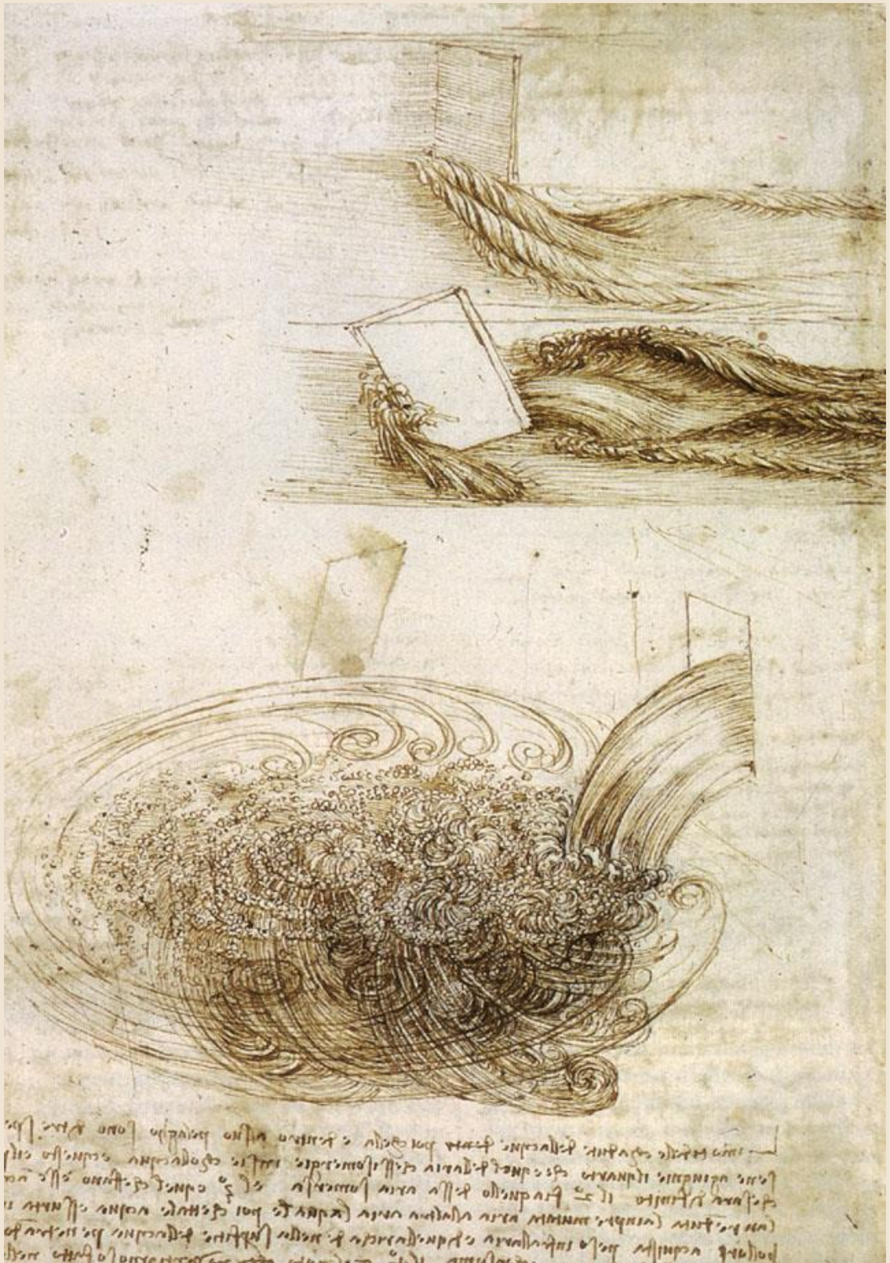
“Essentially, all models are wrong,
but some are useful”

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”*.



Leonardo da Vinci pioneered the flow visualization genre about 500 years ago. The illustrations to the left (*Studies of water passing obstacles and falling* c. 1508-1509) represents perhaps the world's first use of visualization as a scientific tool to study a turbulent flow.

The following da Vinci's observation is close to the Reynold's decomposition.

> ***"Observe the motion of the surface of the water, which resembles that of hair, which has two motions, of which one is caused by the weight of the hair, the other by the direction of the curls; thus the water has eddying motions, one part of which is due to the principal current, the other to the random and reverse motion."***

The following da Vinci's description is maybe the earliest reference to the importance of vortices in fluid motion.

> ***"So moving water strives to maintain the course pursuant to the power which occasions it and, if it finds an obstacle in its path, completes the span of the course it has commenced by a circular and revolving movement."***

The following da Vinci's observation is an analogy to the energy cascade and coherent structures.

> ***"...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"***

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
    - Dissipation
    - Large Reynolds numbers
    - Three-Dimensional Vorticity fluctuations
    - Continuum
    - Feature of a flow, not fluid

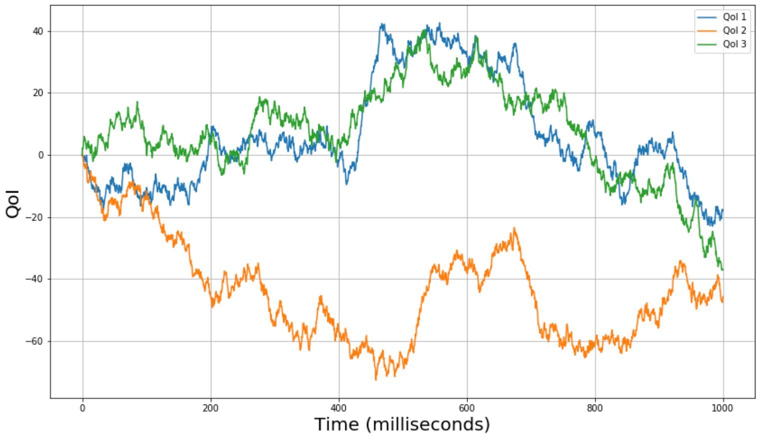
[1] H. Tennekes and J. L. Lumley. "A first course in turbulence". MIT Press, 1972.

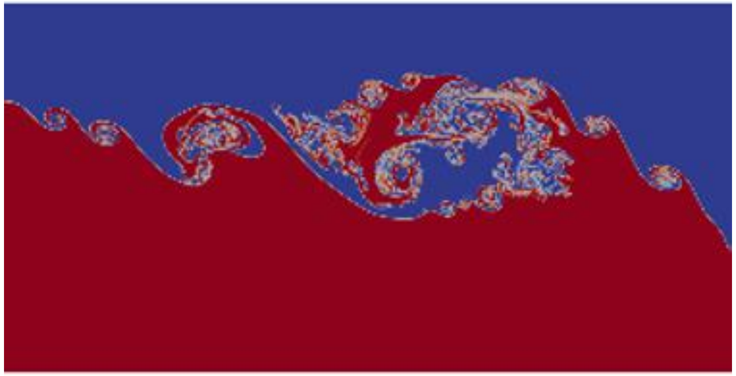
Turbulent flows have the following characteristics

- 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 cause 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.

What is turbulence?

- For the purpose of this training, 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.

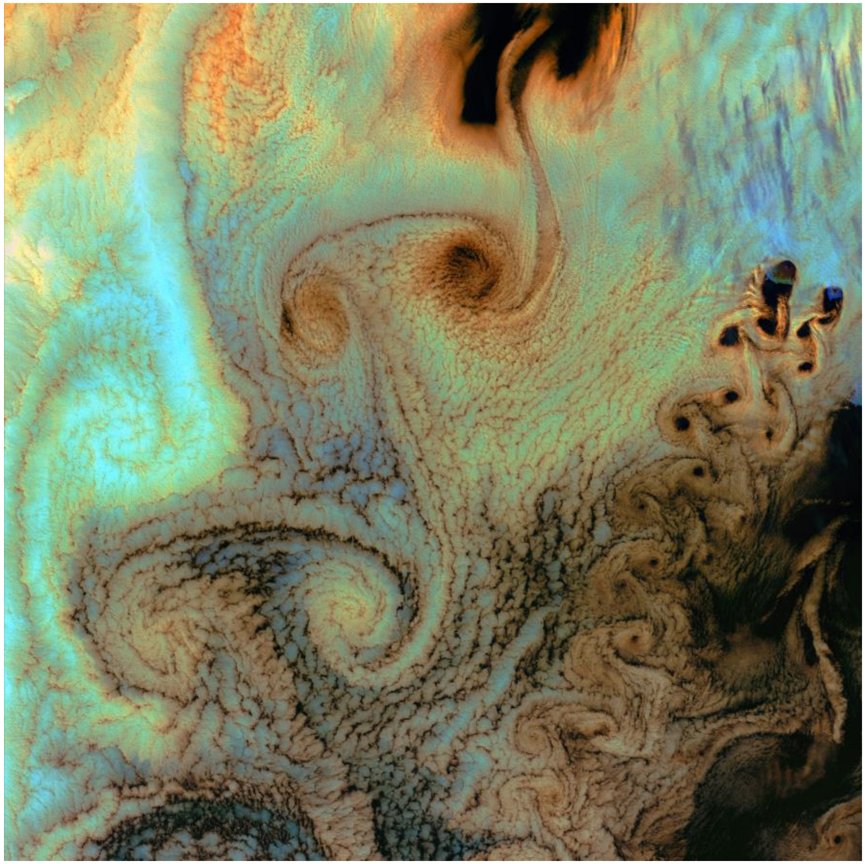




The turbulent world around us

Buoyant plume of smoke rising from a stick of incense
Photo credit: https://www.flickr.com/photos/jlhopgood/

The turbulent world around us



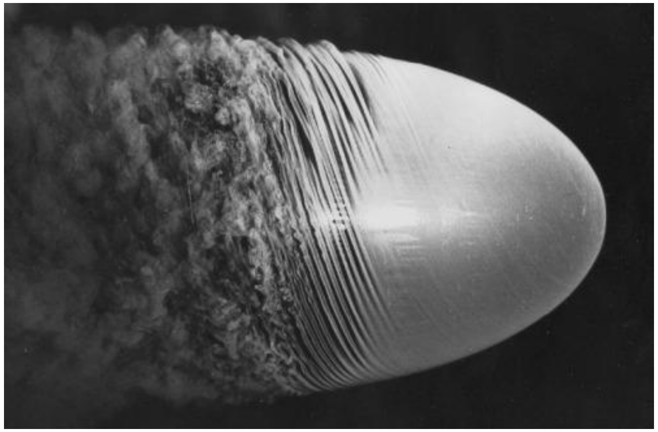
Von Karman vortices created when prevailing winds sweeping east across the northern Pacific Ocean encountered Alaska's Aleutian Islands
Photo credit: USGS EROS Data Center Satellite Systems Branch.

Von Karman Vortex Streets in the northern Pacific Photographed from the International Space Station
Photo credit: NASA

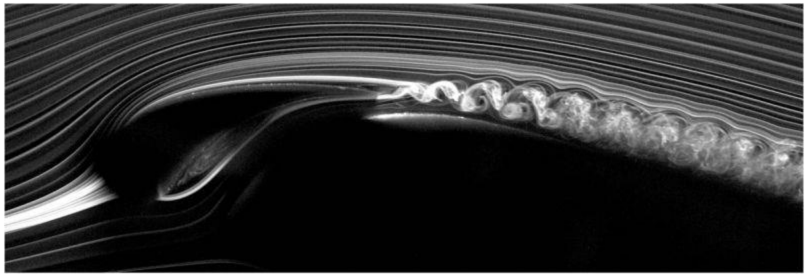
The turbulent world around us

Tugboat riding on the turbulent wake of a ship
Photo credit: https://www.flickr.com/photos/oneeighteen/

Trailing vortices
Photo credit: Steve Morris. AirTeamImages.

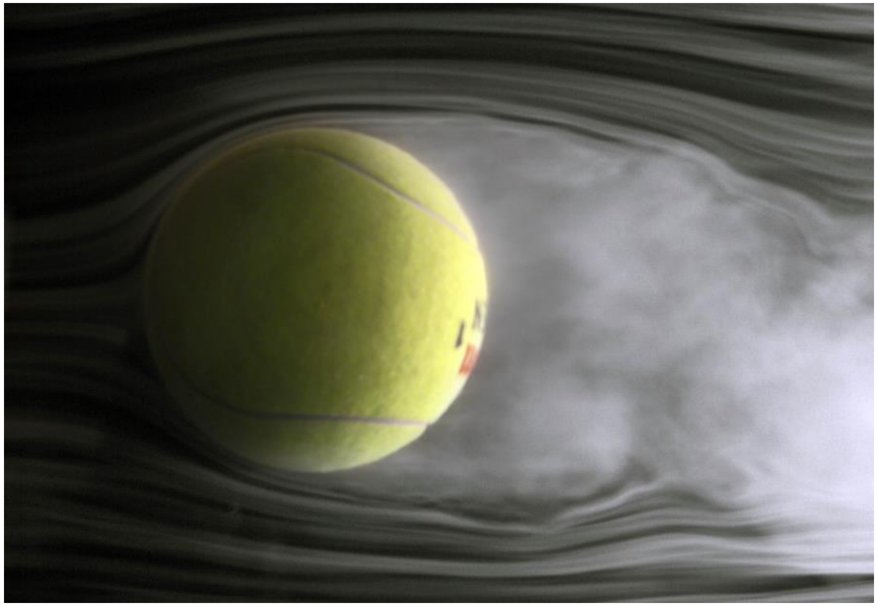


Flow visualization over a spinning spheroid
Photo credit: Y. Kohama.



Flow around an airfoil with a leading-edge slat
Photo credit: S. Makiya et al.

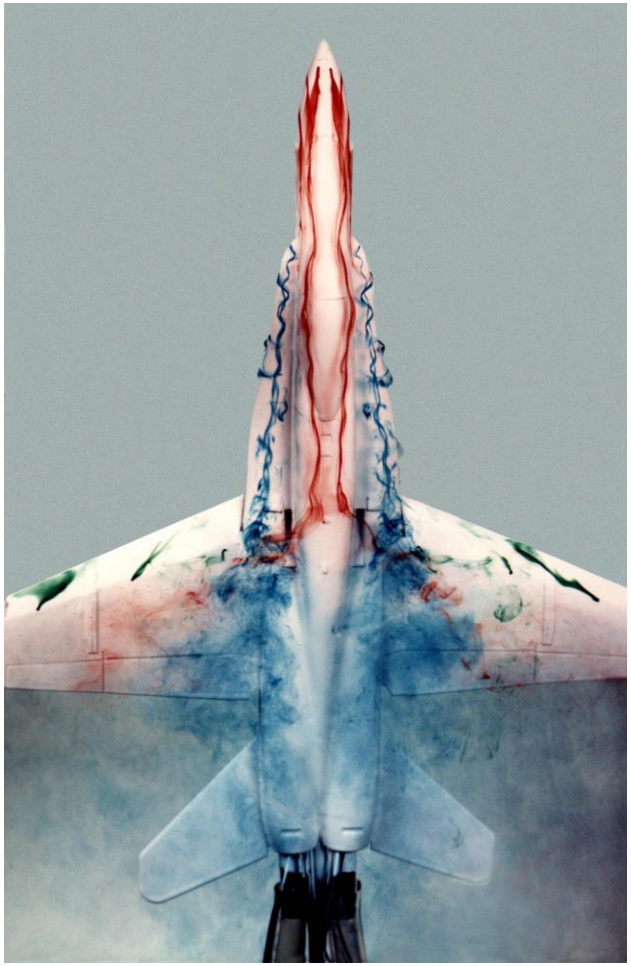
The turbulent world around us

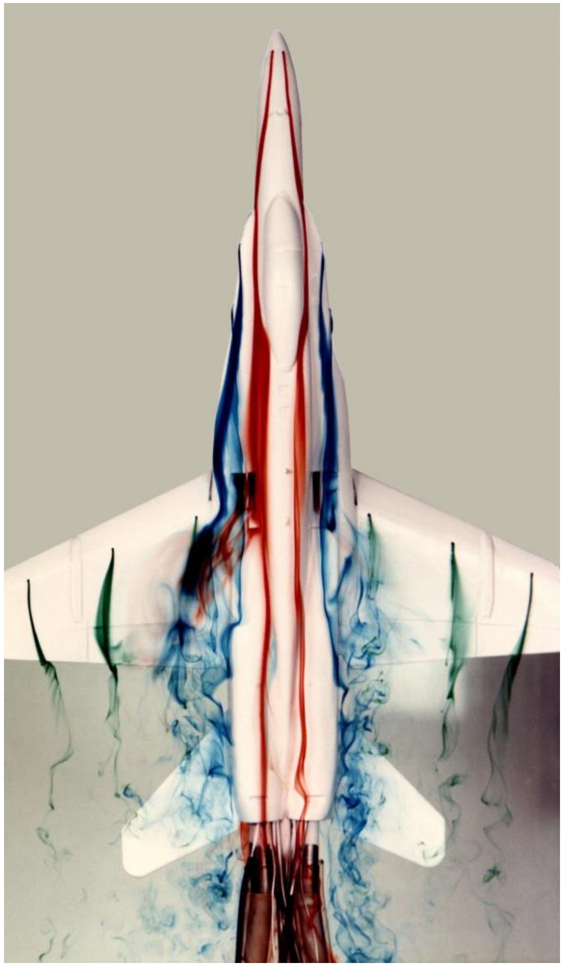


Wind Tunnel Test of New Tennis Ball
Photo credit: NASA
http://tennisclub.gsfc.nasa.gov/tennis.windtunnelballs.html

Wake turbulence behind individual wind turbines
Photo credit: NREL's wind energy research group.

