What is turbulence?

- Due to the extremely complex nature of turbulence and its incomplete understanding, there is not a single accepted definition of turbulence.
  - Leonardo da Vinci was so intrigued by turbulence that he depicted it in many of his sketches (see previous slide). While observing the flow of water, he gave one of the very first definitions of turbulence (if not the first one),

> *"...the smallest eddies are almost numberless, and large things are rotated only by large eddies and not by small ones and small things are turned by small eddies and large"*

  - Richardson [1] in 1922 stated that,

> *"Big whorls have little whorls,*
> *which feed on their velocity,*
> *and little whorls have lesser whorls,*
> *and so on to viscosity"*

[1] L. F. Richardson. "Weather Prediction by Numerical Process". Cambridge University Press, 1922.

What is turbulence?

- Due to the extremely complex nature of turbulence and its incomplete understanding, there is not a single accepted definition of turbulence.
  - T. von Karman [1] who is known for his studies about Fluid Dynamics, quotes G. I. Taylor with the following definition of turbulence in 1937,

> *"Turbulence is an irregular motion which in general makes its appearance in fluids, gaseous or liquid, when they flow past solid surfaces or even when neighboring streams same fluid past or over one another."*

  - J.O. Heinz [2] offers yet another definition for turbulence in 1959,

> *"Turbulent fluid motion is an irregular condition of the flow in which quantities show a random variation with time and space coordinates, so that statistically distinct average values can be discerned."*

[1] T. Von Karman. "Some remarks on the statistical theory of turbulence". Proc. 5th Int. Congr. Appl. Mech, Cambridge, MA, 347, 1938.
[2] J. O. Hinze. "Turbulence". McGraw-Hill, New York, 1959.

What is turbulence?

- Due to the extremely complex nature of turbulence and its incomplete understanding, there is not a single accepted definition of turbulence.
  - A more modern and highly specific definition of turbulence is given by G. T. Chapman and M. Tobak [1],

> *"Turbulence is any chaotic solution to the 3D Navier–Stokes equations that is sensitive to initial data and which occurs as a result of successive instabilities of laminar flows as a bifurcation parameter is increased through a succession of values."*

  - S. Rodriguez [2], gives an even more modern definition linked to the use of approximations to deliver solutions,

> *"Turbulent flows is the dynamic superposition of an extremely large number of eddies with random (irregular) but continuous spectrum of sizes and velocities that are interspersed with small, discrete pockets of laminar flow (as a result of the Kolmogorov eddies that decayed, as well as in the viscous laminar sublayer and in the intermittent boundary). In this sense, turbulent flows are intractable in its fullest manifestation; this is where good, engineering common sense and approximations can deliver reasonable solutions, albeit approximate."*

[1] G. T. Chapman and M. Tobak. "Observations, Theoretical Ideas, and Modeling of Turbulent Flows — Past, Present and Future, in Theoretical Approaches to Turbulence". Dwoyeret al.(eds), Springer-Verlag, New York, pp. 19–49, 1985.
[2] S. Rodriguez. "Applied Computational Fluid Dynamics and Turbulence modeling". Springer, 2019.

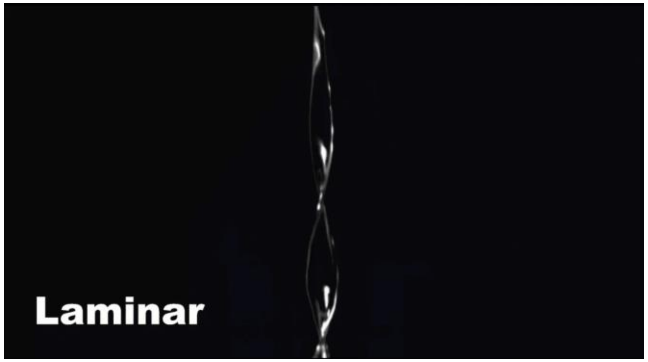
What is turbulence?

- Due to its complexity, a definition does not work properly for turbulence, instead of it, it’s better to explain its characteristics.
- Tennekes and Lumley [1] in their book called “*A First Course in Turbulence*”, list the characteristics of turbulence:
    - Irregularity.
    - Diffusivity.
    - Large Reynolds numbers.
    - Three-Dimensional vorticity fluctuations.
    - Dissipation.
    - Continuum.
    - Feature of a flow, not fluid.

[1] H. Tennekes and J. L. Lumley. “A first course in turbulence”. MIT Press, 1972.

Turbulent flows have the following characteristics

- In summary:
  - One characteristic of turbulent flows is their **irregularity** (or randomness). A fully deterministic approach to characterize turbulent flows is very difficult. Turbulent flows are usually described statically. Turbulent flows are always chaotic. But not all chaotic flows are turbulent. Magma flowing can be chaotic but not necessarily turbulent.
  - The **diffusivity** of turbulence causes rapid mixing and increased rates of momentum, heat, and mass transfer. A flow that looks random but does not exhibit the spreading of velocity fluctuations through the surrounding fluid is not turbulent.
  - Turbulent flows are **dissipative**. Kinetic energy gets converted into heat due to viscous shear stresses. Turbulent flows die out quickly when no energy is supplied.
  - Turbulent flows always occur at **high Reynolds numbers**. They are caused by a complex interaction between the viscous forces and convection.
  - Turbulent flows are **rotational**, that is, they have non-zero vorticity. Mechanisms such as the stretching of three-dimensional vortices play a key role in turbulence.
  - Turbulence is a **continuum** phenomenon. Even the smallest eddies are significantly larger than the molecular scales.
  - Turbulence is a **feature of fluid flow** and is not a property of the flow. A liquid or a gas at high Reynolds number will exhibit the same dynamics.

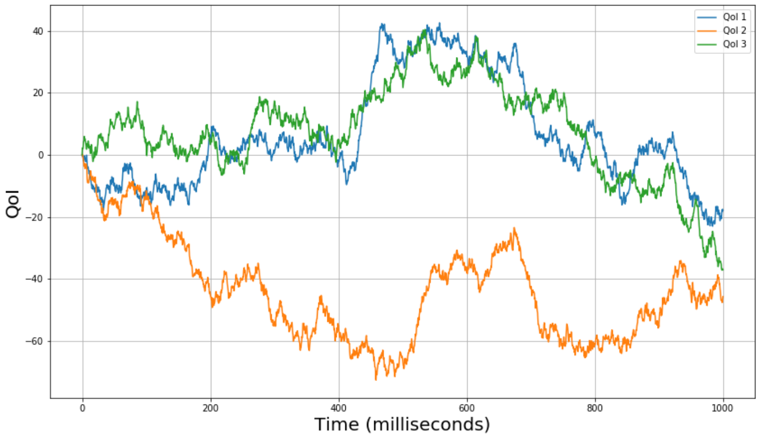


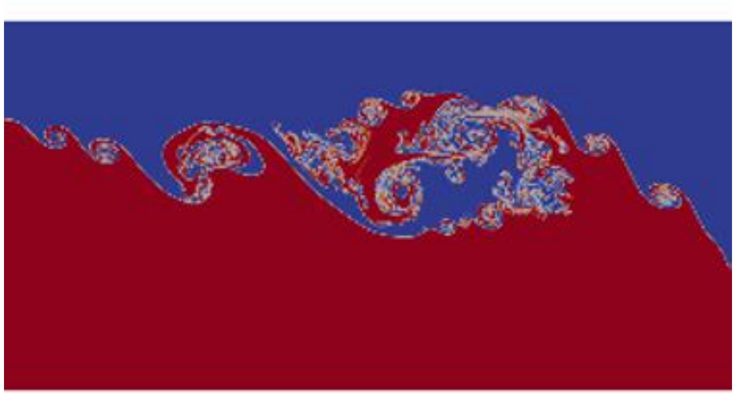
Video credit: https://www.youtube.com/watch?v=9dz7fUJBFOk
This video has been edited to fit this presentation.

Reference: H. Tennekes and J. L. Lumley. “A first course in turbulence”. MIT Press, 1972.

… so, after all, what is turbulence?

- Turbulent flows share all the previous characteristics.
- In addition, let us state the following:
    - Turbulence is an unsteady, aperiodic motion in which all three velocity components fluctuate in space and time.
    - Every transported quantity shows similar fluctuations (pressure, temperature, species, concentration, and so on)
    - Turbulent flows contains a wide range of eddy sizes (scales):
        - Large eddies derives their energy from the mean flow. The size and velocity of large eddies are on the order of the mean flow.
        - Large eddies are unstable and they break-up into smaller eddies.
        - The smallest eddies convert kinetic energy into thermal energy via viscous dissipation.
        - The behavior of small eddies is more universal in nature.





