST model (Fluent, CFX)
Small scales vs Large scales

• Largest eddy scales:
  
  – Assume $l$ is a characteristic size of a larger eddy.
  
  – Dimensional analysis is sufficient to estimate the order of large eddy supply rate for $k$ as: $k / \tau_{\text{turnover}}$.
  
  – The order of $\tau_{\text{turnover}}$ can be estimated as $l / k^{1/2}$ (i.e., $\tau_{\text{turnover}}$ is a time scale associated with the larger eddies).
  
  – Since $\varepsilon \sim k / \tau_{\text{turnover}}$, $\varepsilon \sim k^{3/2} / l$ or $l \sim k^{3/2} / \varepsilon$.

• Comparing $l$ with $\eta$:

$$\frac{l}{\eta} = \frac{l}{(\nu^3 / \varepsilon)^{1/4}} \approx \frac{l(k^{3/2} / l)^{1/4}}{\nu^{3/4}} \approx \operatorname{Re}_T^{3/4}$$

$$\frac{l}{\eta} \gg 1$$

  – where $\operatorname{Re}_T = k^{1/2} l / \nu$ (turbulent Reynolds number)
Prediction Methods

- Injection of energy
- Large-scale eddies
- Flux of energy
- Resolution length: $L$
- Dissipation of energy
- Dissipating eddies

Resolved

DNS

Modelled

LES

RANS

$\Delta_{DNS} = \frac{\Delta_{LES}}{Re^{3/4}}$
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:

   For a flat plate, Reynolds number ($\text{Re}_l = \frac{\rho VL}{\mu}$) gives $\text{Re}_l = 1.4 \times 10^6$

   (Recall from earlier slide, flow over a surface is turbulent when $\text{Re}_L > 5 \times 10^5$)
Example in Predicting Near-wall Cell Size [2]

- Begin with the definition of \( y^+ \) and rearrange:
  \[
  y^+ = \frac{\rho U_\tau}{\mu} \quad \Longleftrightarrow \quad y = \frac{y^+ \mu}{U_\tau \rho}
  \]
  
- The target \( y^+ \) value and fluid properties are known, so we need \( U_\tau \), which is defined as:
  \[
  U_\tau = \sqrt{\frac{\tau_w}{\rho}}
  \]
  
- The wall shear stress, \( \tau_w \), can be found from the skin friction coefficient, \( C_f \):
  \[
  \tau_w = \frac{1}{2} C_f \rho U^2_\infty
  \]
  
- A literature search suggests a formula for the skin friction on a plate\(^1\) thus:
  \[
  C_f = 0.058 \text{Re}^{-0.2}
  \]

- \( \text{Re} \) is known, so use the definitions to calculate the first cell height
  
  \[
  C_f = 0.058 \text{Re}_{\infty}^{-0.2} = 0.0034
  \]
  \[
  \tau_w = \frac{1}{2} C_f \rho U^2_\infty = 0.83 \text{kg/(m} \cdot \text{s}^2)\]
  \[
  U_\tau = \sqrt{\frac{\tau_w}{\rho}} = 0.82 \text{ m/s}
  \]

- 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.

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