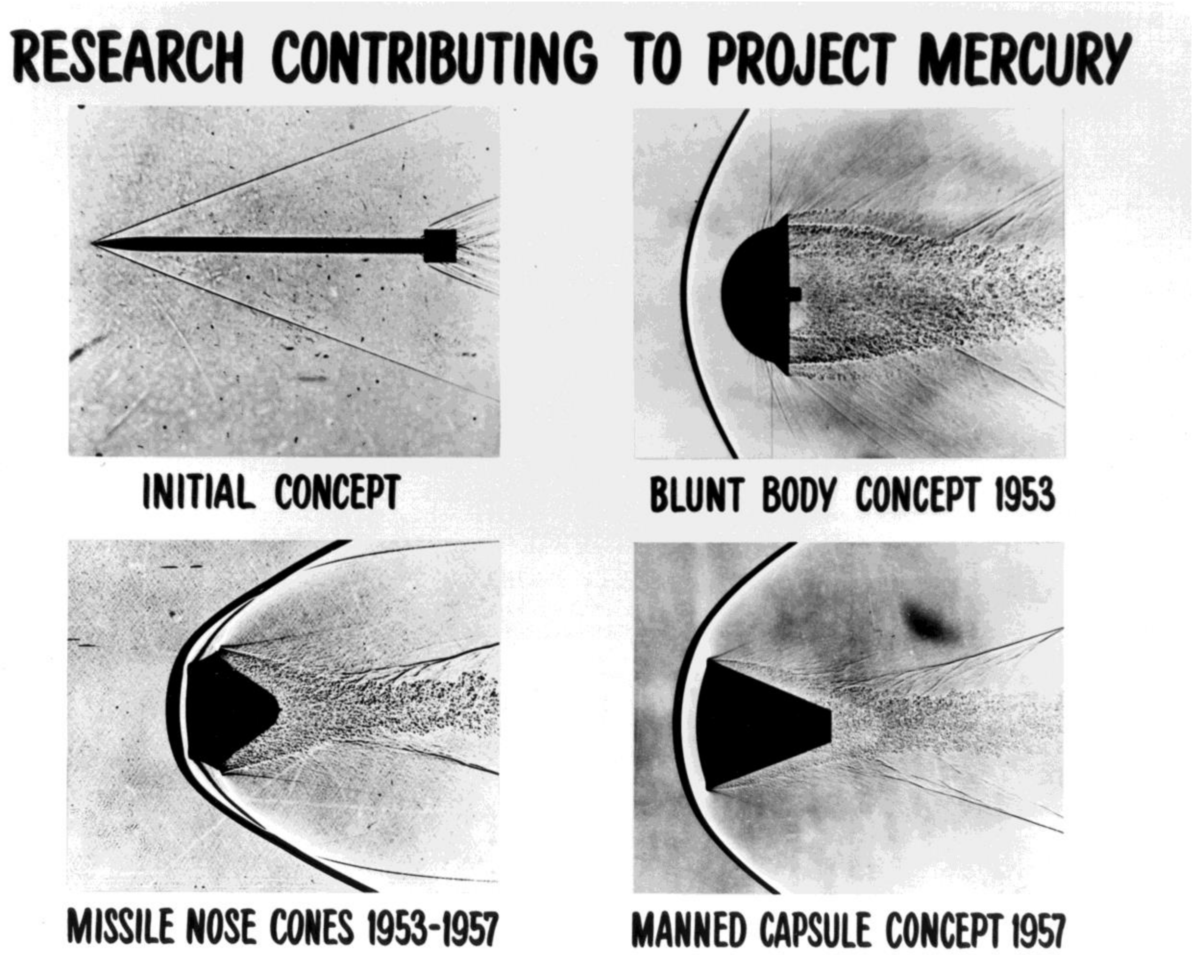
The turbulent world around us





Vortices on a 1/48-scale model of an F/A-18 aircraft inside a Water Tunnel
Photo credit: NASA Dryden Flow Visualization Facility. http://www.nasa.gov/centers/armstrong/multimedia/imagegallery/FVF

The turbulent world around us

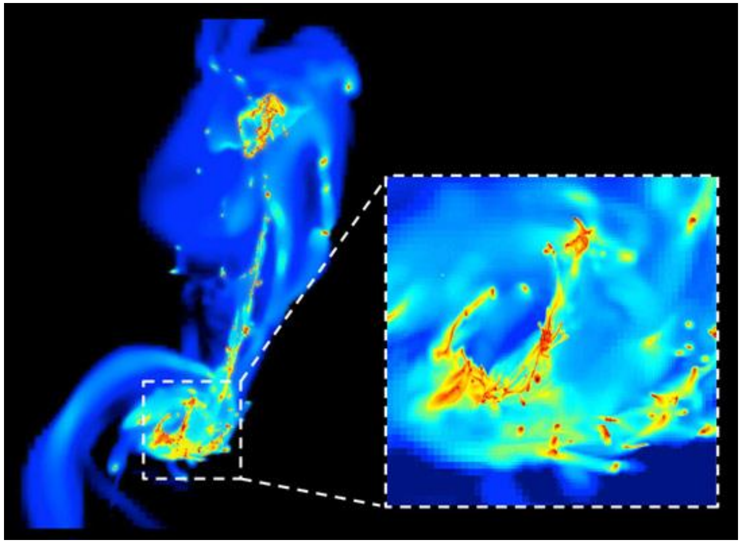


Shadowgraph Images of Re-entry Vehicles
Photo credit: NASA on the Commons. https://www.flickr.com/photos/nasacommons/

The turbulent world around us

Astrophysical, plasma, planetary and quantum turbulence

M8: The Lagoon Nebula
Photo credit: Steve Mazlin, Jack Harvey, Rick Gilbert, and Daniel Verschatse. Star Shadows Remote Observatory, PROMPT, CTIO

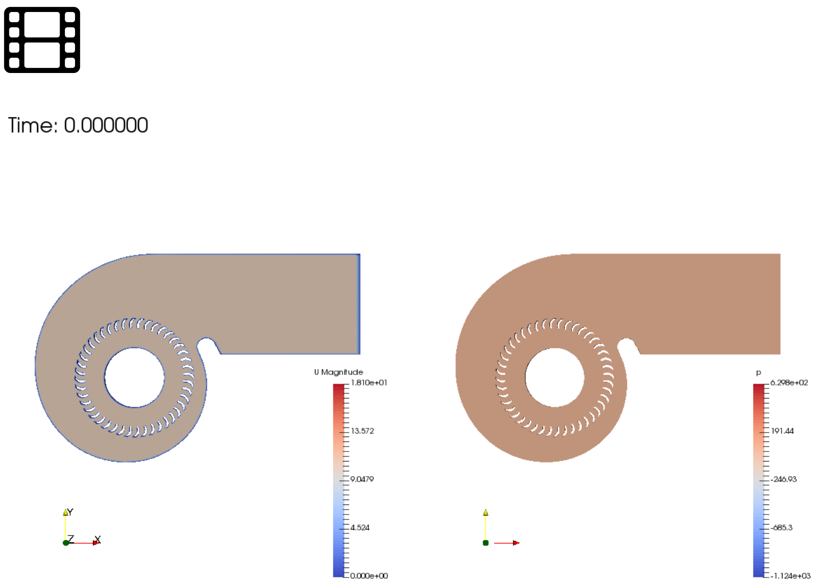


A frame from the simulation of the two colliding Antennae galaxies.
Photo credit: F. Renaud / CEA-Sap.

Jupiter photo taken by Juno's cam.
Photo credit: NASA / JPL / SwRI / MSSS / David Marriott

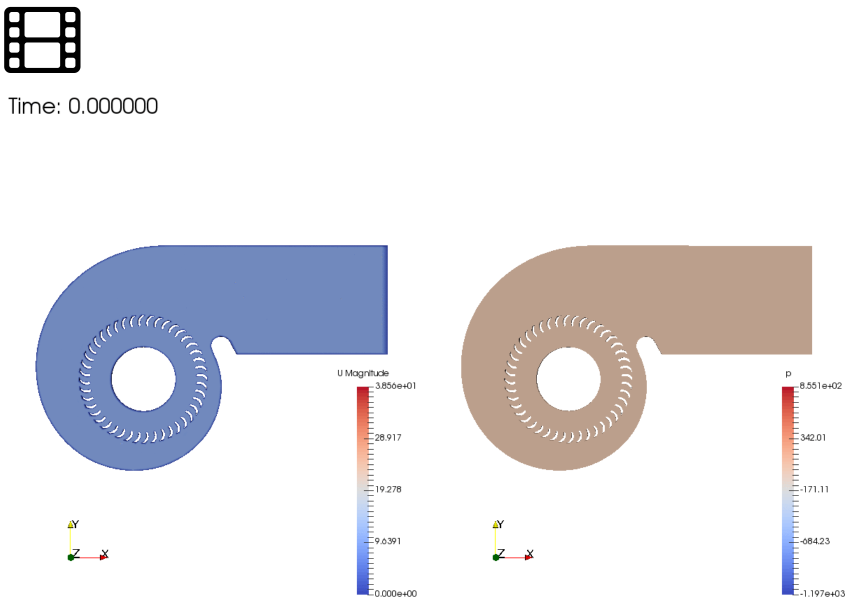
Turbulence, does it matter?

Blower simulation using sliding grids



No turbulence model used (laminar, no turbulence modeling, DNS, unresolved DNS, name it as you want)

http://www.wolfdynamics.com/training/turbulence/image1.gif

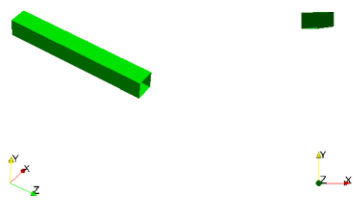


K-epsilon turbulence model

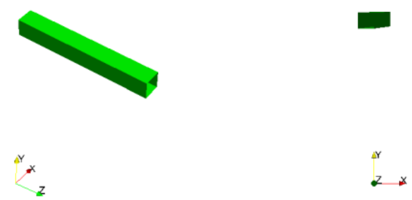
http://www.wolfdynamics.com/training/turbulence/image2.gif

Turbulence, does it matter?

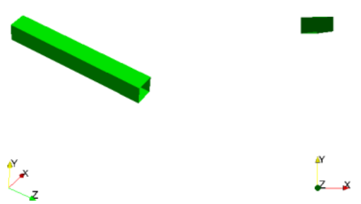
Vortex shedding past square cylinder



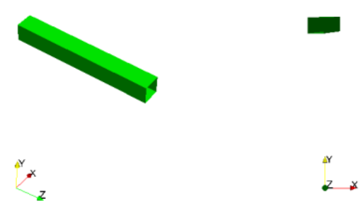
URANS (K-Omega SST with no wall functions) – Vortices visualized by Q-criterion
www.wolfdynamics.com/wiki/squarecil/urans2.gif



LES (Smagorinsky) – Vortices visualized by Q-criterion
www.wolfdynamics.com/wiki/squarecil/les.gif



Laminar (no turbulence model) – Vortices visualized by Q-criterion
www.wolfdynamics.com/wiki/squarecil/laminar.gif



DES (SpalartAllmarasDDES) – Vortices visualized by Q-criterion
www.wolfdynamics.com/wiki/squarecil/des.gif

Turbulence, does it matter?

Vortex shedding past square cylinder

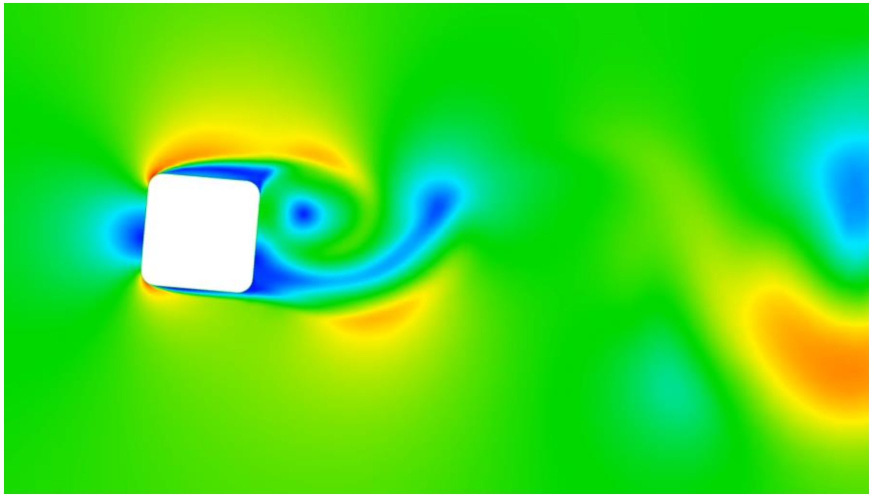
Turbulence model	Drag coefficient	Strouhal number	Computing time (s)
Laminar	2.81	0.179	93489
LES	2.32	0.124	77465
DES	2.08	0.124	70754
SAS	2.40	0.164	57690
URANS (WF)	2.31	0.130	67830
URANS (No WF)	2.28	0.135	64492
RANS	2.20	-	28246 (10000 iter)
Experimental values	2.05-2.25	0.132	-

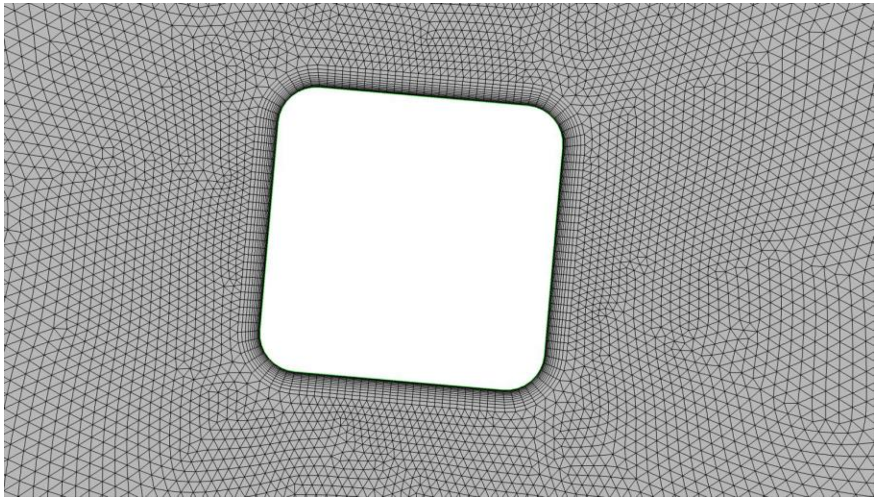
Note: all simulations were run using 4 cores.

References:
D. A. Lyn and W. Rodi. “The flapping shear layer formed by flow separation from the forward corner of a square cylinder”. *J. Fluid Mech., 267, 353, 1994.*
D. A. Lyn, S. Einav, W. Rodi and J. H. Park. “A laser-Doppler velocimetry study of ensemble-averaged characteristics of the turbulent near wake of a square cylinder”. *Report. SFB 210 /E/100.*

Turbulence, does it matter?