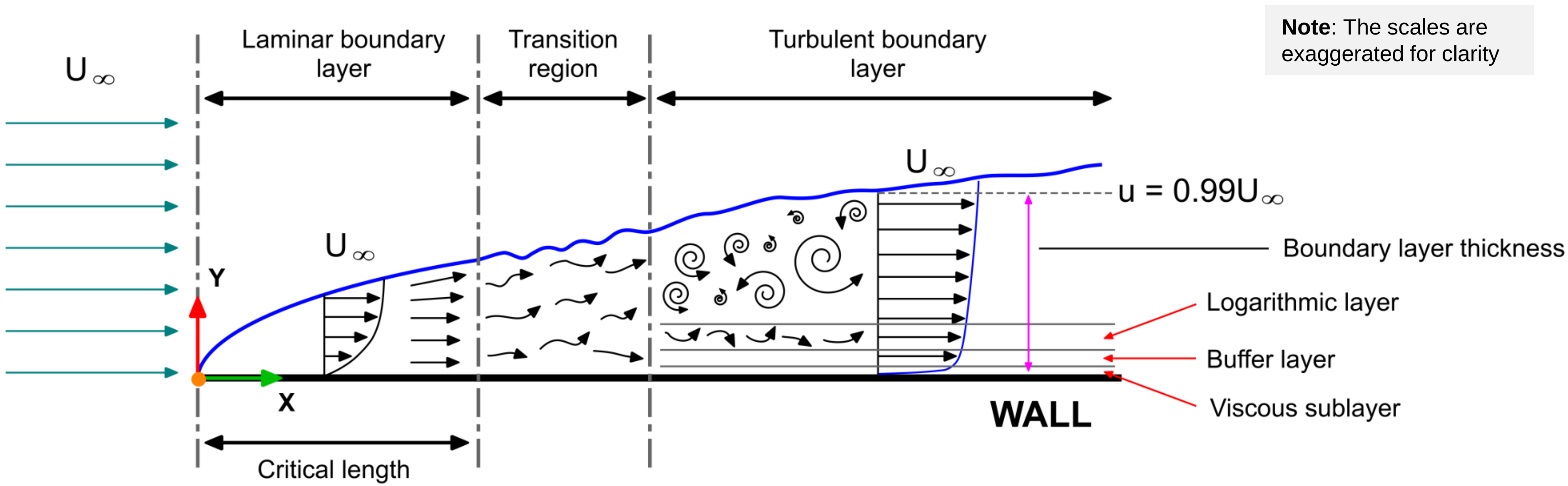
Turbulence modeling in engineering

- Most natural and engineering flows are turbulent, hence the necessity of modeling turbulence.
- The goal of turbulence modeling is to develop equations that predict the time averaged velocity, pressure, temperature fields without calculating the complete turbulent flow pattern as a function of time.
- Turbulence can be wall bounded or free shear. Depending on what you want to simulate, you will need to choose an appropriate turbulence model.
- There is no universal turbulence model, hence you need to know the capabilities and limitations of the turbulence models.
- Due to the multi-scale and unsteady nature of turbulence, modeling it is not an easy task.
- Simulating turbulent flows in any general CFD solver (*e.g.*, OpenFOAM®, SU2, Fluent, CFX, Star-CCM+) requires selecting a turbulence model, providing initial conditions and boundary conditions for the closure equations of the turbulent model, selecting a near-wall modeling, and choosing runtime parameters and numerics.

Why turbulent flows are challenging?

- Unsteady aperiodic motion.
- All fluid properties and transported quantities exhibit random spatial and temporal variations.
- They are intrinsically three-dimensional due to vortex stretching.
- Strong dependence from initial conditions.
- Contains a wide range of scales (eddies).
- Therefore, in order to accurately model/resolve turbulent flows, the simulations must be three-dimensional, time-accurate, and with fine enough meshes such that all spatial scales are properly captured.
- Additional physics that makes turbulence modeling even harder:
  - Buoyancy, compressibility effects, heat transfer, multiphase flows, transition to turbulence, surface finish, combustion, and so on.

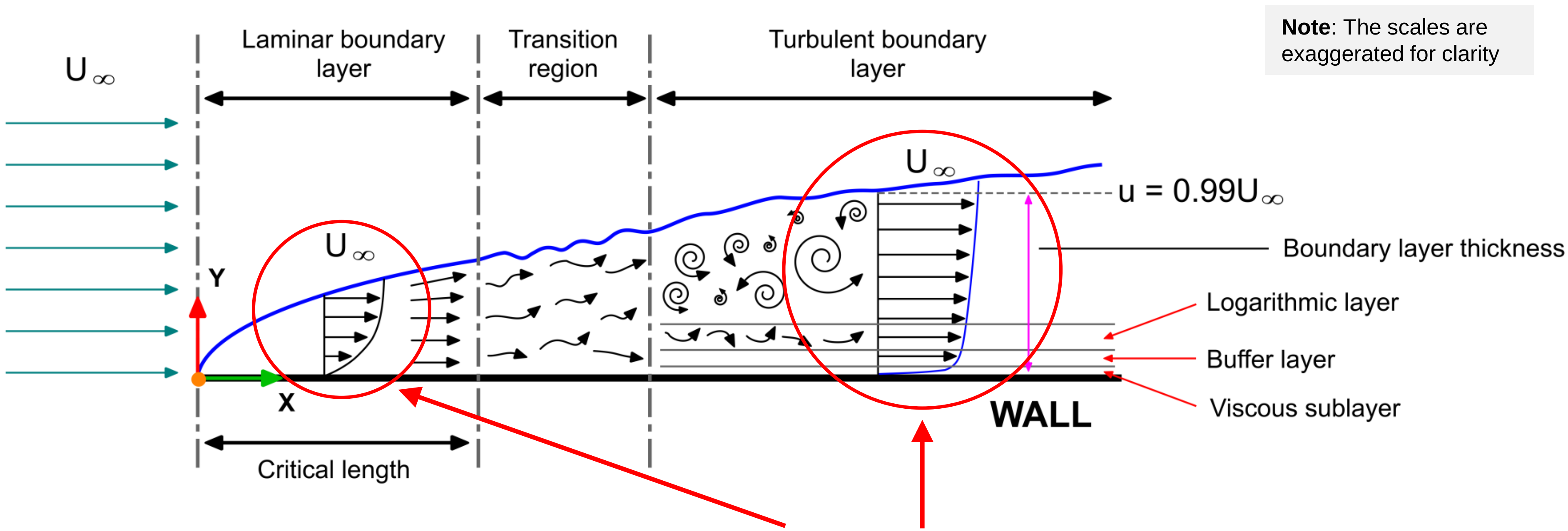
Turbulence modeling – Boundary layer



Boundary layer (Laminar-Transitional-Turbulent flow)

- Near walls, in the boundary layer, the velocity changes rapidly.
- In this case, a laminar boundary layer starts to form at the leading edge.
- As the flow proceeds further downstream, large shear stresses and velocity gradient develop within the boundary layer. At one point the flow becomes turbulent.
- The turbulent motion increases the mixing and the boundary layer mixing.
- What is happening in the transition region is not well understood. The flow can become laminar again or can become turbulent.
- The velocity profiles in the laminar and turbulent regions are different.

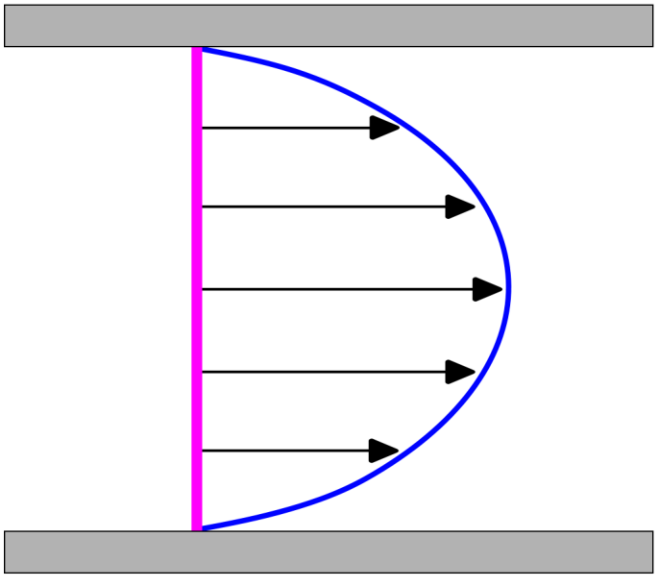
Turbulence modeling – Boundary layer



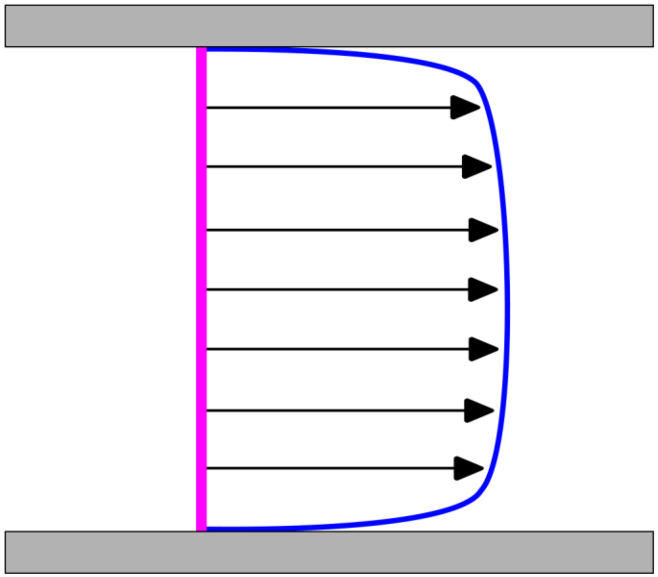
Actual profile – Physical velocity profile

- Near walls, in the boundary layer, the velocity changes rapidly.
- In turbulence modeling in CFD, the most important zones are the viscous sublayer and the log-law layer.
- The buffer layer is the transition layer which we try to avoid as much as possible.
- Turbulence modeling in CFD requires different considerations depending on whether you solve the viscous sublayer, model the log-law layer, or solve the whole boundary layer (including the buffer zone).