Transitional flow past square cylinder with rounded corners – Re = 54000

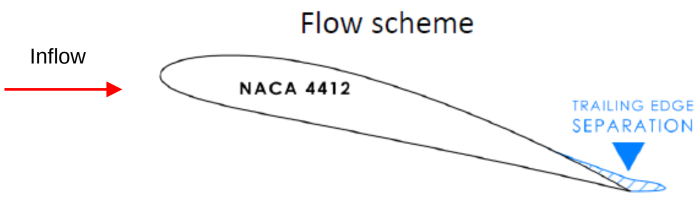




Velocity magnitude
www.wolfdynamics.com/wiki/turb/media1.mp4

Turbulence model	Drag coefficient	Lift coefficient
DNS	0.06295	0.07524
LES	0.1146	0.03269
SAS	0.1058	0.0258
URANS (No WF)	0.1107	0.00725
Transition K-KL-Omega	0.059	-0.0104
Transition K-Omega SST	0.0987	-0.0143
Experimental values	0.045 to 0.075	-0.011 to -0.015

Separated flow around a NACA-4412 airfoil



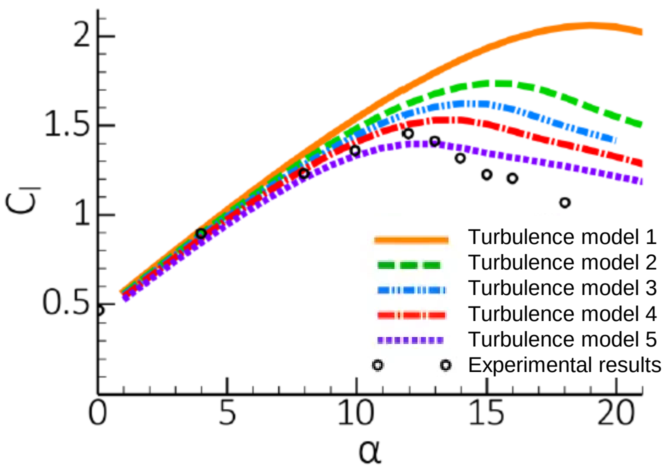
Incompressible flow

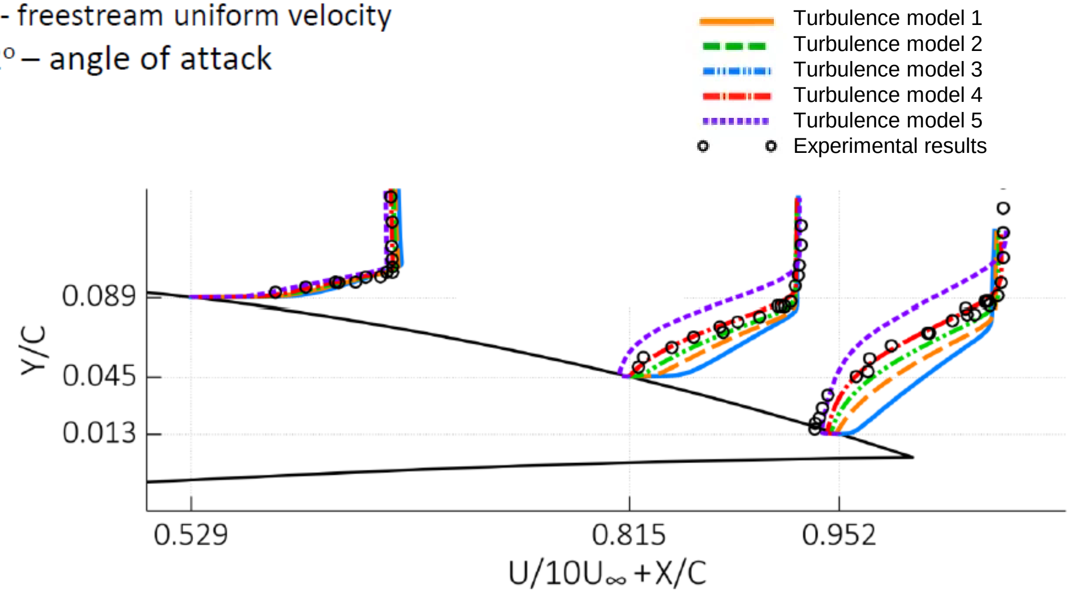
$Re = U_\infty \cdot C/\nu = 1.64 \cdot 10^6$

C - airfoil chord

U_∞ - freestream uniform velocity

$\alpha = 12^{o}$ – angle of attack





- CFD has been around since the late 1970s, and after all these years is not that easy to compute the flow around 2D airfoils.
- In particular, predicting the maximum lift and stall characteristics is not trivial.

References:
F. Menter. "A New Generalized k-omega model. Putting flexibility into Turbulence models (GEKO)", Ansys Germany
A. J. Wadcock. "Investigation of Low-Speed Turbulent Separated Flow Around Airfoils", NASA Contractor Report 177450

Turbulence, does it matter?

Turbulence is not a trivial problem

"Turbulence is the most important unresolved problem of classical physics"

Richard Feynman

"Turbulence was probably invented by the devil on the seventh day of creation when the good lord was not looking"

Peter Bradshaw (1994)

"Turbulence is the graveyard of theories"

Hans W. Liepmann (1997)

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 of 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.

Reynolds number and Rayleigh number

- It is well known that the Reynolds number characterizes if the flow is laminar or turbulent.
- So before doing a simulation or experiment, check if the flow is turbulent.
- The Reynolds number is defined as follows,

Convective effects

$$Re_L = \frac{\rho U L}{\mu}$$

Viscous effects

- Where *U* is a characteristic velocity, *e.g.*, free-stream velocity.
- And *L* is representative length scale, *e.g.*, length, height, diameter, etc.

Reynolds number and Rayleigh number

- If you are dealing with natural convection, you can use the Rayleigh number, Grashof number, and Prandtl number to characterize the flow.

Specific heat

Thermal expansion coefficient

Buoyancy effects

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

Viscous effects

Thermal conductivity

Momentum diffusivity

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

Thermal diffusivity

$$Gr = \frac{g\beta (T_S - T_\infty) L^3}{\nu^2}$$

Reynolds number and Rayleigh number

- Turbulent flow occurs at large Reynolds number.
  - For external flows,

$$Re_x \geq 500000$$ Around slender/streamlined bodies (surfaces)

$$Re_d \geq 20000$$ Around an obstacle (bluff bodies)

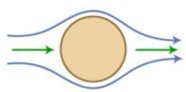
  - For internal flows,

$$Re_{d_h} \geq 2300$$

- Notice that other factors such as free-stream turbulence, surface conditions, blowing, suction, roughness and other disturbances, may cause transition to turbulence at lower Reynolds number.
- If you are dealing with natural convection and buoyancy, turbulent flows occurs when

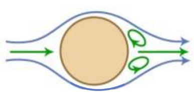
$$\frac{Ra}{Pr} \geq 10^9$$

What happens when we increase the Reynolds number?



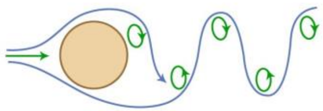
Creeping flow (no separation)
Steady flow

$Re < 5$



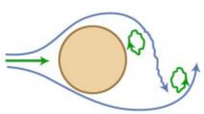
A pair of stable vortices in the wake
Steady flow

$5 < Re < 40 - 46$



Laminar vortex street (Von Karman street)
Unsteady flow

$40 - 46 < Re < 150$

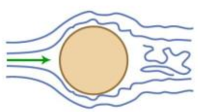


Laminar boundary layer up to the separation point, turbulent wake
Unsteady flow

$150 < Re < 300$

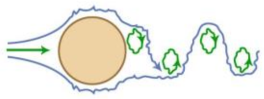
Transition to turbulence

$300 < Re < 3 \times 10^5$



Boundary layer transition to turbulent
Unsteady flow

$3 \times 10^5 < Re < 3 \times 10^6$



Turbulent vortex street, but the wake is narrower than in the laminar case
Unsteady flow

$3 \times 10^6 > Re$

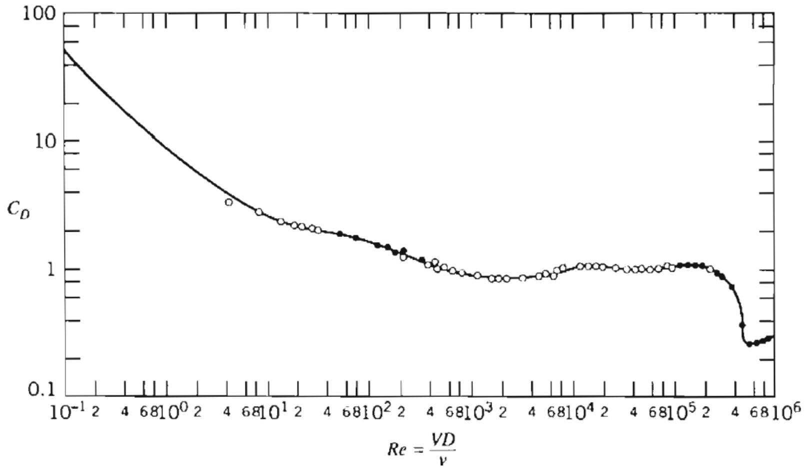
- Easy to simulate
- Steady

- Relative easy to simulate.
- It becomes more challenging when the boundary layer transition to turbulent
- Unsteady

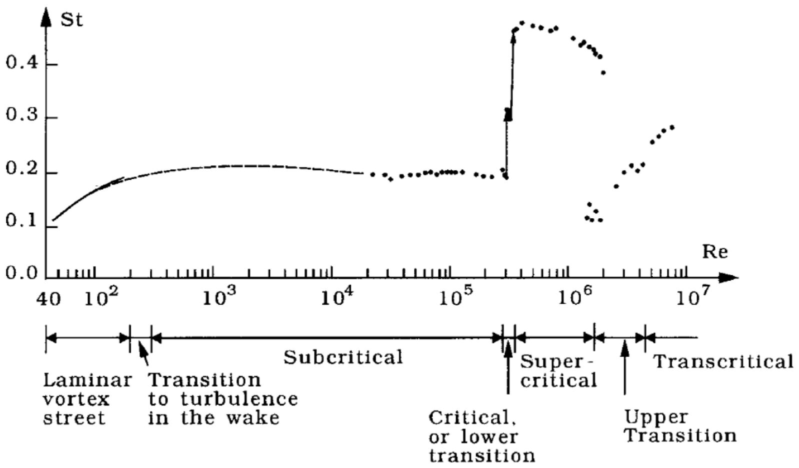
- Challenging to simulate
- Unsteady

Vortex shedding behind a cylinder and Reynolds number

What happens when we increase the Reynolds number?



Drag coefficient as a function of Reynolds number for a smooth cylinder [1]

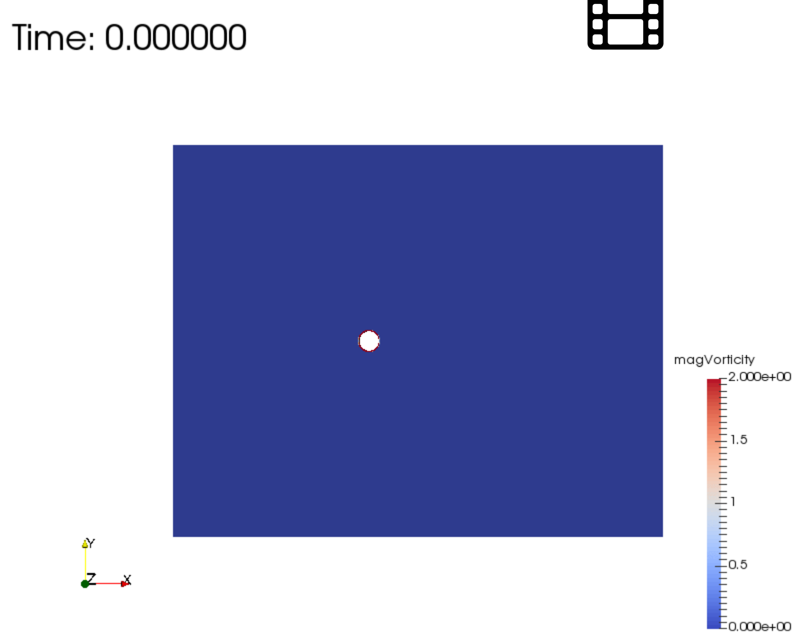


Strouhal number for a smooth cylinder [2]

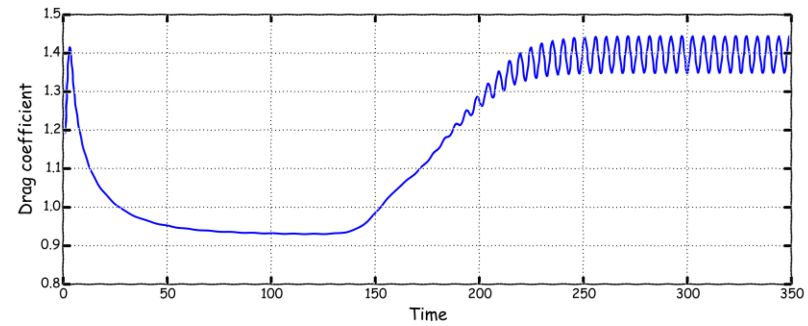
References:

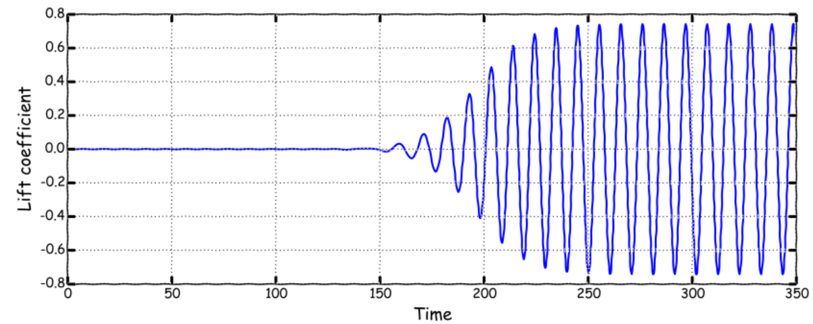
1. Fox, Robert W., et al. Introduction to Fluid Mechanics. Hoboken, NJ, Wiley, 2010
2. Sumer, B. Mutlu, et al. Hydrodynamics Around Cylindrical Structures. Singapore, World Scientic, 2006

Vorticity does not always mean turbulence



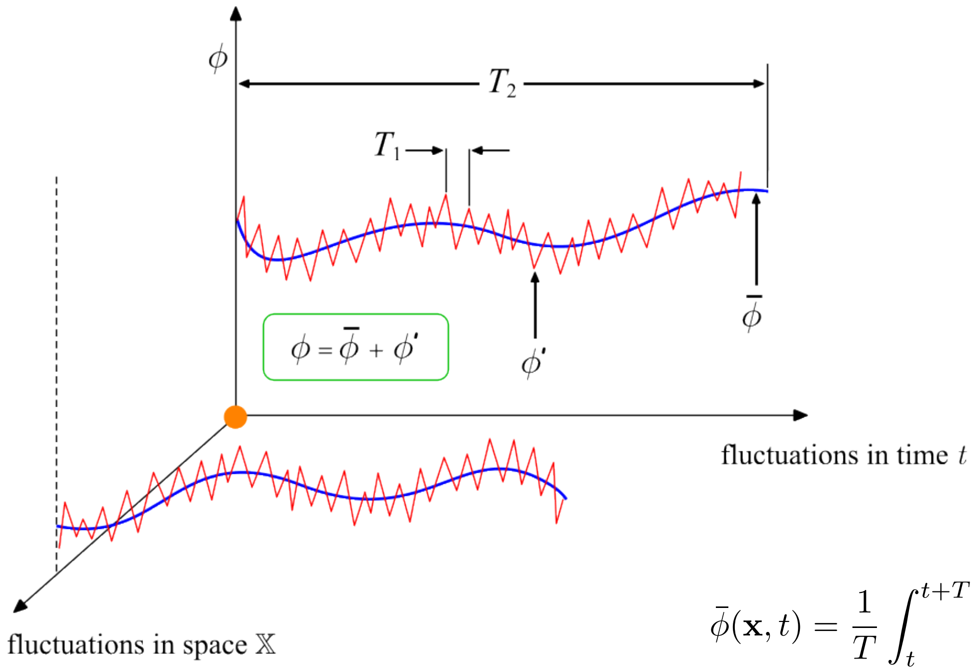
Instantaneous vorticity magnitude field
http://www.wolfdynamics.com/training/turbulence/image6.gif





- The Reynolds number in this case is 100, for these conditions, the flow still is laminar.
- We are in the presence of the Von Karman vortex street, which is the periodic shedding of vortices caused by the unsteady separation of the fluid around blunt bodies.
- Vorticity is not a direct indication of turbulence.
- However, turbulent flows are rotational, they exhibit vortical structures.