Turbulence modeling – Fluctuations of transported quantities



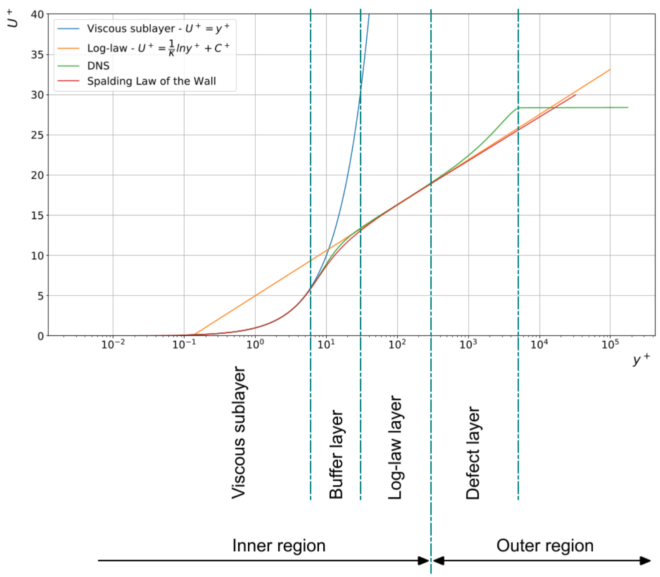
Laminar flow in a pipe



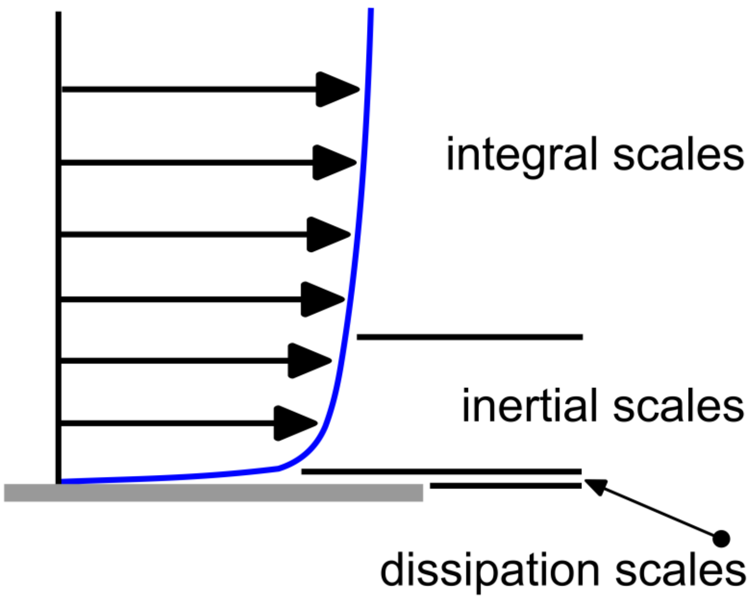
Turbulent flow in a pipe

- Turbulence has a direct effect on the velocity profiles and mixing of transported quantities.
- In the laminar case, the velocity gradient close to the walls is small (therefore the shear stresses are lower).
- The turbulent case shows two regions. One thin region close to the walls with very large velocity gradients (hence large shear stresses), and a region far from the wall where the velocity profile is nearly uniform.
- In the illustration, the velocity profile of the turbulent case has been averaged (in reality, there are fluctuations).

Non-dimensional profile against physical velocity profile



Non-dimensional profile

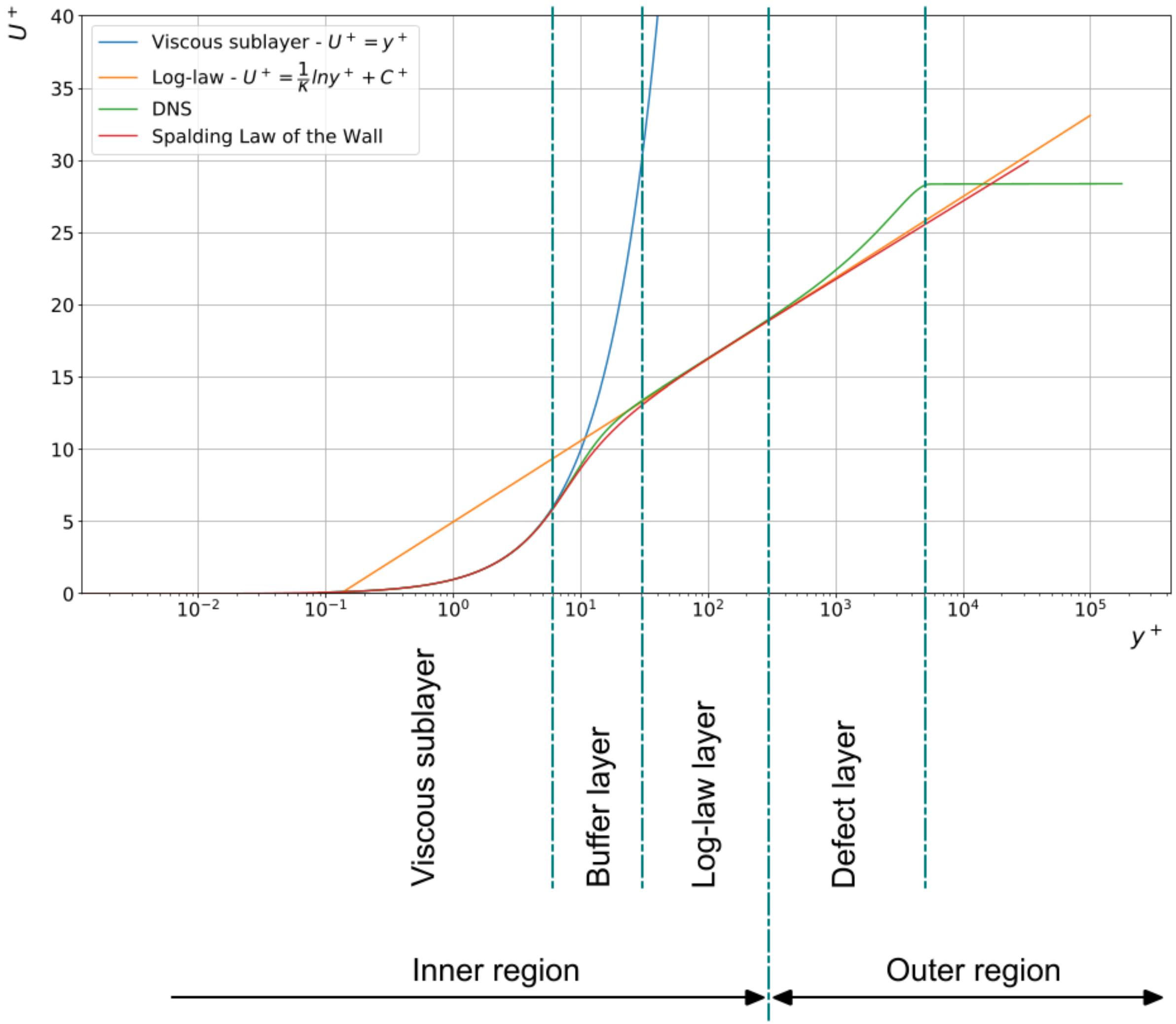


Physical velocity profile – Actual profile

- The use of the non-dimensional velocity u^+ and non-dimensional distance from the wall y^+, results in a predictable boundary layer profile for a wide range of flows.
- Under standard working conditions this profile is the same, however, under non-equilibrium conditions (production and dissipation of turbulent kinetic energy not balanced), rough walls, porous media, buoyancy, viscous heating, strong pressure gradients, and so on, the profile might be different.
- While the non-dimensional velocity profile is the same for many flows, the physical velocity profile is different.