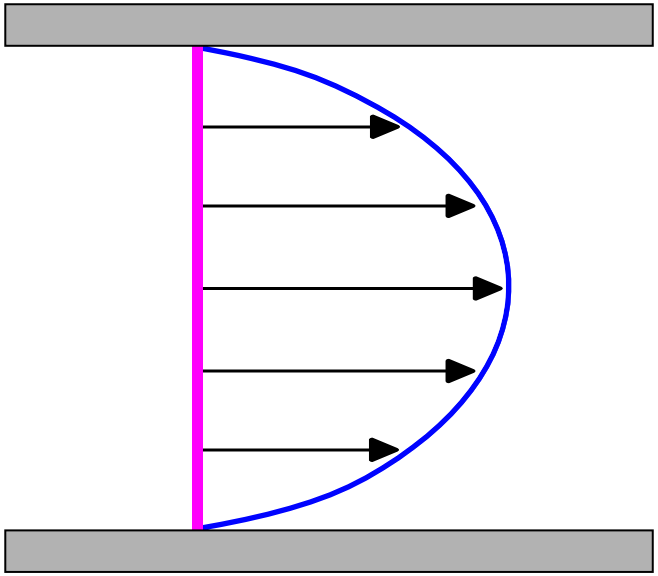
Turbulence modeling – Fluctuations of transported quantities



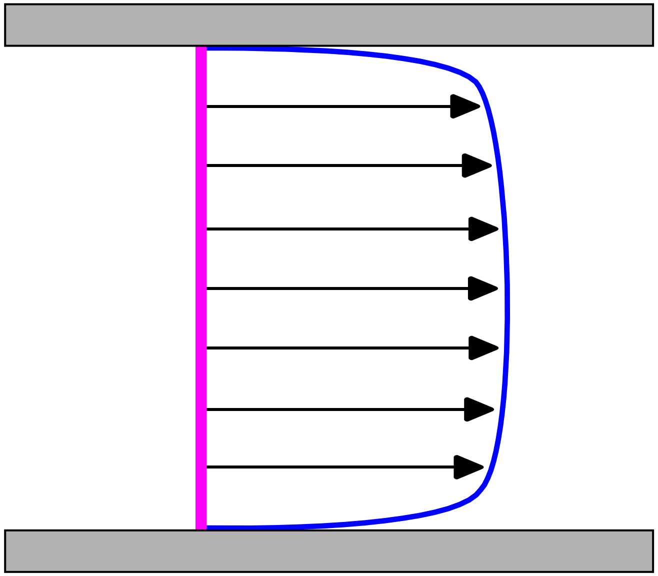
$$\bar{\phi}(\mathbf{x}, t) = \frac{1}{T}\int_t^{t+T} \phi(\mathbf{x}, t)\mathrm{d}t, \quad T_1 << T << T_2$$

- We have defined turbulence as an unsteady, aperiodic motion in which velocity components and every transported quantity fluctuate in space and time.
- For most engineering applications it is impractical to account for all these instantaneous fluctuations.
- Therefore, we need to somehow remove those small scales by using models.
- To remove the instantaneous fluctuations (or small scales), two methods can be used: Reynolds averaging and Filtering.
- Both methods introduce additional terms that must be modeled for closure.
- We are going to talk about closure methods later.

Turbulence modeling – Fluctuations of transported quantities



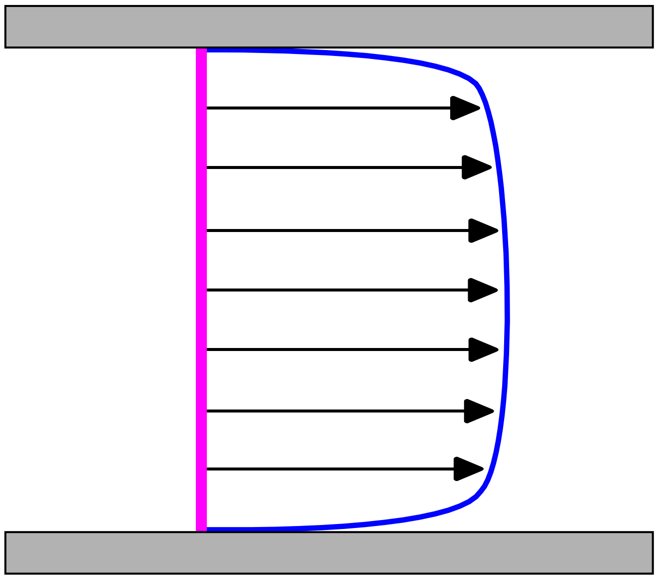
Laminar flow in a pipe



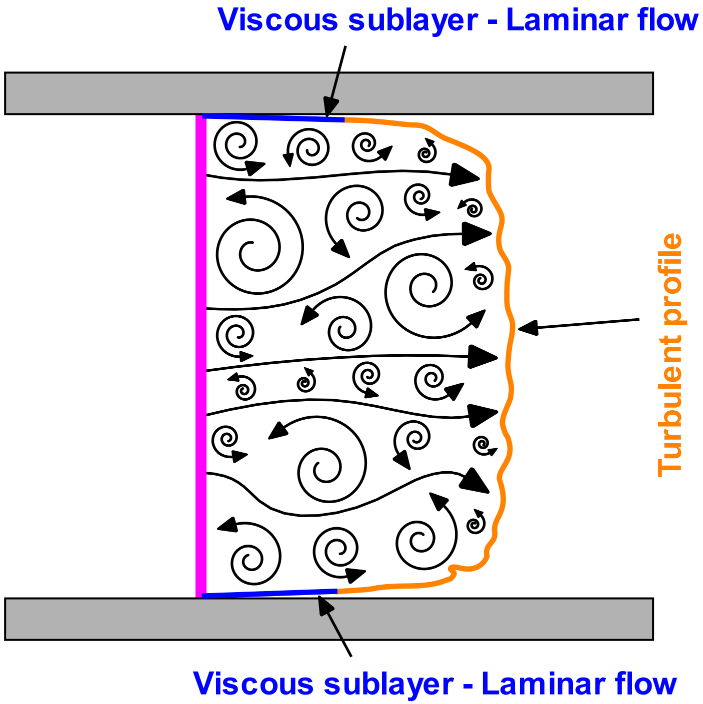
Turbulent flow in a pipe

- 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).

Turbulence modeling – Fluctuations of transported quantities



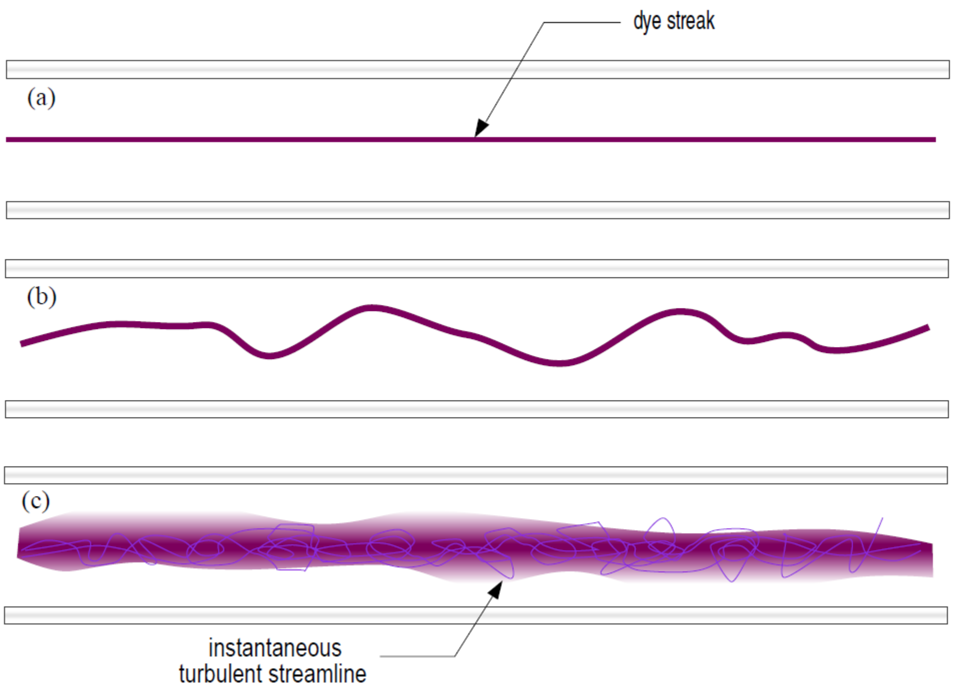
Averaged turbulent flow



Instantaneous turbulent flow

- In the left figure, the velocity profile has been averaged.
- In reality, the velocity profile fluctuates in time (right figure).
- The thin region close to the walls has very large velocity gradients and is laminar.
- Far from the flows, the flow becomes turbulent.

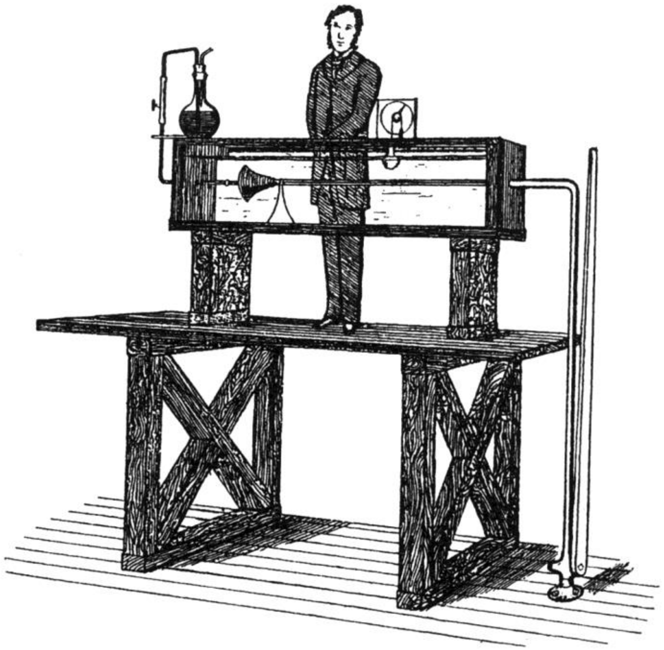
Turbulence modeling – Fluctuations of transported quantities

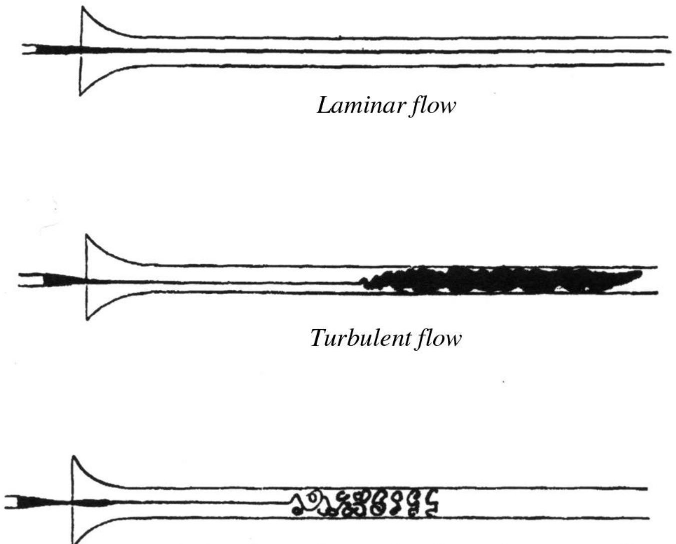


Flow in a pipe. (a) Laminar, (b) Transitional, (c) Turbulent

- Turbulence has a direct effect on the velocity profiles and mixing of transported quantities.
- Case (a) correspond to a laminar flow, where the dye can mix with the main flow only via molecular diffusion, this kind of mixing can take very long times.
- Case (b) shows a transitional state where the dye streak becomes wavy but the main flow still is laminar.
- Case (c) shows the turbulent state, where the dye streak changes direction erratically, and the dye has mixed significantly with the main flow due to the velocity fluctuations.

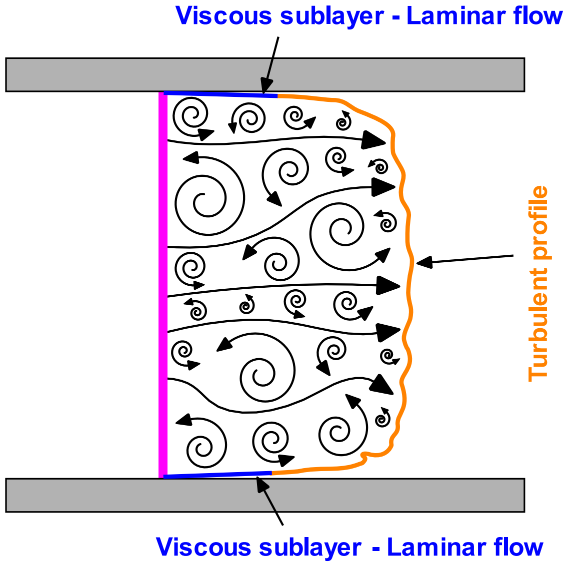
Turbulence modeling – Fluctuations of transported quantities



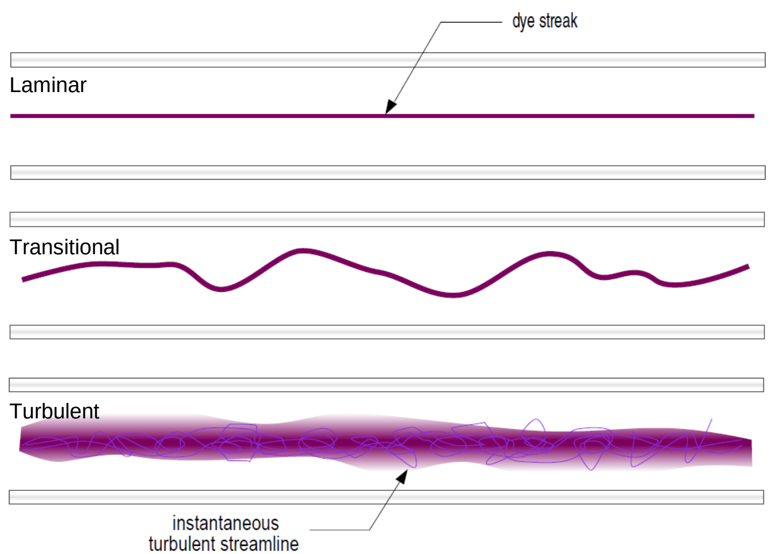


- Illustration taken from Osborne Reynolds' 1883 influential paper *"An experimental investigation of the circumstances which determine whether the motion of water in parallel channels shall be direct or sinuous and of the law of resistance in parallel channels".*
- Water flows from the tank near the experimenter down to below the ground, through a transparent tube; and dye is injected in the middle of the flow.
- The turbulent or laminar nature of the flow can therefore be observed precisely.