Brief overview of turbulence modeling in CFD

Turbulence near the wall – Relations according to y^+ value



$$30 < y^+ < 300$$

$$u^+ = \frac{1}{\kappa} \ln y^+ + C^+$$

$$\kappa \approx 0.41 \quad C^+ \approx 5.0$$

$$y^+ < 6$$

$$u^+ = y^+$$

$$6 < y^+ < 30$$

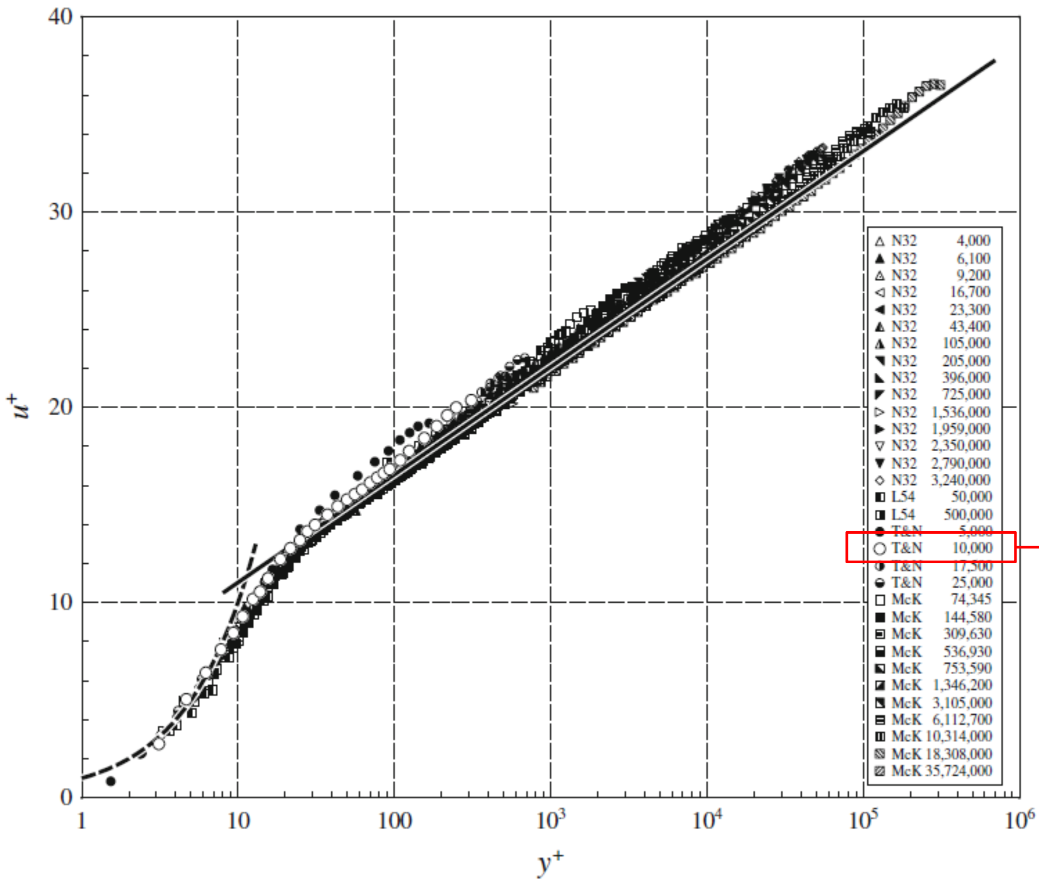
$$u^+ \neq y^+$$

$$u^+ \neq \frac{1}{\kappa} \ln y^+ + C^+$$

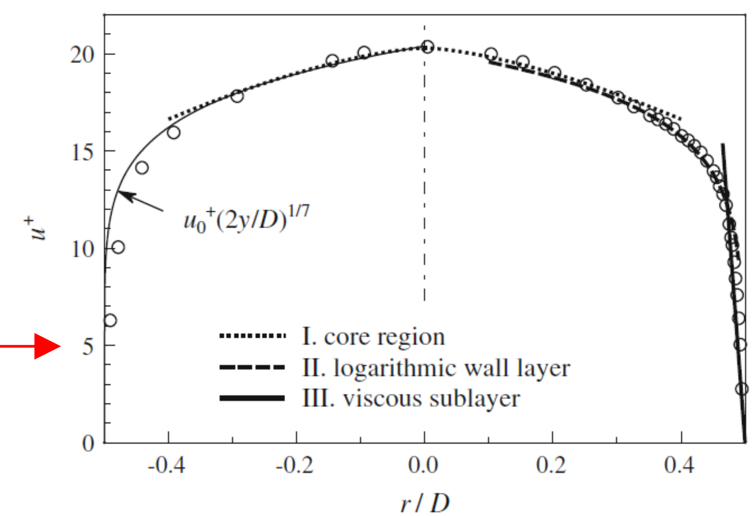
Note: the range of y^+ values might change from reference to reference but roughly speaking they are all close to these values.

Brief overview of turbulence modeling in CFD

Turbulence near the wall – Experimental data



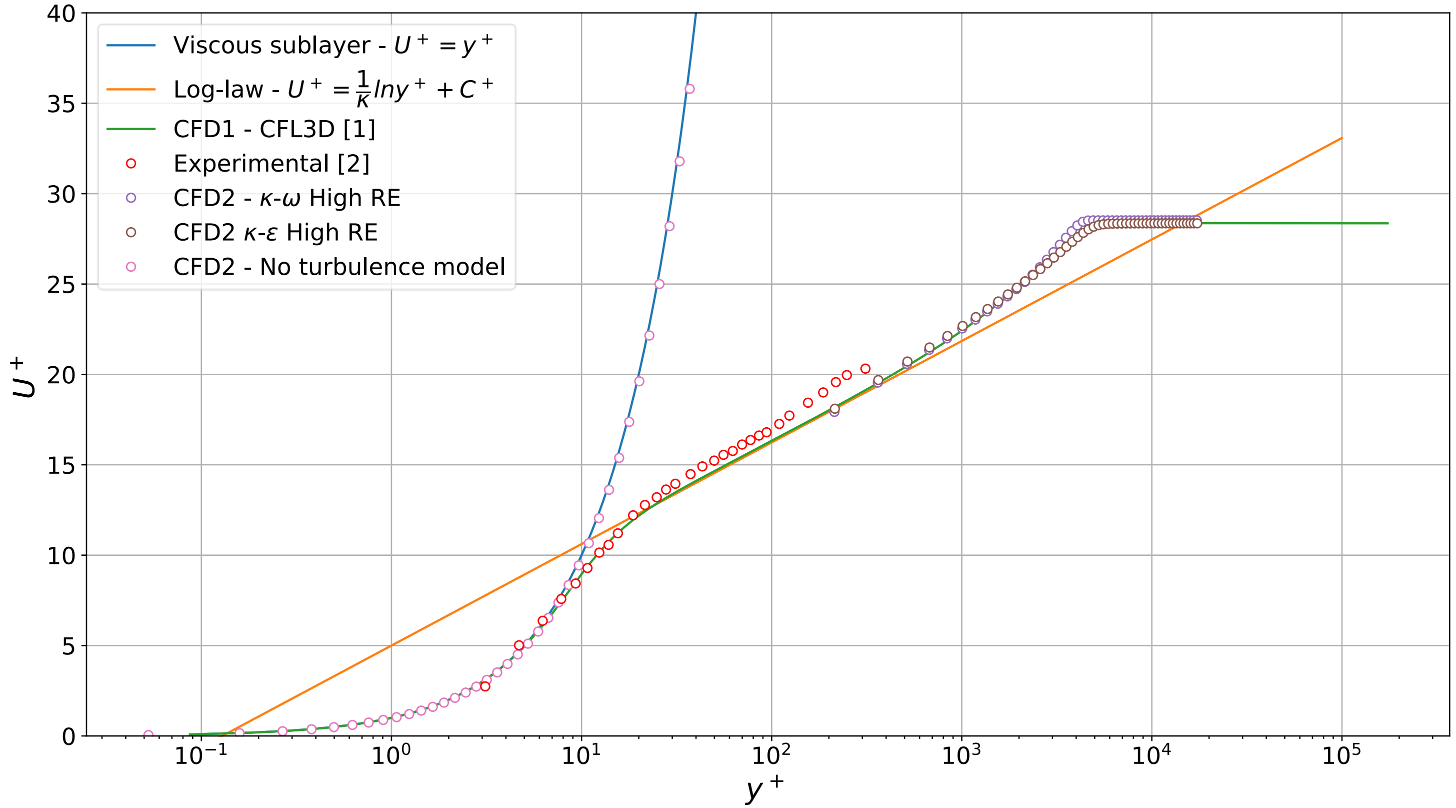
Dimensionless mean velocity profile u⁺ as a function of the dimensionless wall distance y⁺ for turbulent pipe flow with Reynolds numbers between 4000 and 3600000.



The dimensionless mean velocity profile u^+ of a turbulent flow in a pipe at Re = 10000 as a function of the distance r from the centerline of a pipe with diameter D, compared to experimental data.

References:
F. Nieuwstadt, B. Boersma, J. Westerweel. Turbulence. Introduction to Theory and Applications of Turbulent Flows. Springer, 2016.

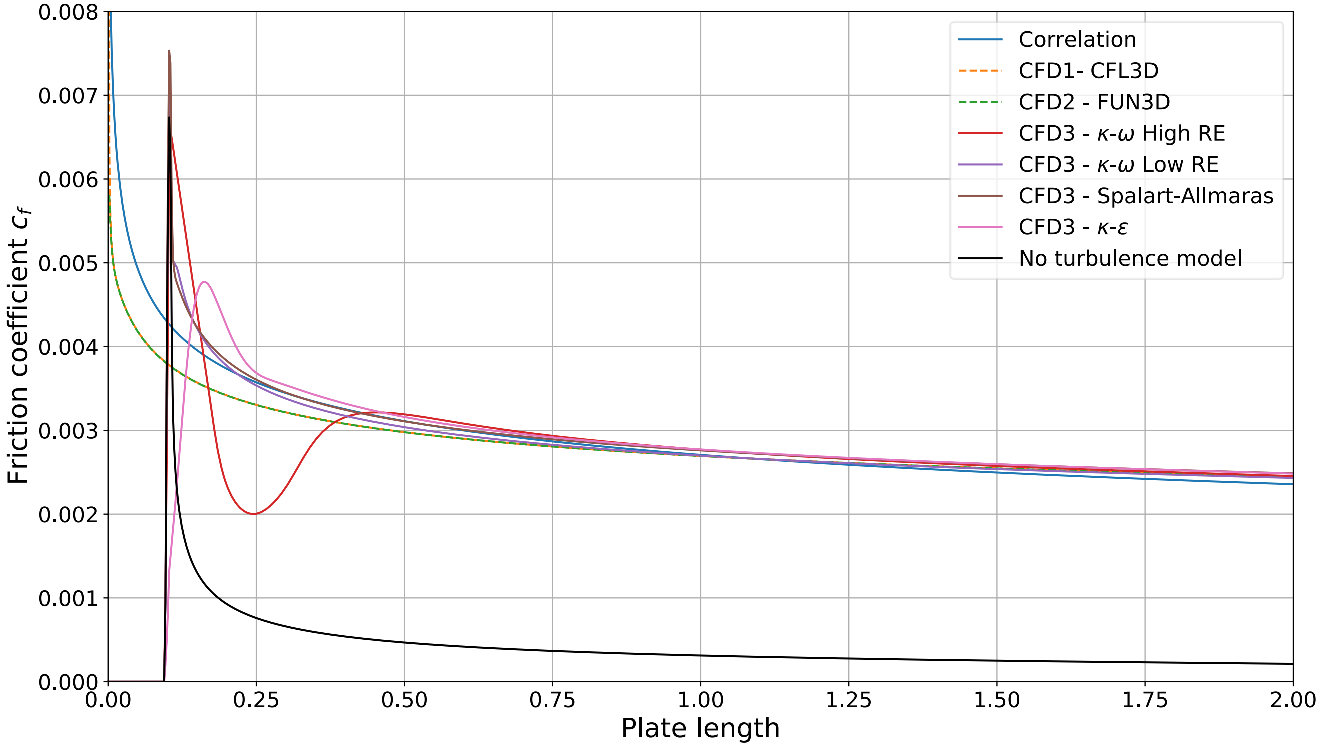
Turbulence near the wall – Relations according to y+ value



- Plot of the non-dimensional velocity profile.
- Notice that all cases plotted correspond to different physics and Reynolds numbers.