Turbulence modeling – Fluctuations of transported quantities



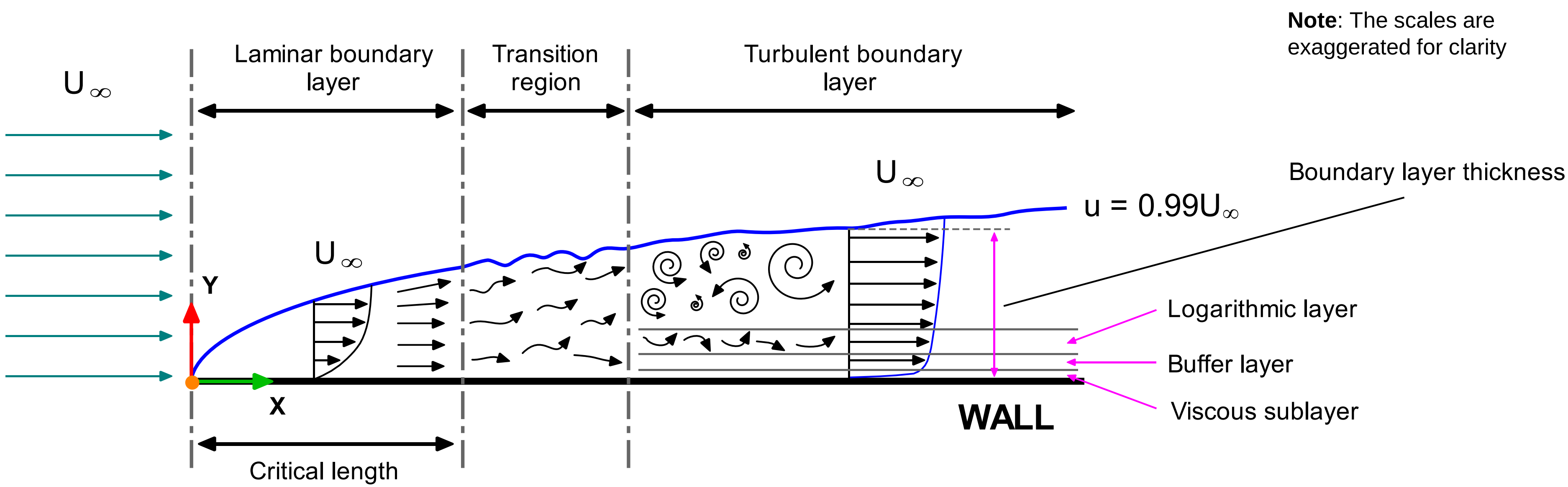
Wall bounded turbulence



Shear free turbulence

- Turbulent flows can originate at the walls. When this is the case, we talk about wall bounded turbulence.
- Turbulent flows can also originate in the absence of walls (or far from walls). When this is the case, we talk about shear free turbulence (usually jets, heated walls, atmospheric flows).

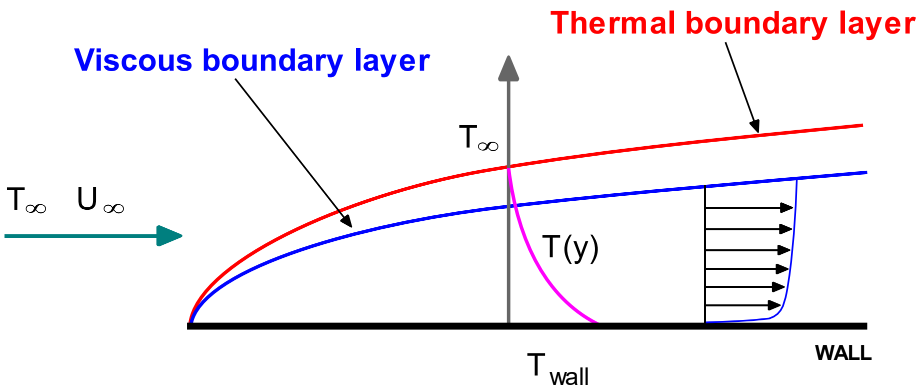
Turbulence modeling – Boundary layer



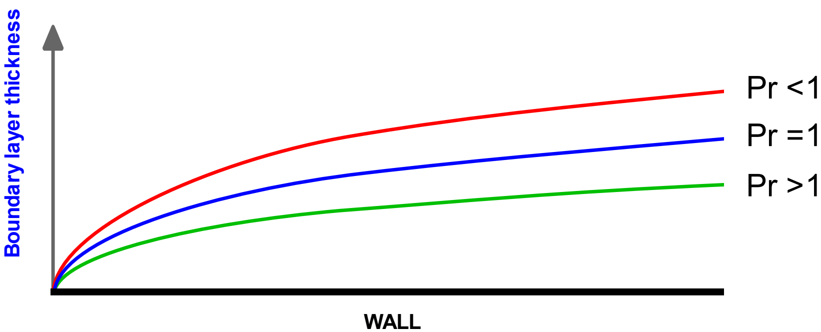
Boundary layer (Laminar-Transitional-Turbulent flow)

- 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.
- As for the pipe flow, the velocity profiles in the laminar and turbulent regions are different.

Turbulence modeling – Thermal boundary layer



Thermal boundary layer vs. Viscous boundary layer
Forced convection

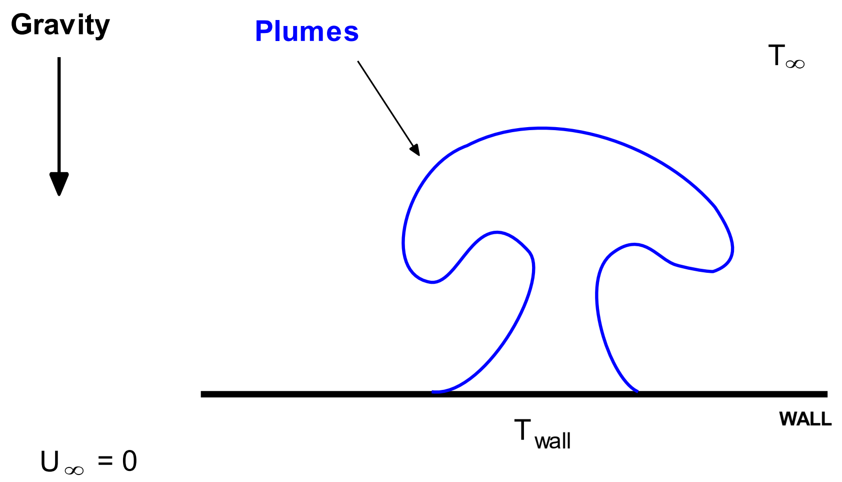


Thermal boundary layer in function of Prandtl number (Pr)

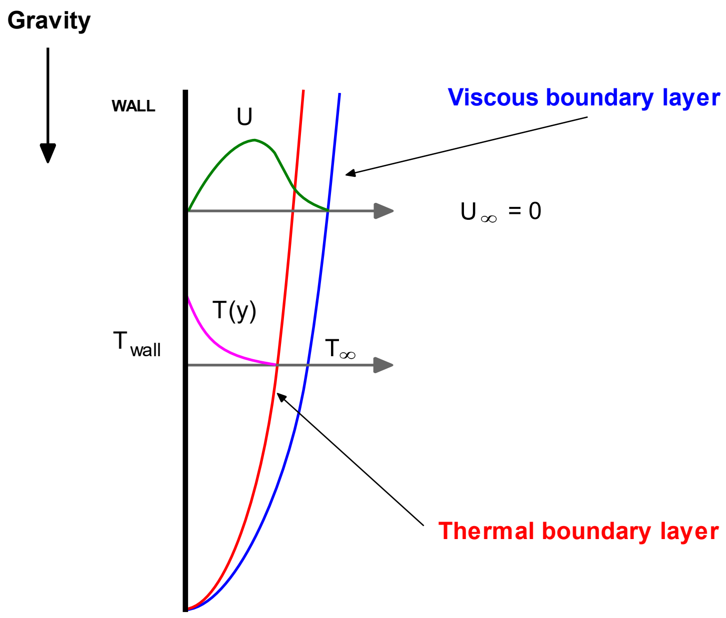
Momentum and thermal boundary layer

- Just as there is a viscous boundary layer in the velocity distribution (or momentum), there is also a thermal boundary layer.
- Thermal boundary layer thickness is different from the thickness of the viscous sublayer (momentum), and is fluid dependent.
- The thickness of the thermal sublayer for a high Prandtl number fluid (e.g. water) is much less than the momentum sublayer thickness.
- For fluids of low Prandtl numbers (*e.g.*, air), it is much larger than the momentum sublayer thickness.
- For Prandtl number equal 1, the thermal boundary layer is equal to the momentum boundary layer.

Turbulence modeling – Thermal boundary layer



Horizontal heated plate immersed in a quiescent fluid.
Natural convection

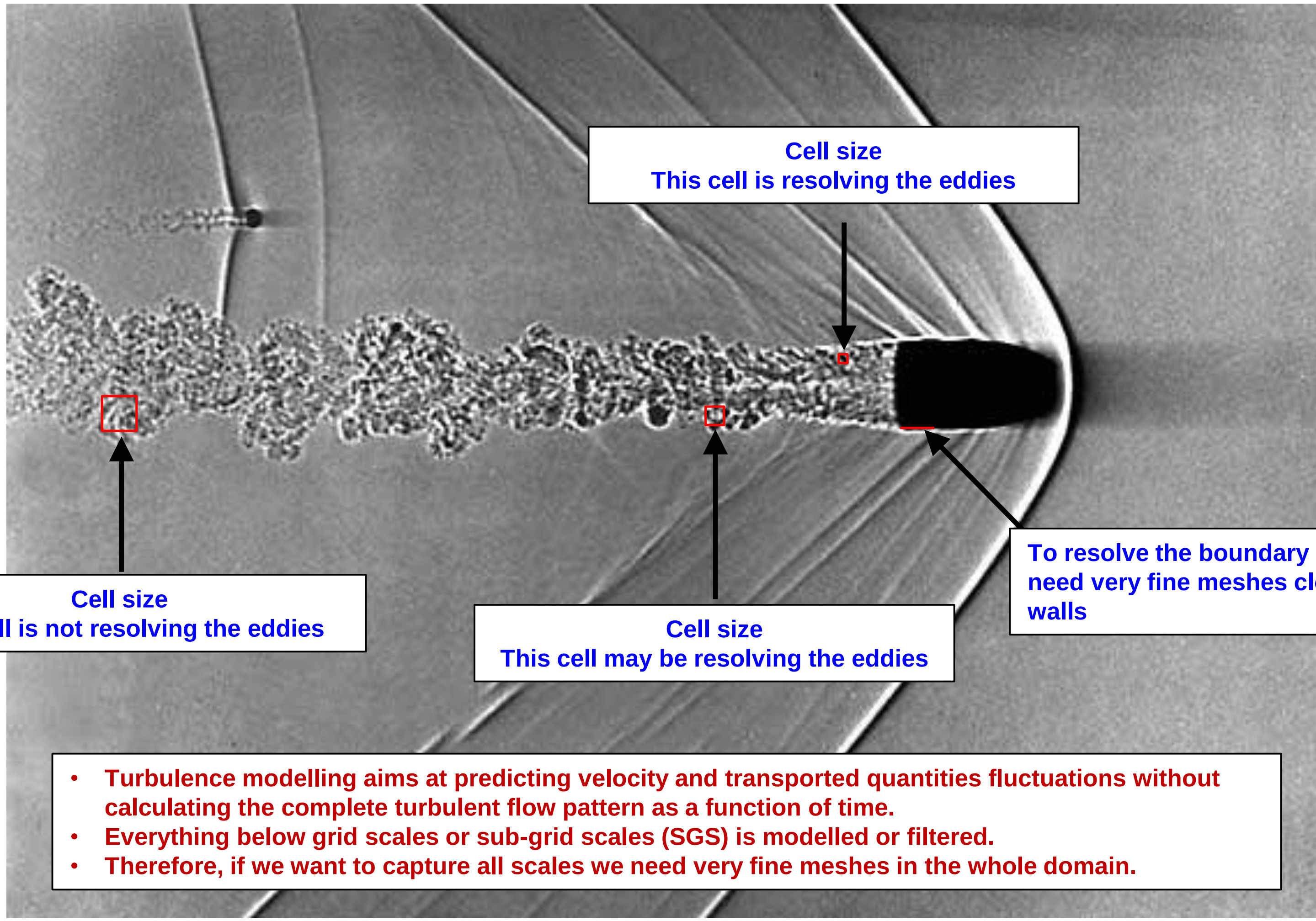


Vertical heated plate immersed in a quiescent fluid.
Natural convection.

Natural convection in a heated plate

- As the fluid is warmed by the plate, its density decreases and a buoyant force arises which induces flow motion in the vertical or horizontal direction.
- The force is proportional to $(\rho - \rho_\infty) \times g$, therefore gravity must be considered.

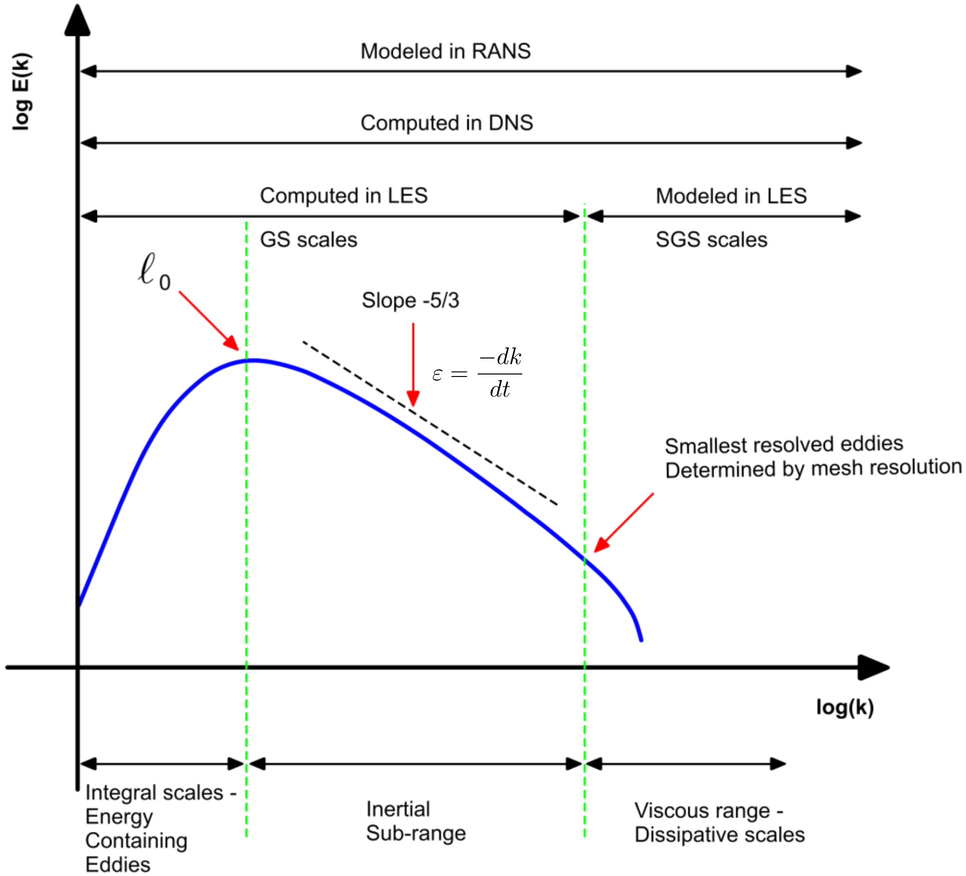
Turbulence modeling – Grid scales



- Turbulence modelling aims at predicting velocity and transported quantities fluctuations without calculating the complete turbulent flow pattern as a function of time.
- Everything below grid scales or sub-grid scales (SGS) is modelled or filtered.
- Therefore, if we want to capture all scales we need very fine meshes in the whole domain.

Bullet at Mach 1.5
Photo credit: Andrew Davidhazy. Rochester Institute of Technology.