References:
[1] https://turbmodels.larc.nasa.gov
[2] J. M. J. den Toonder and F. T. M. Nieuwstadt. Reynolds number effects in a turbulent pipe flow for low to moderate Re. Physics of Fluids 9, 3398 (1997).

Turbulence near the wall – Relations according to y^+ value



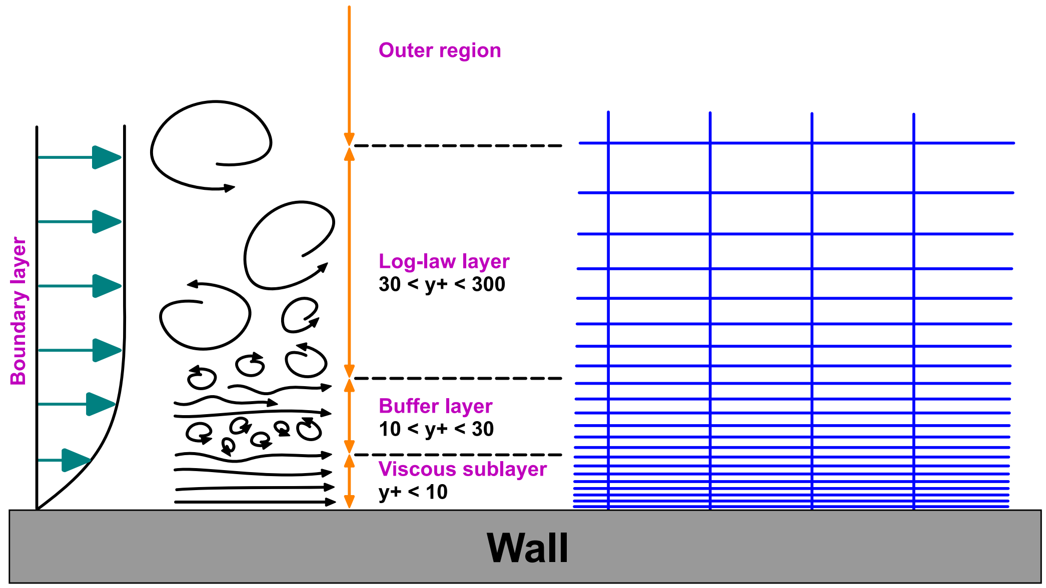
- Plot of friction coefficient in function of length for the flat plate case [1].
- Notice that the case where we did not use turbulence model (DNS simulation), it highly under predicts the friction coefficient value.
- The importance of using a turbulence model.

References:
https://turbmodels.larc.nasa.gov/flatplate.html

Brief overview of turbulence modeling in CFD

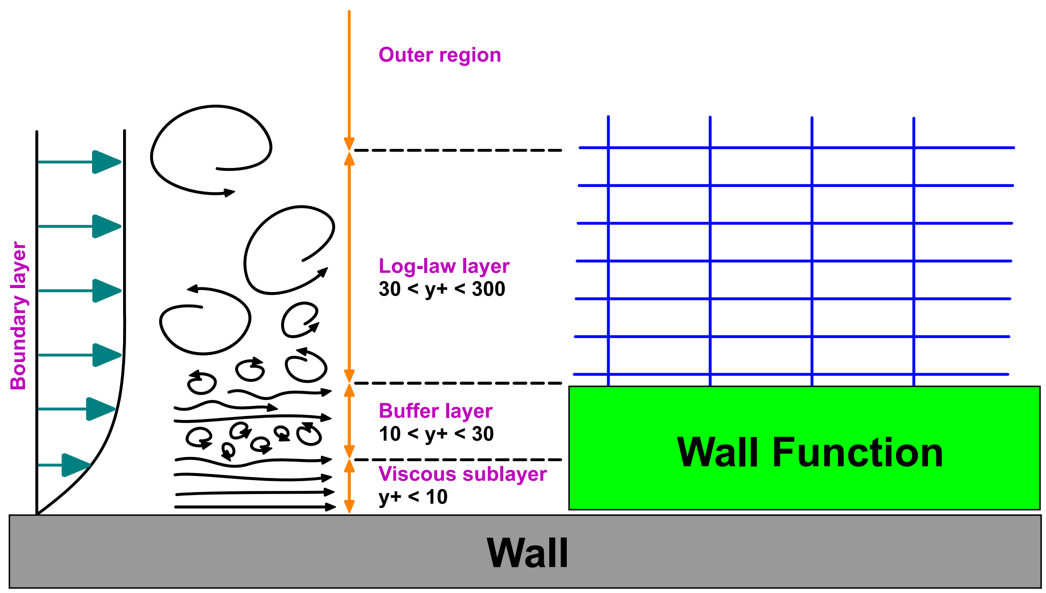
Near-wall treatment and wall functions

- When dealing with wall turbulence, we need to choose a near-wall treatment.
- If you want to resolve the boundary layer up to the viscous sub-layer you need very fine meshes close to the wall.
- In terms of y^+, you need to cluster at least 6 to 10 layers at $y^+ < 10$. You need to properly resolve the velocity profile.
- But for good accuracy, usually you will use 15 to 30 layers.
- This is the most accurate approach, but it is computationally expensive.



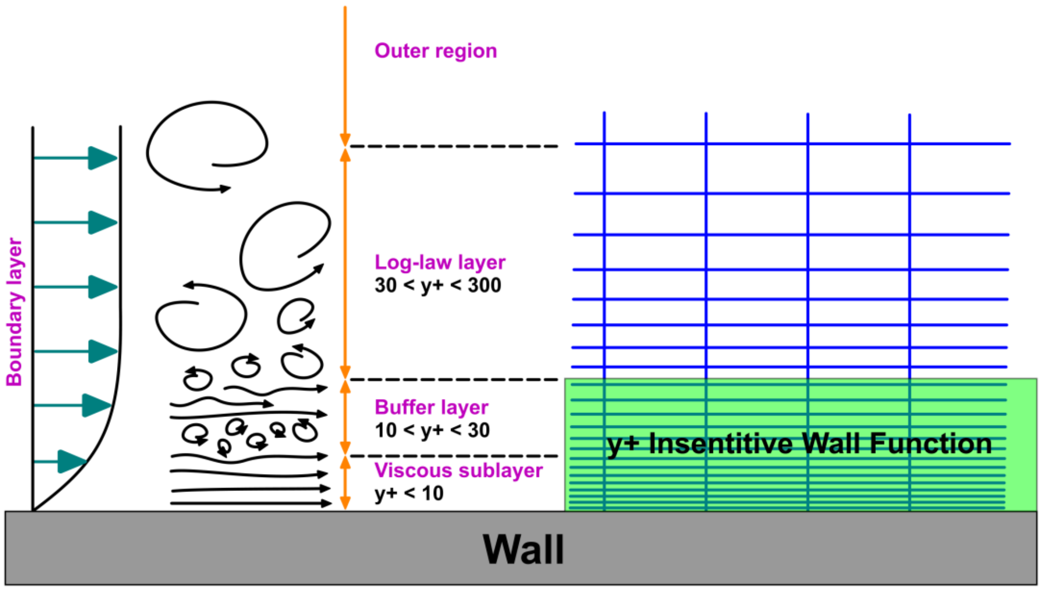
Near-wall treatment and wall functions

- When dealing with wall turbulence, we need to choose a near-wall treatment.
- If you are not interested in resolving the boundary layer up to the viscous sub-layer, you can use wall functions.
- In terms of y^+, wall functions will model everything below $y^+ < 30$ or the target y^+ value.
- This approach use coarser meshes, but you should be aware of the limitations of the wall functions.
- You will need to cluster at least 5 to 10 layers to resolve the profiles (U and k).



Near-wall treatment and wall functions

- When dealing with wall turbulence, we need to choose a near-wall treatment.
- You can also use the y^+ insensitive wall treatment (sometimes known as continuous wall functions or scalable wall functions). This kind of wall functions are valid in the whole boundary layer.
- In terms of y^+, you can use this approach for values between $1 \leq y^+ \leq 300$.
- This approach is very flexible as it is independent of the y^+ value, but is not available in all turbulence models
- You also should be aware of the limitations this wall treatment method.