Energy spectrum and energy cascade

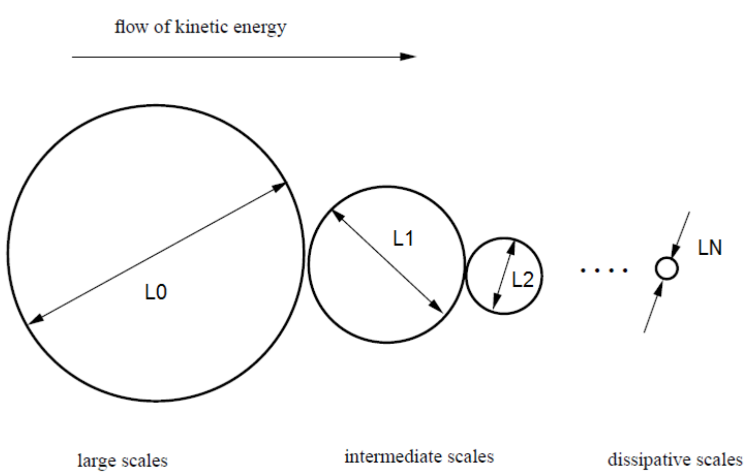


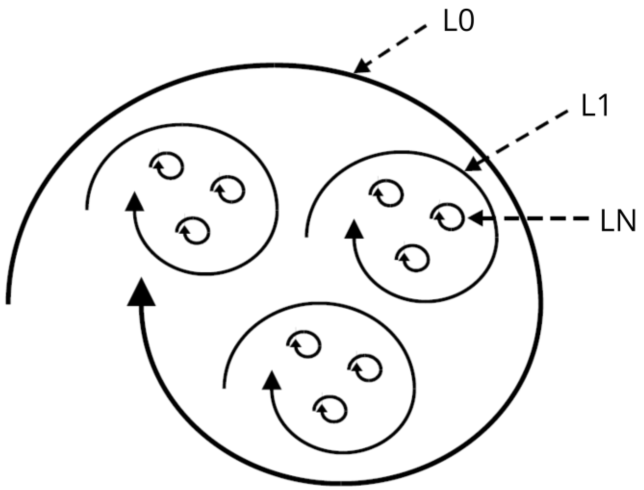
- Notice that this kind of graph is local. It will be different for each and every point in the domain.
- In the x axis the wave number is plotted,

$$\kappa = \frac{2\pi}{l}$$

- The turbulent power spectrum represents the distribution of the turbulent kinetic energy across the various length scales. It is a direct indication of how energy is dissipated with eddies size.
- The mesh resolution determines the fraction of the energy spectrum directly resolved.
- Eddies cannot be resolved down to the molecular dissipation limit.

Energy spectrum and energy cascade





- The energy-containing eddies are denoted by L0; L1 and L2 denotes the size of the eddies in the inertial subrange such that L2 < L1 < L0; LN is the size of the dissipative eddies.
- The large, energy containing eddies transfer energy to smaller eddies via vortex stretching.
- Smallest eddies convert kinetic energy into thermal energy via viscous dissipation.
- Large eddies derive their energy from the mean flow.
- The size and velocity of large eddies are on the order of the mean flow

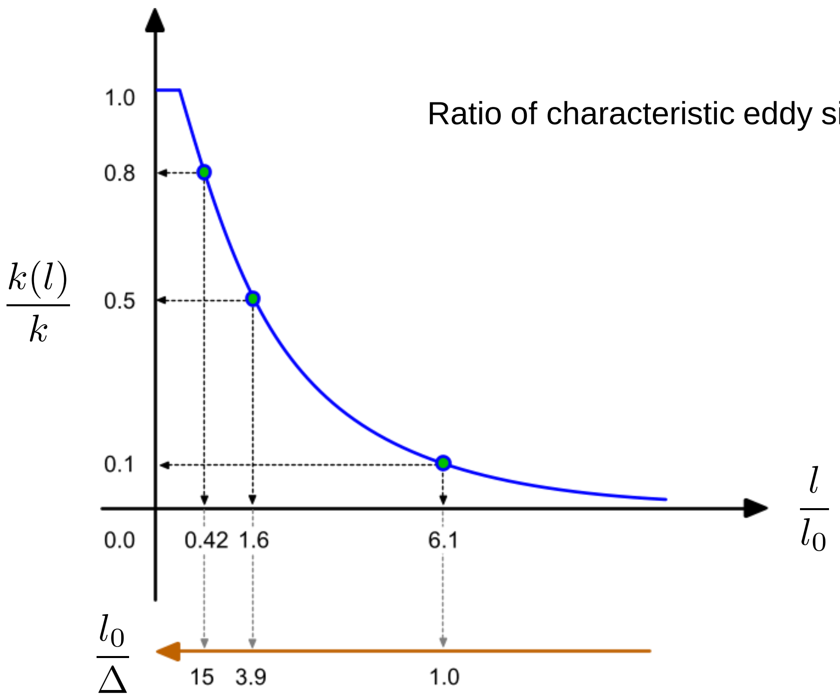
References:
S. Pope. Turbulent Flows. Cambridge University Press. 2014.

Energy spectrum and mesh resolution

- DNS simulations are quite expensive, they require a lot of grid points/cells in order to resolve all the turbulent scales.
- Thus, in a DNS simulation the gridding requirements scales proportional to $Re_T^{9/4}$ or approximately proportional to Re_T^3 for a single time step.
- And every time step should be sufficiently resolved in time (CFL condition less than 1, and the ideal value should be less than 0.5).
- An alternative to DNS, is the use of large eddy simulations (LES).
- In a good resolved LES simulation, we aim at resolving 80% of the turbulent kinetic energy.
- If the mesh requirements of a LES are too high, we can do a VLES (very large eddy simulation).
- In a VLES we aim at resolving 50% of the turbulent kinetic energy.
- If LES requirements are still high (which is the case for most of the industrial applications), we use RANS/URANS models.
- In RANS/URANS simulations the whole energy spectrum is modeled.
- The mesh spacing should be sufficiently to capture well integral scales l_0 and model/resolve the boundary layer.

Energy spectrum and mesh resolution

- From the previous discussion, we can see that the mesh resolution determines the fraction of the turbulent kinetic energy directly resolved.
- So, let us suppose that we want to resolve 80% of the turbulent kinetic energy *k(l)* in a LES simulation. Then, approximately 15 cells will be needed across the integral length scale l_0.
- In the same way, if you would like to solve 50% of the turbulent kinetic energy *k(l)* (VLES), you will need approximately 4 cells across the integral length scale l_0.



Cumulative turbulent kinetic energy against lengths scale of eddies

Ratio of integral length scale to grid length scale

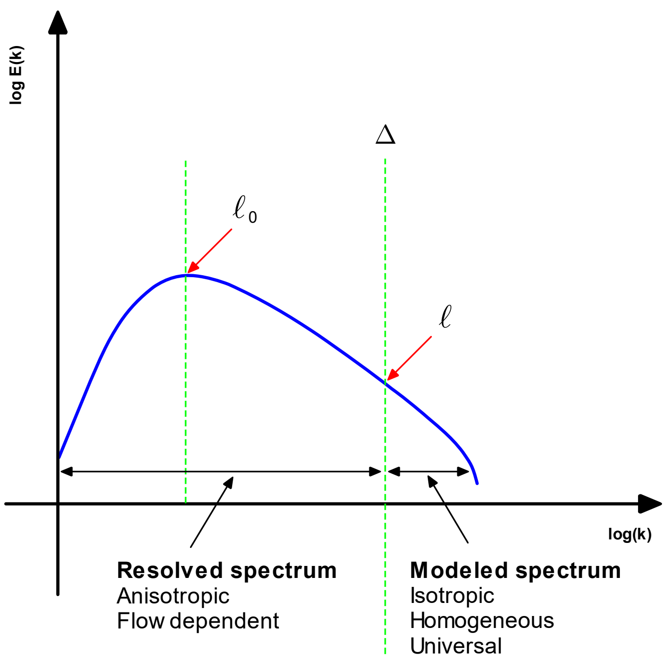
Ratio of characteristic eddy size to integral length scale

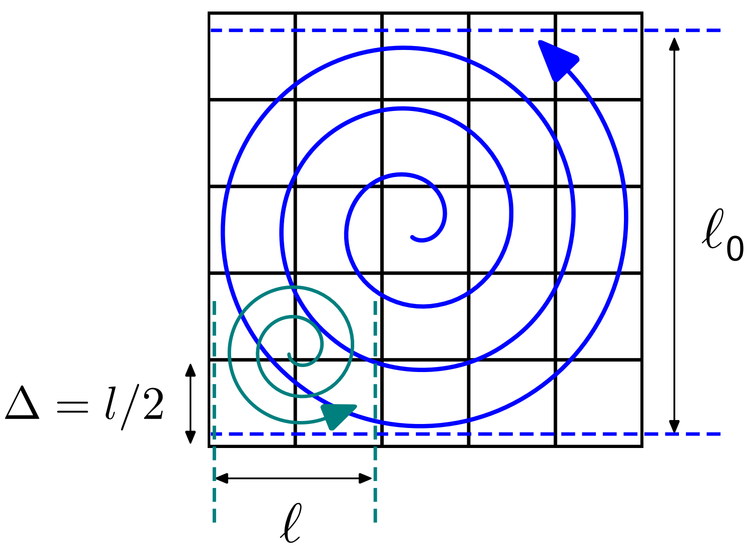
	l/l_0	l_0/Δ
$k(l) = 0.1k$ (10%)	6.1	1.0
$k(l) = 0.5k$ (50%)	1.6	3.9
$k(l) = 0.8k$ (80%)	0.42	15
$k(l) = 0.9k$ (90%)	0.16	38

Characteristics turbulent kinetic energy and length scales of the energy spectrum. For a rigorous explanation of these results, please refer to Turbulent Flows by S. Pope

Energy spectrum and mesh resolution

- In LES simulations, it is a good practice to have at least 10-15 cells across the integral length scale l_0.
- If you are interested in VLES, 4 to 6 cells per integral length scale l_0 are recommended (this also applies to RANS/URANS).
- To resolve an eddy with a length scale l (where l is the smallest scales that can be resolved with the grid or Δ), at least a couple of cells need to be used in each direction.
- Remember, eddies cannot be resolved down to the molecular dissipation limit (it is too expensive).





Integral length scale and grid length scale

- The integral length scale l_0 can be roughly estimate as follows,
  - Based on a characteristic length, such as the size of a bluff body or pipe diameter.
  - From correlations.
  - From experimental results.
  - From a precursor RANS simulation.

- Remember, turbulent kinetic energy peaks at integral length scale l_0.
- Therefore, these scales must be sufficiently resolved in LES/DES simulations, or capture (be able to track) in RANS/URANS simulations.
- After identifying the integral scales, try to cluster enough cells in the domain regions where you expect to find the integral scales (or large eddies).
- In other words, put sufficient cells in the wake or core of the flow.
- In RANS/URANS/VLES simulations, it is acceptable to use a minimum of 5 cells across integral length scales.
- LES simulations have higher requirements.

Integral length scale and grid length scale

- The integral length scales l_0 can be computed from a precursor RANS simulation as follows.
- You will need to use a two-equation model ($k - \epsilon$ family or $k - \omega$ family).
- Depending on the model selected, you can compute l_0 as follows,

$$l_0 = \frac{k^{1.5}}{\epsilon} \quad \text{or} \quad l_0 = \frac{k^{0.5}}{C_\mu \omega} \quad \text{where} \quad C_\mu = 0.09$$

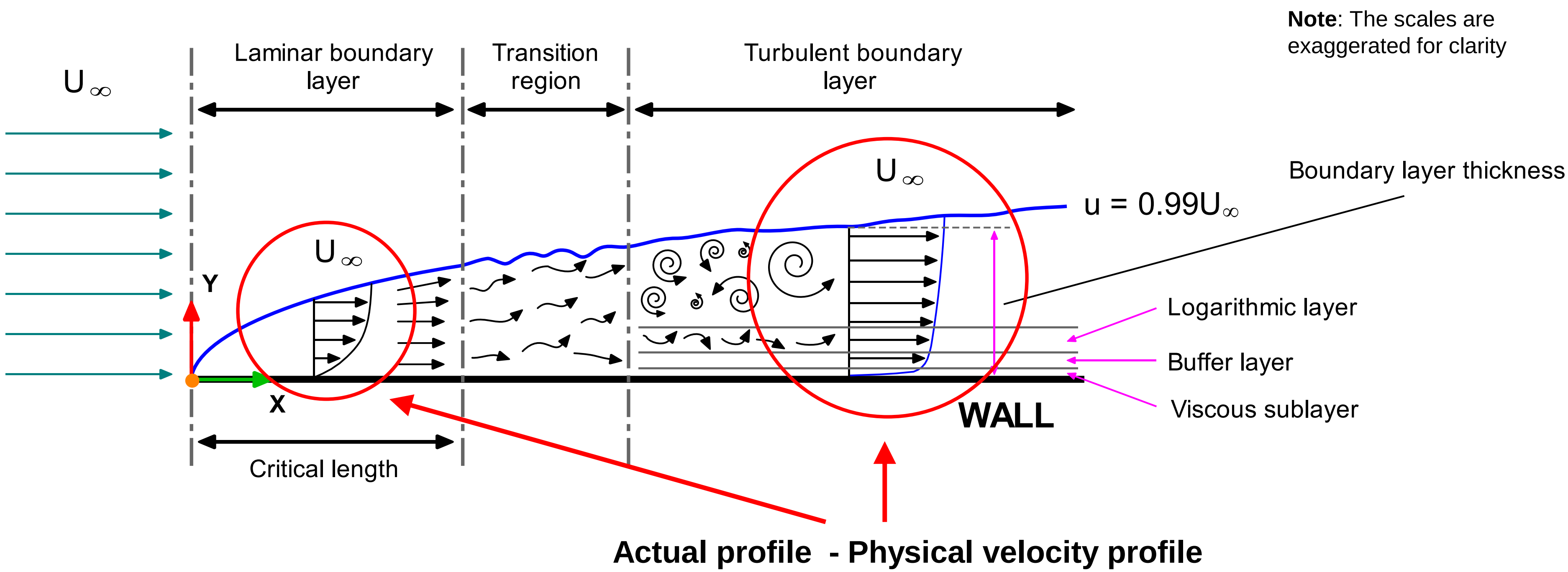
- The ratio of integral length scale to grid length scale R_l can be computed as follows,

$$R_l = \frac{l_0}{\Delta} \quad \text{where } \Delta \text{ can be approximated as follows } \Delta \approx \sqrt[3]{\text{cell volume}}$$

This approximation is accurate if the aspect ratios are modest (less than 1.2)

- The recommended value of R_l should be $R_l > 5 - 10$
- Where 5 should be considered the lowest limit of resolution (for RANS/URANS and VLES) and 10 is the desirable lower limit (for LES/DES).
- Higher values can be used if computer power and time constrains permits.
- This is a very rough estimate, which is likely problem dependent.
- Remember, in well resolved LES simulations equal mesh resolution should be provided in all directions.

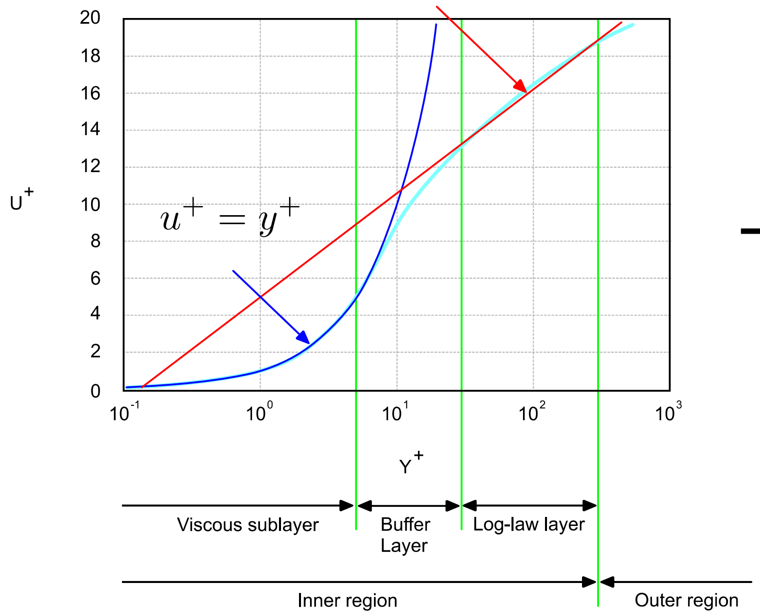
Turbulence near the wall - Law of the wall



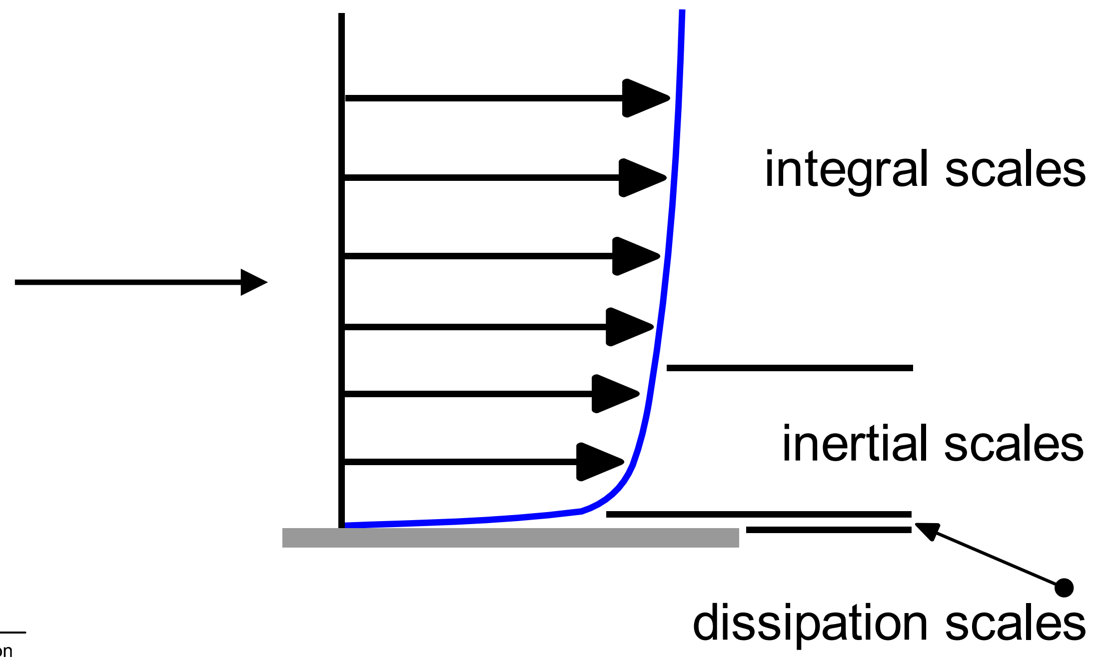
- Near walls, in the boundary layer, the velocity changes rapidly.
- In turbulence modeling in CFD, the most important zones are the viscous sublayer and the log-law layer.
- The buffer layer is the transition layer which we try to avoid as much as possible.
- Turbulence modeling in CFD requires different 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 near the wall - Law of the wall

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



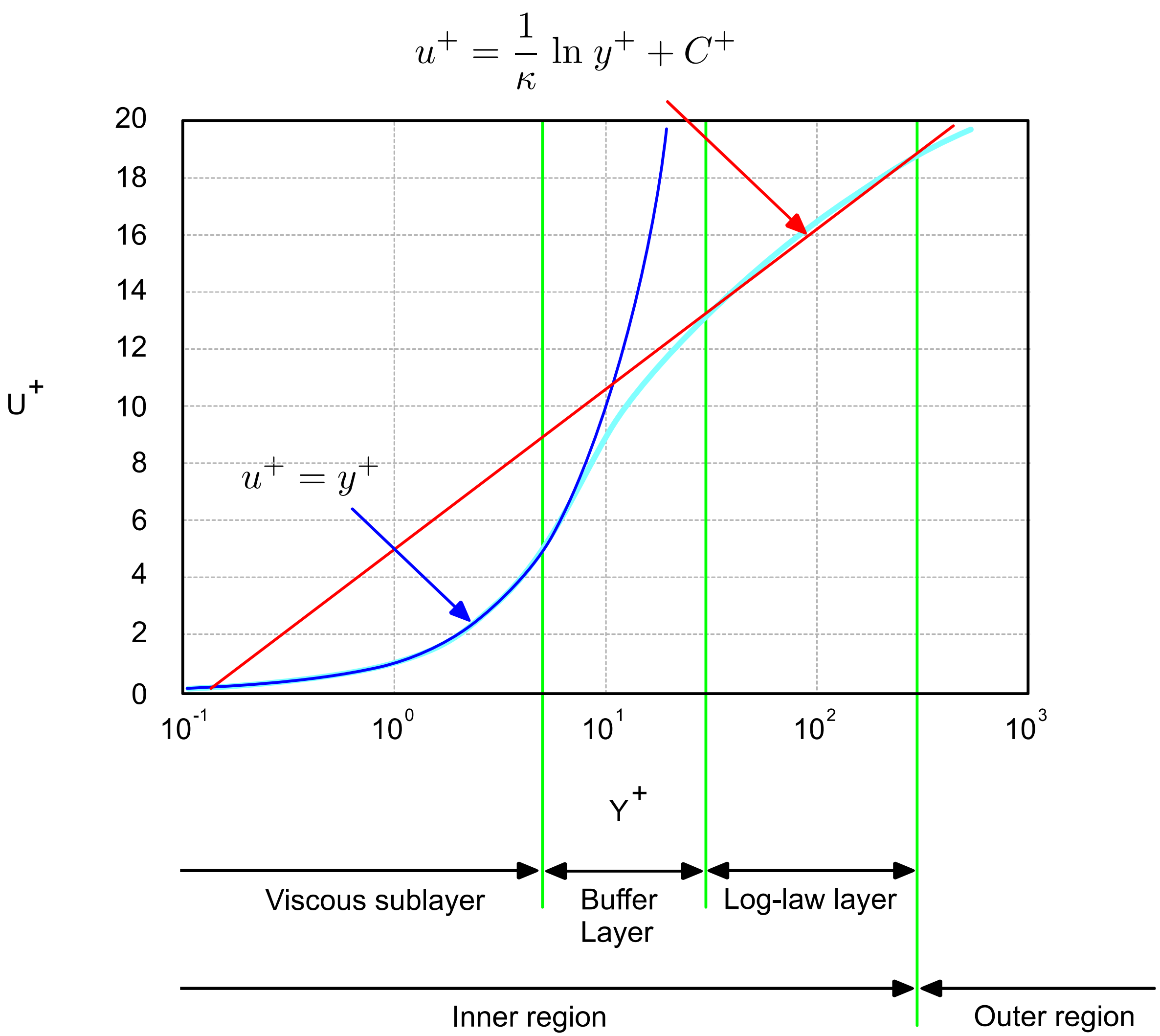
Non-dimensional profile



Actual profile - Physical velocity 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.

Turbulence near the wall - Law of the wall



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

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

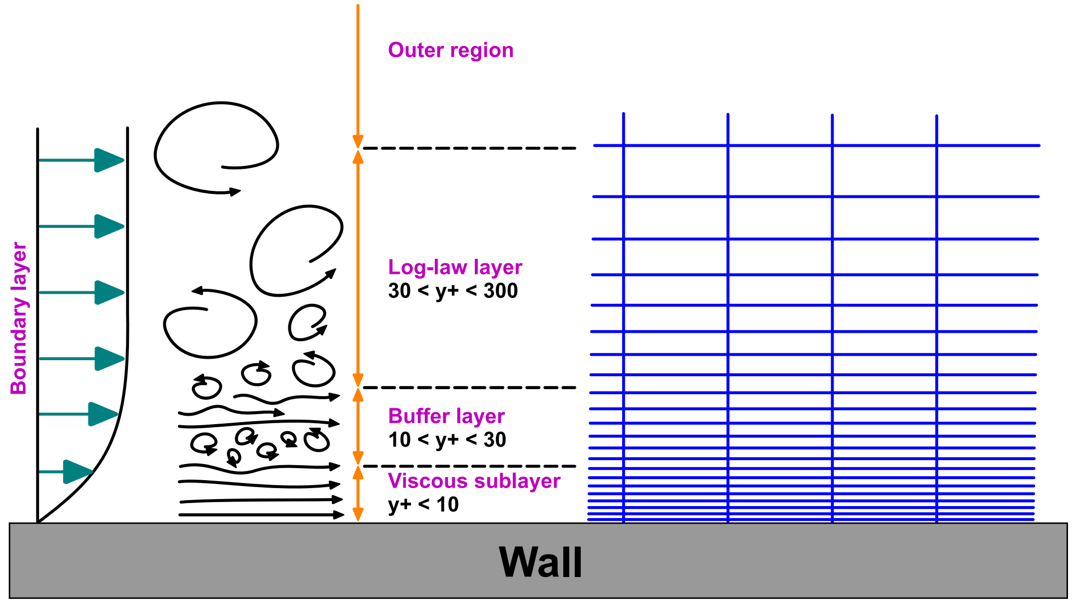
$$u^+ = \frac{U}{U_\tau}$$

Where y is the distance normal to the wall, U_τ is the shear velocity, and u^+ relates the mean velocity to the shear velocity

y^+ or wall distance units is a very important concept when dealing with turbulence modeling, remember this definition as we are going to use it a lot.

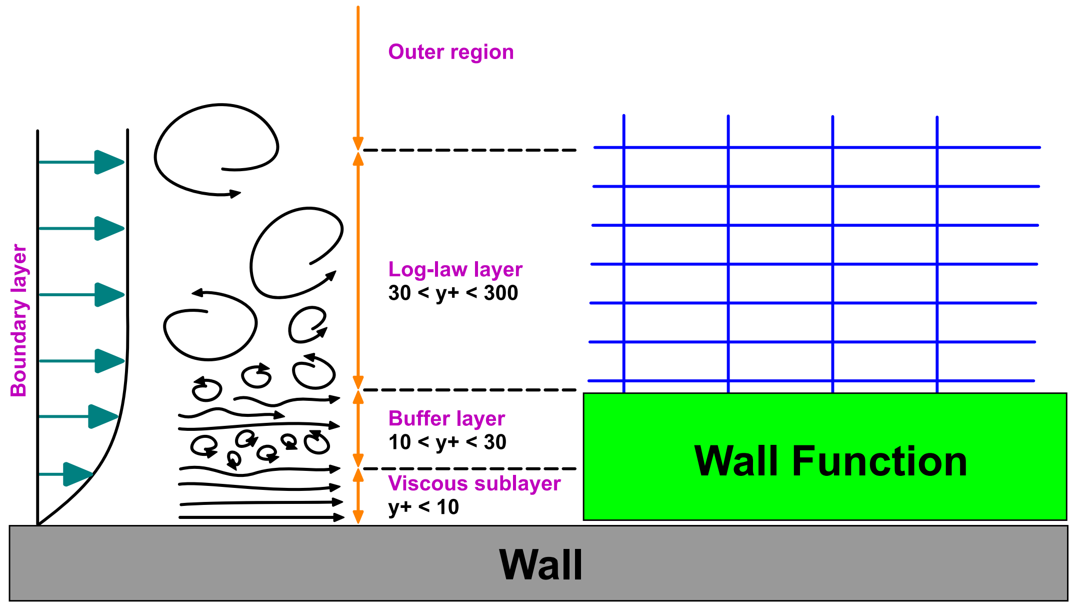
Near-wall treatment and wall functions

- 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 $y^+ < 10$.
- But for good accuracy, usually you will use 20 to 30 layers.
- This is the most accurate approach, but it is computationally expensive.



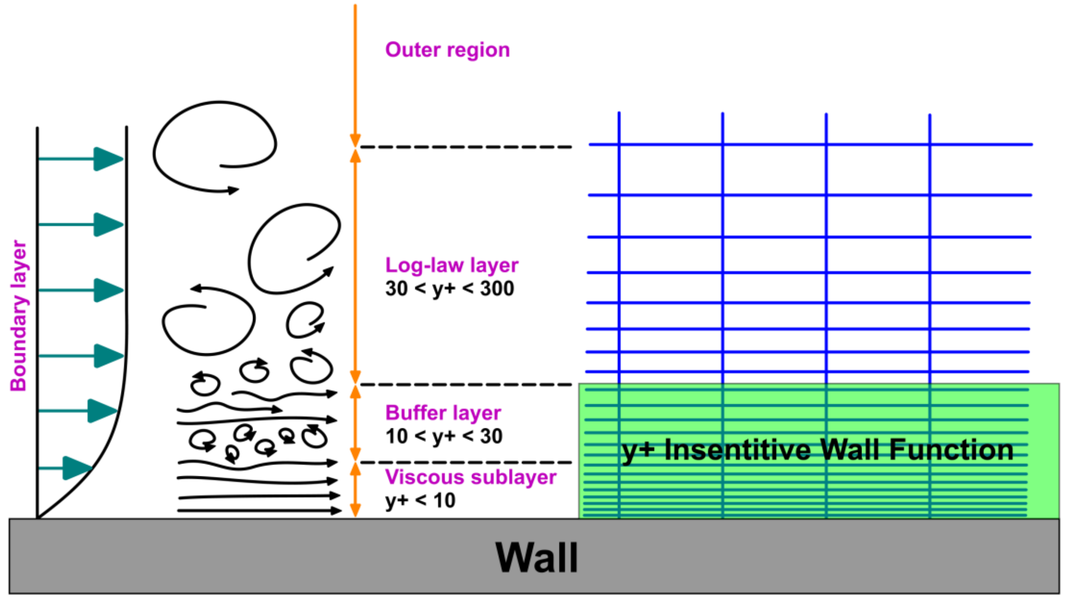
Near-wall treatment and wall functions

- 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 8 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.

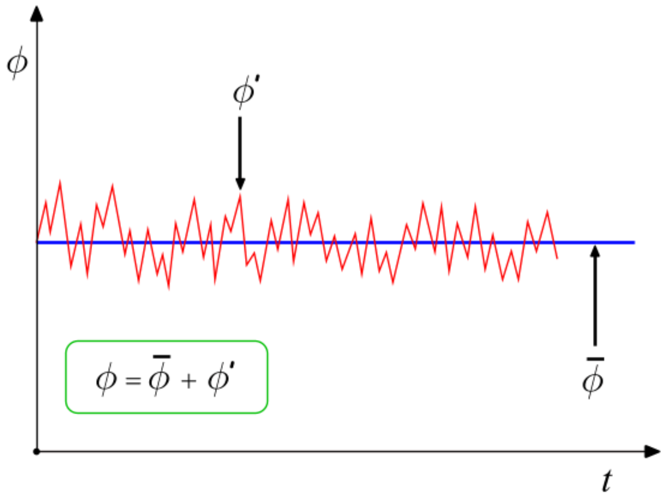


Near-wall treatment and wall functions

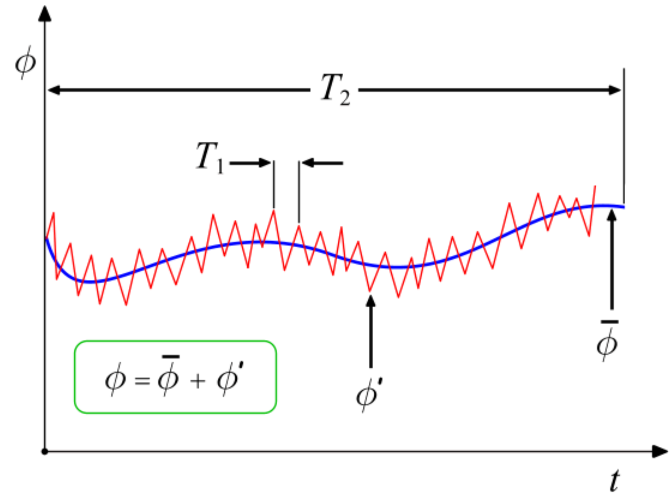
- Generally speaking, wall functions is the approach to use if you are more interested in the mixing in the outer region, rather than the forces on the wall.
- If accurate prediction of forces or heat transfer on the walls are key to your simulation (aerodynamic drag, turbomachinery blade performance, 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 nobody knows what is going on there.
- 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 $y^+ < 6$
- If you do not have any restrictions in the near-wall treatment, use wall functions.
- Wall functions can be used in RANS, DES and LES.
- If you are doing LES, it is highly recommended to use wall functions. Otherwise, your meshing requirements will be very similar to DNS.