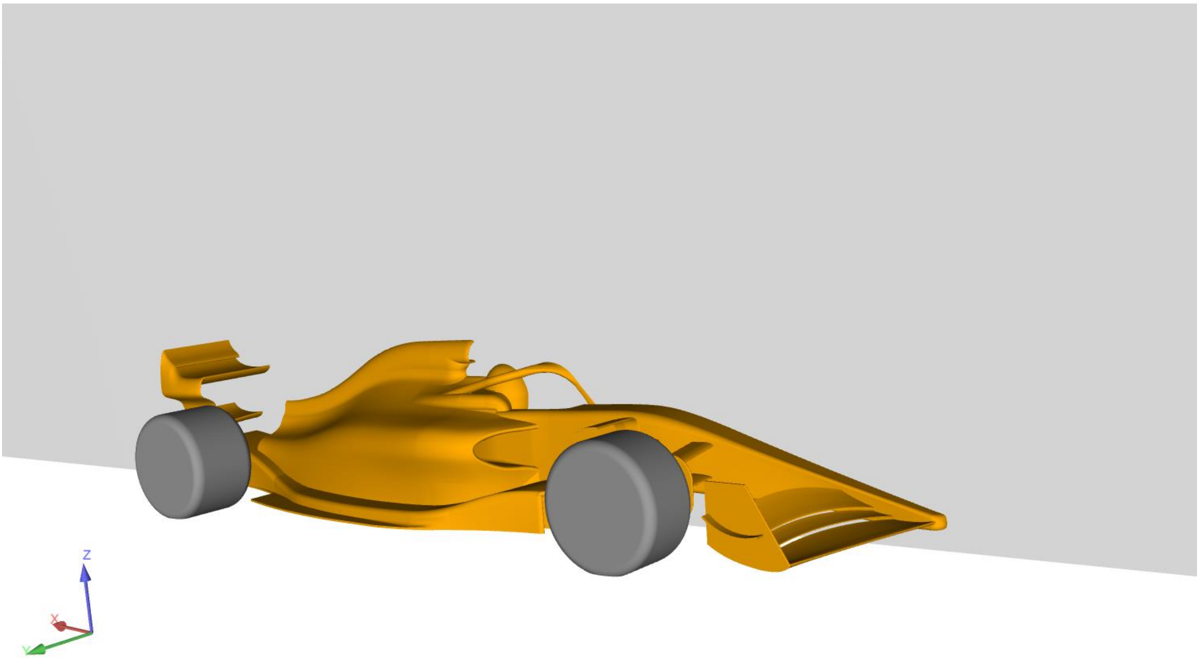


Near-wall treatment and wall functions

- Generally speaking, wall functions is the approach to use if you are more interested in the mixing in the outer region, rather than the forces on the wall.
- If accurate prediction of forces or heat transfer on the walls are key to your simulation (aerodynamic drag, turbomachinery blade performance, heat transfer) you might not want to use wall functions.
- The wall function approach is also known as high-RE (HRN).
- The approach where you do not use wall functions is known as low-RE (LRN).
- Wall functions should be avoided if $10 < y^+ < 30$. This is the transition region, and wall function do not perform very well here (nobody knows what is going on in this region).
- The low-RE approach is computational expensive as it requires clustering a lot cells near the walls.
- To get good results with LRF, you will need to cluster at least 10 layers for $y^+ < 6$. But values up to $y^+ < 10$ are acceptable. It is primordial solving the velocity profile.
- If you do not have any restrictions in the near-wall treatment, it is recommended to use wall functions.

Brief overview of turbulence modeling in CFD

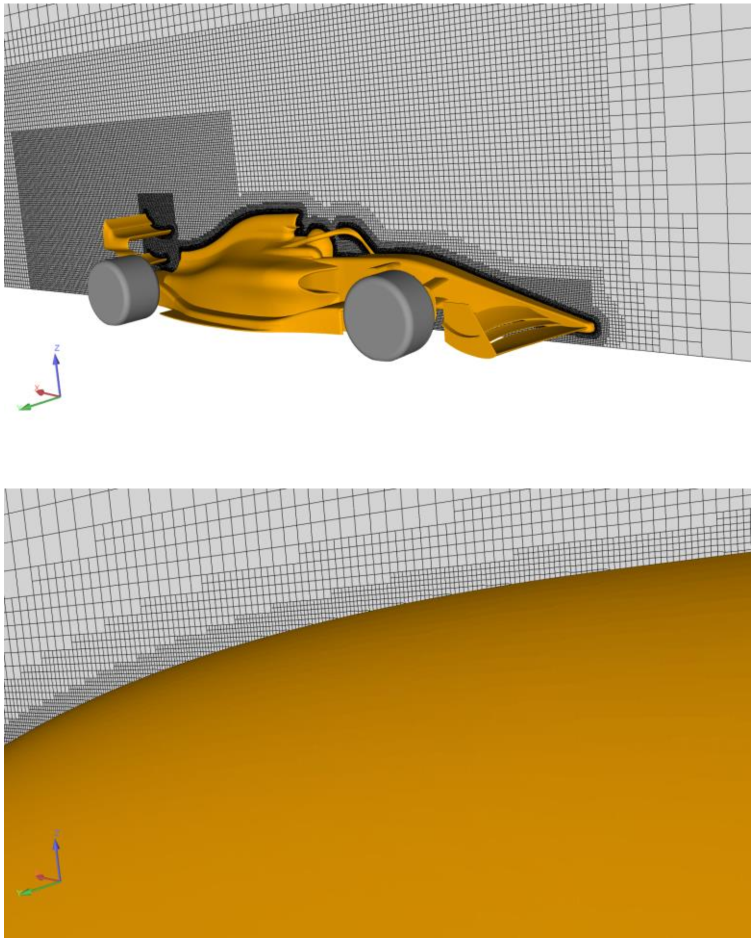
Influence of near-wall treatment in the cell count



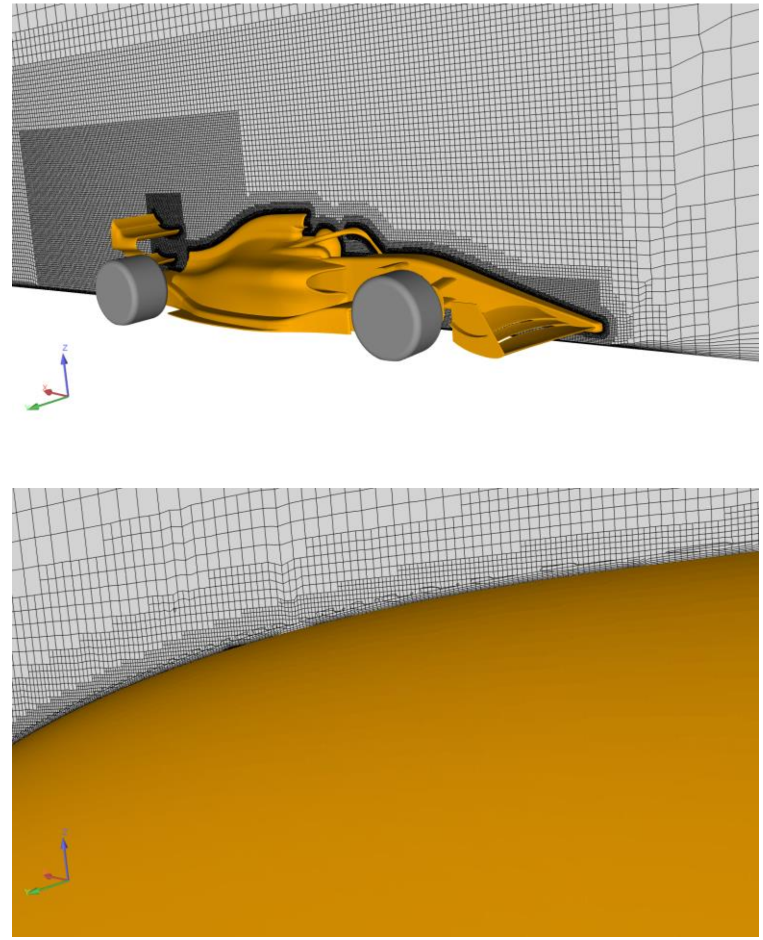
	Mesh 1	Mesh 2
Number of cells	57 853 037	111 137 673

- Can you guest the difference between the meshes?

Influence of near-wall treatment in the cell count



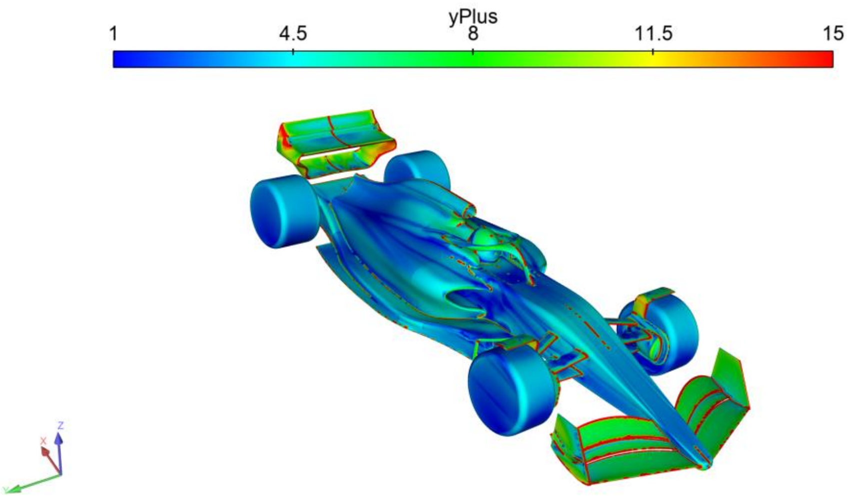
Average y^+ approximately 60



Average y^+ approximately 7

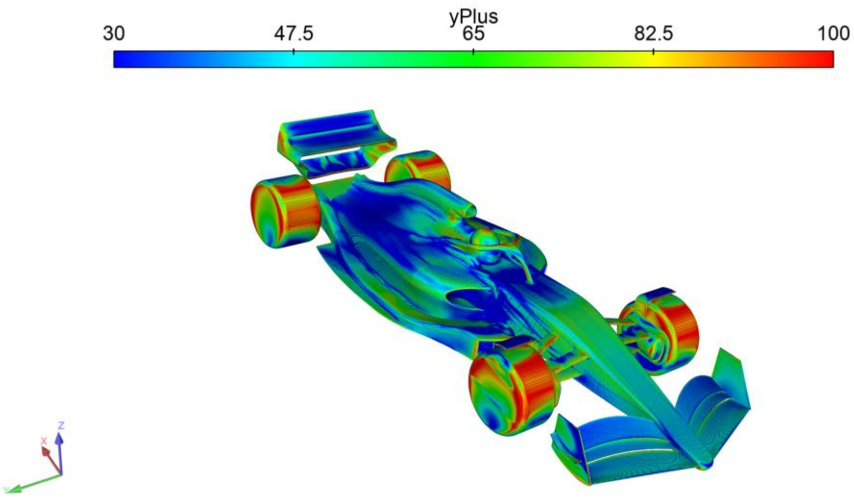
- By only adding the inflation layer to resolve the boundary layer we almost doubled the number of cells.

Influence of near-wall treatment in the cell count



Wall resolving approach (Low-RE)

	Average y+
FWD. Wing	14
RWD. Wing	12
Body	4

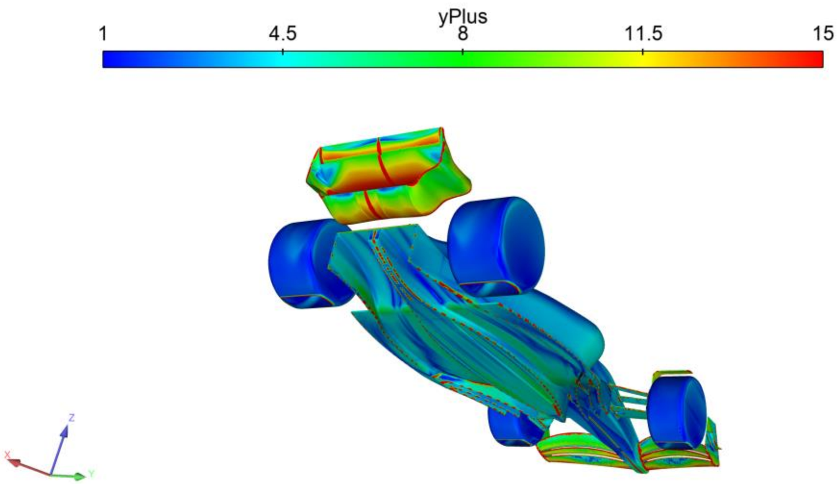


Wall modeling approach (High-RE)

	Average y+
FWD. Wing	56
RWD. Wing	62
Body	46

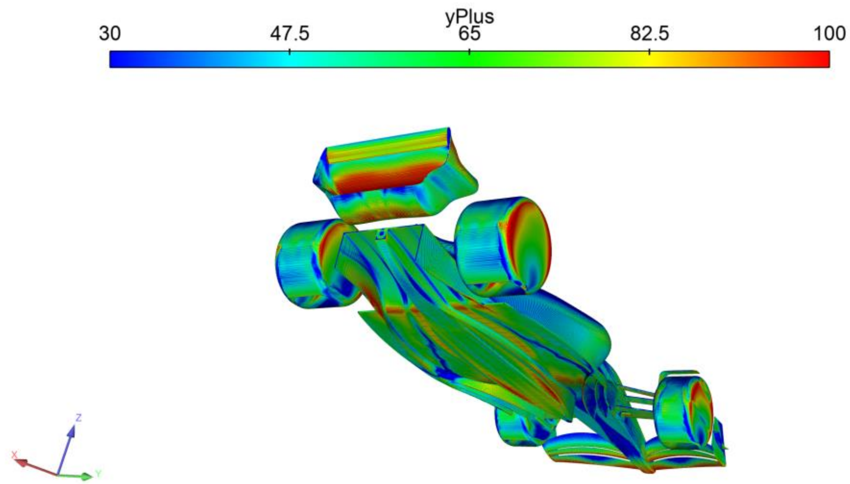
Brief overview of turbulence modeling in CFD

Influence of near-wall treatment in the cell count



Wall resolving approach (Low-RE)

	Average y+
FWD. Wing	14
RWD. Wing	12
Body	4

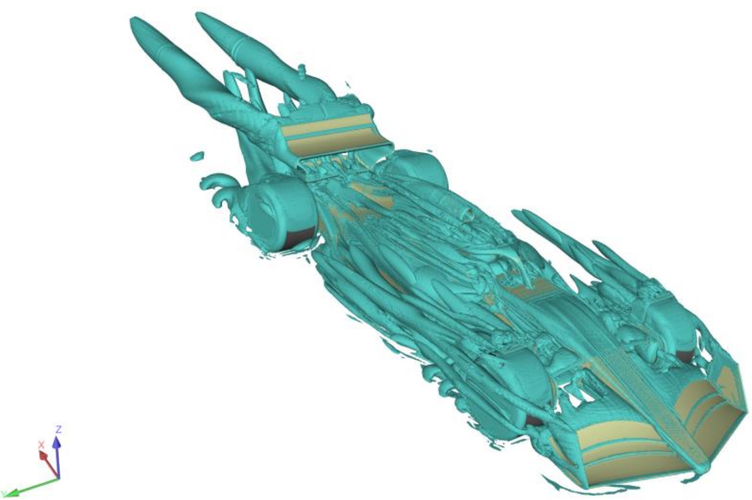


Wall modeling approach (High-RE)

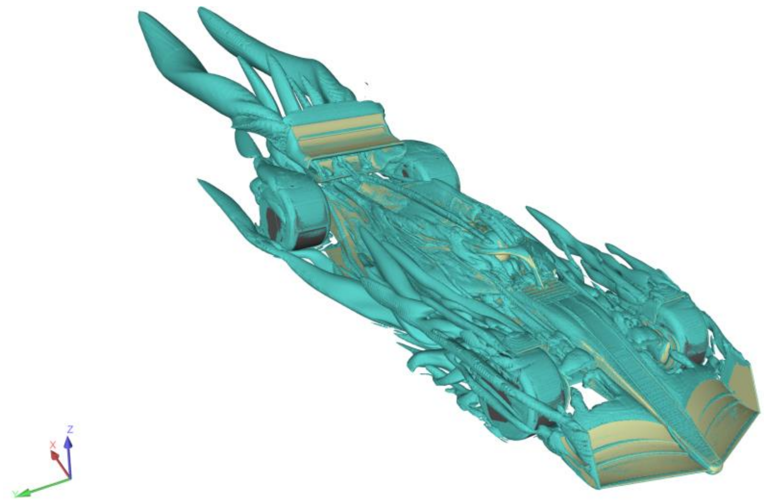
	Average y+
FWD. Wing	56
RWD. Wing	62
Body	46

Brief overview of turbulence modeling in CFD

Influence of near-wall treatment in the cell count



Wall resolving approach (Low-RE)

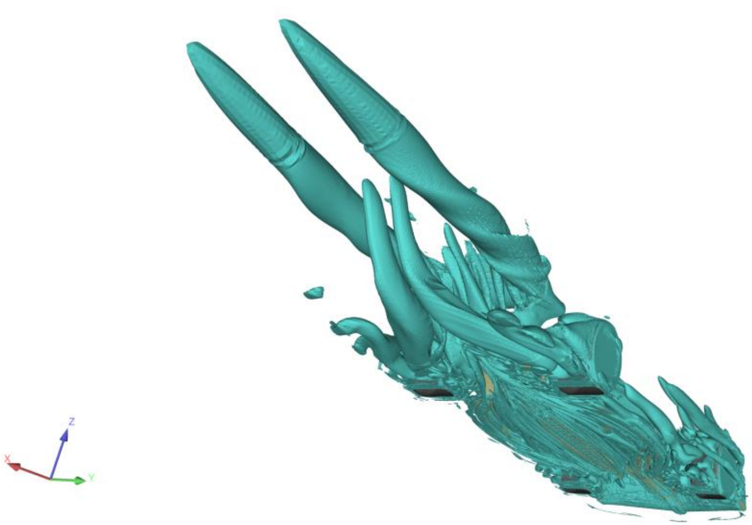


Wall modeling approach (High-RE)

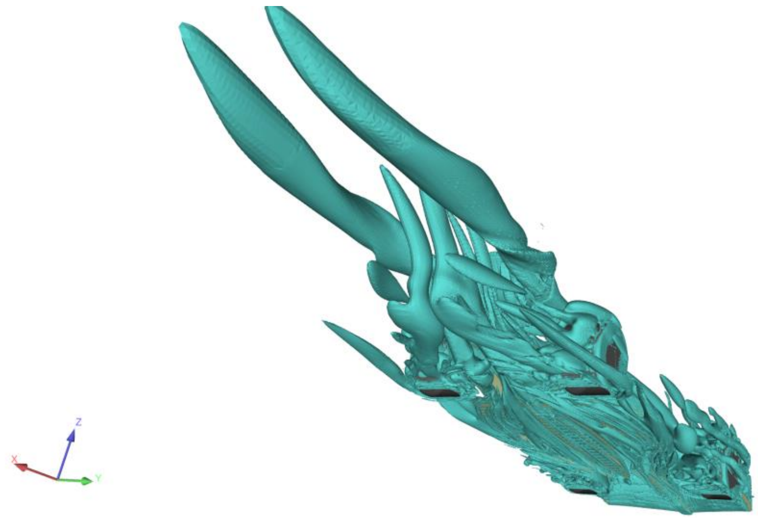
- Qualitative post-processing.
- The vortical structures are visualized using the Q-criterion.
- By the way, if you switch off the turbulence model, it is likely probably that your results will be garbage (unless you have an extremely fine mesh that resolves all scales).

Brief overview of turbulence modeling in CFD

Influence of near-wall treatment in the cell count



Wall resolving approach (Low-RE)



Wall modeling approach (High-RE)

- Qualitative post-processing.
- The vortical structures are visualized using the Q-criterion.
- By the way, if you switch off the turbulence model, it is likely probably that your results will be garbage (unless you have an extremely fine mesh that resolves all scales).

Brief overview of turbulence modeling in CFD

Turbulence modeling – Starting equations

NSE

$$
\begin{cases}
\dfrac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0 \\
\dfrac{\partial (\rho \mathbf{u})}{\partial t} + \nabla \cdot (\rho \mathbf{u}\mathbf{u}) = -\nabla p + \nabla \cdot \boldsymbol{\tau} + \mathbf{S}_u \\
\dfrac{\partial (\rho e_t)}{\partial t} + \nabla \cdot (\rho e_t \mathbf{u}) = \nabla \cdot q - \nabla \cdot (p\mathbf{u}) + \boldsymbol{\tau}\mathbf{:}\nabla \mathbf{u} + \mathbf{S}_e
\end{cases}
$$

$$+$$

Additional equations to close the system (thermodynamic variables)

Additionally, relationships to relate the transport properties

Additional closure equations for the turbulence models

- Turbulence models equations cannot be derived from fundamental principles.
- All turbulence models contain some sort of empiricism.
- Some calibration to observed physical solutions is contained in the turbulence models.
- Also, some intelligent guessing is used.
- A lot of uncertainty is involved!

Brief overview of turbulence modeling in CFD

Turbulence modeling – Starting equations

- Let us write down the governing equations for an incompressible flow.
- When conducting DNS simulations (no turbulence models involved), this is our starting point,

$$\nabla \cdot (\mathbf{u}) = 0$$

$$\frac{\partial \mathbf{u}}{\partial t} + \nabla \cdot (\mathbf{u}\mathbf{u}) = \frac{-\nabla p}{\rho} + \nu \nabla^2 \mathbf{u}$$

- When using RANS turbulence models, these are the governing equations (for incompressible flows),

If we retain this term we talk about URANS equations
and if we drop it we talk about RANS equations

Reynolds stress tensor
This term requires modeling

$$\nabla \cdot (\bar{\mathbf{u}}) = 0$$

$$\frac{\partial \bar{\mathbf{u}}}{\partial t} + \nabla \cdot (\bar{\mathbf{u}}\bar{\mathbf{u}}) = \frac{-\nabla \bar{p}}{\rho} + \nu \nabla^2 \bar{\mathbf{u}} - \frac{1}{\rho} \nabla \cdot \tau^R$$

Brief overview of turbulence modeling in CFD

Turbulence modeling – Starting equations

- The difference between the DNS equations and RANS equations are the overbar over the primitive variables and the appearance of the Reynolds stress tensor.
- The overbar over the primitive variables in the RANS equations means that the quantities have been averaged (time average, spatial average or ensemble average).
- In the RANS equations, the Reynolds stress tensor requires modeling. Therefore, we need to define how to model this term and introduce closure equations (turbulence modeling).
- There are many turbulence models available, and none of them is universal.
- Therefore, it is essential to understand their range of applicability and limitations.

Brief overview of turbulence modeling in CFD

Turbulence modeling – Starting equations

- Let us take a glimpse to a very popular turbulence model, the $k-\omega$ turbulence model [1].
- Remember, as we are introducing additional equations, we need to define boundary conditions and initial conditions for the new variables.
- These new variables, in this case, k and ω, do not have any physical meaning.
- They were introduced to model the Reynolds stress tensor (which contains the velocity fluctuations).
- In Lecture 6, we will study many turbulence models (including this one).

[1] D. Wilcox. Turbulence Modeling for CFD. DCW Industries Inc., 2010.

Brief overview of turbulence modeling in CFD

$k - \omega$ Turbulence model equations

- It is called $k - \omega$ because it solves two additional equations for modeling the turbulent flow, namely,
  - The turbulent kinetic energy k .
  - The specific rate of dissipation ω.

- The closure equations of the $k - \omega$ turbulence model are,

$$\rho \frac{\partial k}{\partial t} + \rho \nabla . \left(\bar{\mathbf{u}} k \right) = \tau^R : \nabla \bar{\mathbf{u}} - \beta^* \rho k \omega + \nabla . \left[\left(\mu + \sigma^* \mu_T \right) \nabla k \right]$$

$$\rho \frac{\partial \omega}{\partial t} + \rho \nabla . \left(\bar{\mathbf{u}} \omega \right) = \alpha \frac{\omega}{k} \tau^R : \nabla \bar{\mathbf{u}} - \beta \rho \omega^2 + \nabla . \left[\left(\mu + \sigma \mu_T \right) \nabla \omega \right]$$

- These are not physical properties.
- They kind of represent the generation and destruction of turbulence.

$k-\omega$ Turbulence model equations

- The previous equations are used to compute the turbulent viscosity μ_t.
- In the $k-\omega$ model, the turbulent viscosity is computed as follows,

$$\mu_t = \frac{\rho k}{\omega}$$

- As we have done so far, if you check the dimensional groups, you will see that this combination of variables result in viscosity units.
- The turbulent viscosity is introduced to take into account the increased mixing and shear stresses due to the turbulence.
- So at this point the question is, how do we model the Reynolds stress tensor?
- There are many methods, we will briefly outline the most widely used.

Brief overview of turbulence modeling in CFD

$k-\omega$ Turbulence model equations

- The most widely approach used to model the Reynolds stress tensor is to use the Boussinesq hypothesis.
- By using the Boussinesq hypothesis, we can relate the Reynolds stress tensor to the mean velocity gradient such that,

$$\tau^R = -\rho\left(\overline{\mathbf{u}'\mathbf{u}'}\right) = 2\mu_T\overline{\mathbf{D}}^R - \frac{2}{3}\rho k\mathbf{I} = \mu_T\left[\nabla\overline{\mathbf{u}} + \left(\nabla\overline{\mathbf{u}}\right)^{\mathrm{T}}\right] - \frac{2}{3}\rho k\mathbf{I}$$

Computed from k and ω

- Each turbulence model will compute the turbulent viscosity in a different way.

Brief overview of turbulence modeling in CFD

Final touches to the incompressible RANS equations

- Recall the incompressible RANS equations,

$$\nabla \cdot (\bar{\mathbf{u}}) = 0$$

$$\frac{\partial \bar{\mathbf{u}}}{\partial t} + \nabla \cdot (\bar{\mathbf{u}}\bar{\mathbf{u}}) = -\frac{1}{\rho}(\nabla p) + \nu \nabla^2 \bar{\mathbf{u}} + \frac{1}{\rho} \nabla \cdot \boldsymbol{\tau}^R \qquad \text{where} \qquad \boldsymbol{\tau}^R = -\rho \left(\overline{\mathbf{u}'\mathbf{u}'}\right)$$

- By using the Boussinesq approximation, we can write the governing equations as follows,

$$\nabla \cdot (\bar{\mathbf{u}}) = 0$$

$$\frac{\partial \bar{\mathbf{u}}}{\partial t} + \nabla \cdot (\bar{\mathbf{u}}\bar{\mathbf{u}}) = -\frac{1}{\rho}\left(\nabla \bar{p} + \frac{2}{3}\rho \nabla k\right) + \nabla \cdot \left[\frac{1}{\rho} \overbrace{(\mu + \mu_t)}^{\text{Effective viscosity}} \nabla \bar{\mathbf{u}}\right]$$

Normal stresses arising from the Boussinesq approximation

Turbulent viscosity

- Then the turbulent viscosity μ_t is computed using a closure model.

Brief overview of turbulence modeling in CFD

$k-\omega$ Turbulence model equations

- Let me remind you the base units of the derived quantities used in the $k-\omega$ turbulence model.

Derived quantity	Symbol	Dimensional units	SI units
Turbulent kinetic energy per unit mass	k	L^2T^{-2}	m^2/s^2
Specific dissipation rate	ω	T^{-1}	1/s
Dynamic viscosity (laminar)	μ	$ML^{-1}T^{-1}$	Kg/m-s
Dynamic viscosity (turbulent)	μ_t	$ML^{-1}T^{-1}$	Kg/m-s

- The turbulent eddy viscosity is not a fluid property, it is a property needed by the turbulence model.
- In turbulence modeling we will use many quantities that appear to be magical. To get an idea of those quantities, always check the dimensional groups.

Brief overview of turbulence modeling in CFD

Overview of the main turbulence modeling approaches.

MODELING APPROACH

RANS
Reynolds-**A**veraged **N**avier-**S**tokes equations

URANS
Unsteady **R**eynolds-**A**veraged **N**avier-**S**tokes equations

- Many more acronyms that fit between **RANS/URANS** and **SRS**.
- Some of the acronyms are used only to differentiate approaches used in commercial solvers.

PANS, **SAS**, **RSM**, **EARSM**, **PITM**, **SBES, ELES**

SRS
Scale-**R**esolving **S**imulations

DES
Detached **E**ddy **S**imulations

LES
Large **E**ddy **S**imulations

DNS
Direct **N**umerical **S**imulations

Increasing computational cost

Increasing modelling and mathematical complexity

Brief overview of turbulence modeling in CFD

Short description of some RANS turbulence models

Model	Short description
Spalart-Allmaras	This is a one equation model. Suitable for external aerodynamics, tubomachinery and high speed flows. Good for mildly complex external/internal flows and boundary layer flows under pressure gradient (e.g. airfoils, wings, airplane fuselages, ship hulls). Performs poorly with flows with strong separation.
Standard k–epsilon	This is a two equation model. Very robust and 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. Can be only used with wall functions.
Realizable k–epsilon	This is a two equation model. 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). It overcome the limitations of the standard k-epsilon model.
Standard k–omega	This is a two equation model. 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).
SST k–omega	This is a two equation model. Offers similar benefits as the 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. Can be used with and without wall functions. Probably the most widely used RANS model.

"Essentially, all models are wrong,
but some are useful"

G. E. P. Box

George Edward Pelham Box
18 October 1919 – 28 March 2013. Statistician, who worked in the areas of quality control, time-series analysis, design of experiments, and Bayesian inference. He has been called *"one of the great statistical minds of the 20th century"*.