Removing small scales

Steady mean value



Unsteady mean value



$$\phi(\mathbf{x},t) = \underbrace{\bar{\phi}(\mathbf{x})}_{mean\,value} + \underbrace{\phi'(\mathbf{x},t)}_{fluctuating\,part}$$

$$\phi(\mathbf{x},t) = \underbrace{\bar{\phi}(\mathbf{x},t)}_{mean\,value} + \underbrace{\phi'(\mathbf{x},t)}_{fluctuating\,part}$$

- We have seen that turbulent flows are characterize by instantaneous fluctuations of velocity, pressure, and all transported quantities.
- In most engineering applications is not of interest resolving the instantaneous fluctuations.
- To remove the instantaneous fluctuations or small scales, two methods can be used:
    - Reynolds averaging
    - Filtering
- Both methods introduce additional terms that must be modeled for closure.

Removing small scales

- Two methods can be used to eliminate the need to resolve the small scales:
  - Reynolds averaging (RANS/URANS):
    - All turbulence scales are modeled.
    - Can be 2D and 3D.
    - Can be steady or unsteady.
  - Filtering (LES/DES):
    - Resolves large eddies.
    - Models small eddies.
    - Intrinsically 3D and unsteady.

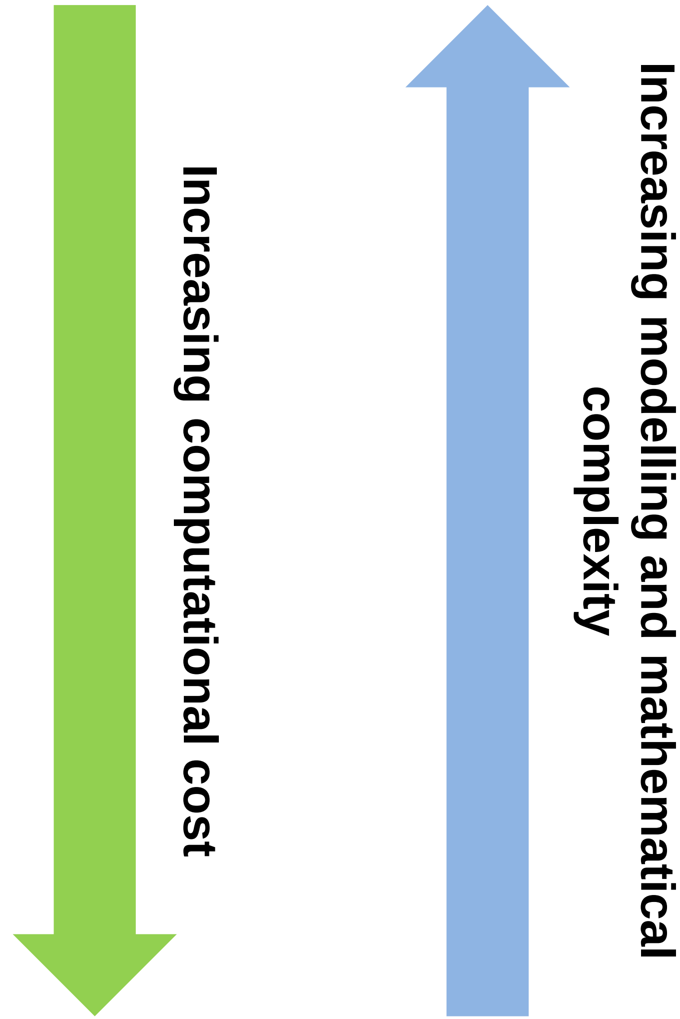
- Both methods introduce additional terms in the governing equations that must be modeled.
- The final goal of turbulence modeling is to find the closure equations to model the additional terms (usually a stress tensor).

Introduction to turbulence modeling

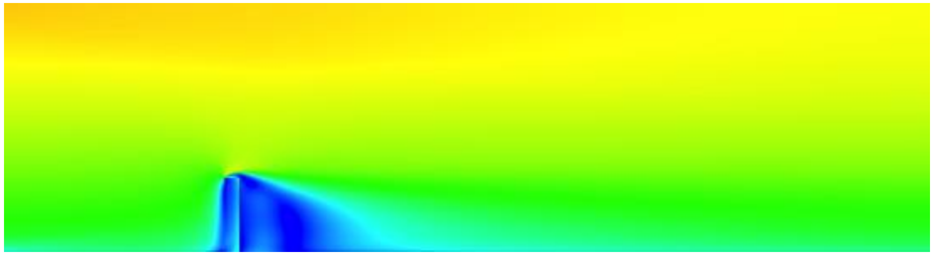
Overview of turbulence modeling approaches

MODEL
RANS **R**eynolds-**A**veraged **N**avier-**S**tokes equations
URANS **U**nsteady **R**eynolds-**A**veraged **N**avier-**S**tokes equations
SAS **S**cale **A**daptive **S**imulations
DES **D**etached **E**ddy **S**imulations
LES **L**arge **E**ddy **S**imulations
DNS **D**irect **N**umerical **S**imulations

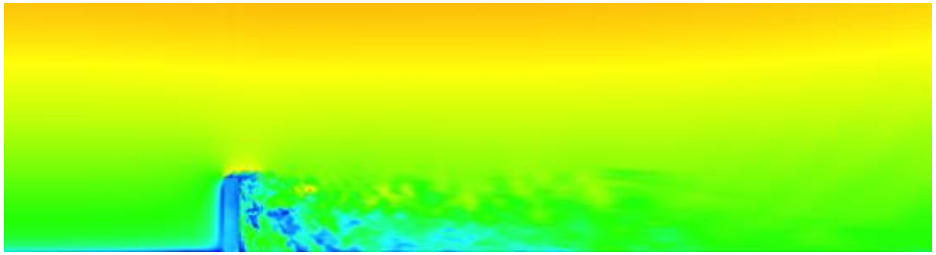
SRS
Scale-**R**esolving **S**imulations



Overview of turbulence modeling approaches



RANS



LES – Instantaneous field

RANS/URANS	DES/LES	DNS
• Solve the time-average NSE. • All turbulent spatial scales are modeled. • Many models are available. One equation models, two equation models, Reynolds stress models, transition models, and so on. • This is the most widely approach for industrial flows. • Unsteady RANS (URANS), use the same equations as the RANS but with the transient term retained. • It can be used in 2D and 3D cases.	• Solve the filtered unsteady NSE. • Sub-grid scales (SGS) are filtered, grid scales (GS) are resolved. • Aim at resolving the temporal scales, hence requires small time-steps. • For most industrial applications, it is computational expensive. However, thanks to the current advances in parallel and scientific computing it is becoming affordable. • Many models are available. • It is intrinsically 3D and asymmetric.	• Solves the unsteady laminar NSE. • Solves all spatial and temporal scales; hence, requires extremely fine meshes and small time-steps. • No modeling is required. • It is extremely computational expensive. • Not practical for industrial flows. • It is intrinsically 3D and asymmetric.

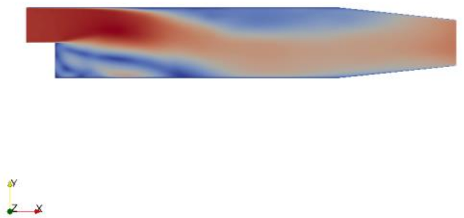
Overview of turbulence modeling approaches

LES – Smagorinsky – nut
http://www.wolfdynamics.com/training/turbulence/image9.gif

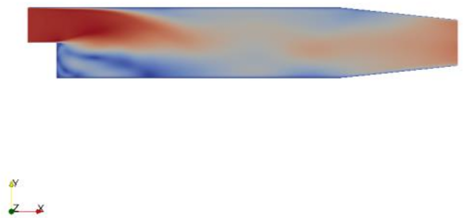
URANS – SA – nut
http://www.wolfdynamics.com/training/turbulence/image10.gif

URANS – kw – nut
http://www.wolfdynamics.com/training/turbulence/image11.gif

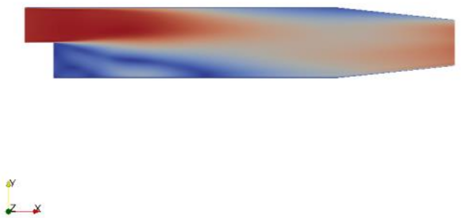
LES – U mean



URANS SA – U mean



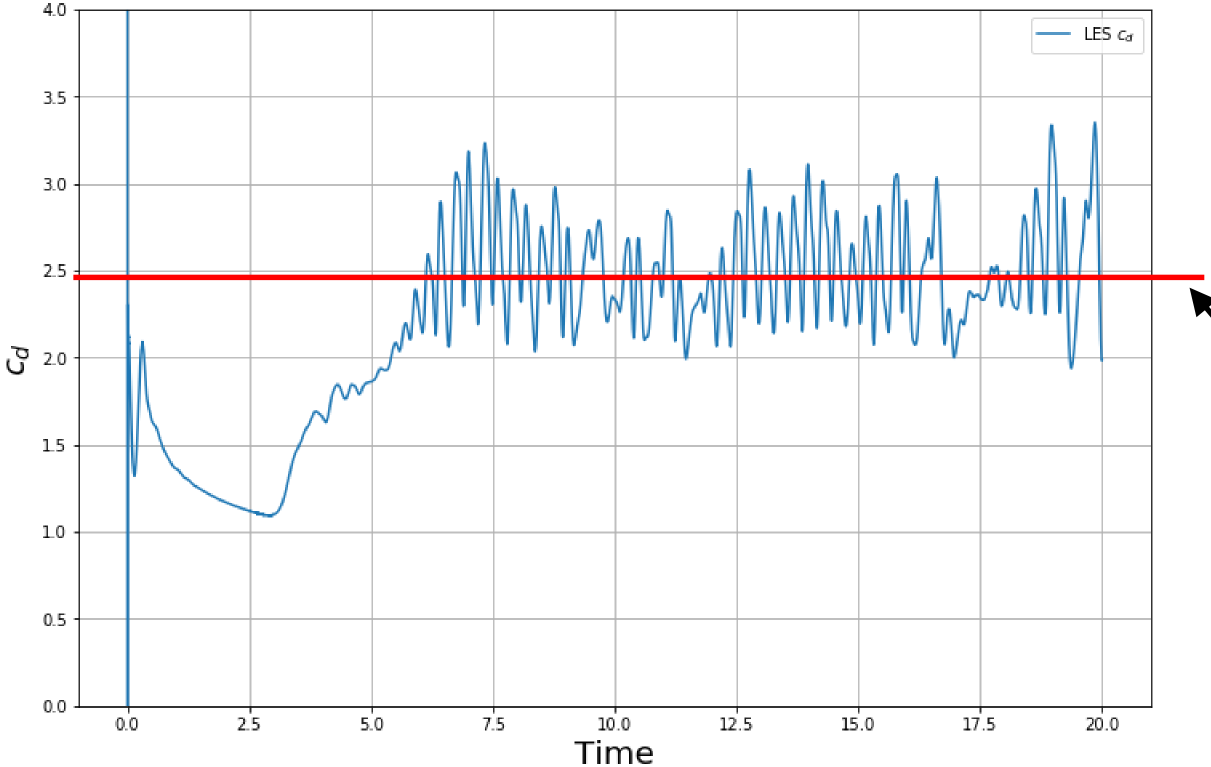
URANS kw – U mean



- The instantaneous field of each method might be slightly different, but when we average the solution, we obtain similar fields.
- All cases use the same mesh and have approximately the same computational overload.
- Unsteadiness makes turbulence simulations long.

Overview of turbulence modeling approaches

- Most of the times we are not interested in the unsteadiness evolution.
- Therefore, we must compute the statistics of the time evolution of the quantities of interest.
- On the other hand, if we use RANS models (steady simulations) we will get the mean results, which is ok for most engineering applications.
- Sometimes the iterative evolution of a RANS simulation might fluctuate; therefore, it is advice to compute the descriptive statistics in a given window.



Mean value	2.495
Standard deviation	0.286
Variance	0.0822
RMS	2.512

Mean value
The initial transient was not considered to compute the descriptive statistics

Turbulence modeling – Starting equations

NSE

$$
\left\{
\begin{aligned}
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) &= 0, \\
\frac{\partial (\rho \mathbf{u})}{\partial t} + \nabla \cdot (\rho \mathbf{u}\mathbf{u}) &= -\nabla p + \nabla \cdot \tau, \\
\frac{\partial (\rho e_t)}{\partial t} + \nabla \cdot (\rho e_t \mathbf{u}) &= \nabla \cdot q - \nabla \cdot (p \mathbf{u}) + \tau \mathbf{:} \nabla \mathbf{u},
\end{aligned}
\right.
$$

$$+$$

Additional closure equations for the 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!

Incompressible RANS equations

- Let us derive the Reynolds average equations for an incompressible flow.
- This is our starting point,

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

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

- And we want to arrive to the following RANS equations,

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

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

Note: if you drop the time derivative, we call it RANS (steady equations), and if you retain the time derivative, we call it URANS (unsteady equations).

Incompressible RANS equations

- To derive the incompressible RANS equations we need to apply Reynolds average to the primitive variables of the starting equations, as follows,

$$\phi(\mathbf{x},t) = \bar{\phi}(\mathbf{x},t) + \phi'(\mathbf{x},t)$$

- Let us recall the following averaging rules,

$$
\begin{aligned}
\bar{\phi'} &= 0, \\
\phi &= \bar{\phi}, \\
\bar{\phi} = \overline{\bar{\phi} + \phi'} &= \bar{\bar{\phi}}, \\
\overline{\frac{\partial \phi}{\partial x}} &= \frac{\partial \bar{\phi}}{\partial x}, \\
\overline{\phi + \varphi} &= \bar{\phi} + \bar{\varphi}\,, \\
\overline{\bar{\phi}\varphi} = \bar{\bar{\phi}}\bar{\varphi} &= \bar{\phi}\bar{\varphi}, \\
\overline{\bar{\phi}\bar{\varphi}} &= \bar{\phi}\bar{\varphi},
\end{aligned}
$$

$$
\begin{aligned}
\overline{\bar{\phi}\varphi'} = \bar{\phi}\bar{\varphi'} &= 0, \\
\overline{\phi\varphi} &= \overline{(\bar{\phi} + \phi')(\bar{\varphi} + \varphi')} \\
&= \overline{\bar{\phi}\bar{\varphi} + \bar{\phi}\varphi' + \bar{\varphi}\phi' + \phi'\varphi'} \\
&= \overline{\bar{\phi}\bar{\varphi}} + \overline{\bar{\phi}\varphi'} + \overline{\bar{\varphi}\phi'} + \overline{\phi'\varphi'} \\
&= \bar{\phi}\bar{\varphi} + \overline{\phi'\varphi'}, \\
\overline{\phi'^2} &\neq 0, \\
\overline{\phi'\varphi'} &\neq 0.
\end{aligned}
$$

Incompressible RANS equations

- Let us recall the Reynolds decomposition for the primitive variables of the starting equations,

$$\mathbf{u}(\mathbf{x},t) = \bar{\mathbf{u}}(\mathbf{x}) + \mathbf{u}'(\mathbf{x},t),$$
$$p(\mathbf{x},t) = \bar{p}(\mathbf{x}) + p'(\mathbf{x},t)$$

- By substituting the previous equations into our starting equations, using the previous averaging rules, and doing some algebra, we arrive to the incompressible URANS/RANS equations,

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

Reynolds stress tensor

$$\nabla \cdot (\bar{\mathbf{u}}) = 0$$
$$\frac{\partial \bar{\mathbf{u}}}{\partial t} + \nabla \cdot (\bar{\mathbf{u}}\bar{\mathbf{u}}) = \frac{-\nabla \bar{p}}{\rho} + \nu \nabla^2 \bar{\mathbf{u}} - \frac{1}{\rho} \nabla \cdot \tau^R$$

Notice that all quantities are averaged (the overbar over the field quantities)

Incompressible RANS equations

- The RANS equations are very similar to the starting equations.
- The only difference is that all quantities are averaged (the overbar over the field quantities).
- And the appearance of Reynolds stress tensor τ^R

$$\tau^R = -\rho\left(\overline{\mathbf{u}'\mathbf{u}'}\right) = -\begin{pmatrix} \rho\overline{u'u'} & \rho\overline{u'v'} & \rho\overline{u'w'} \\ \rho\overline{v'u'} & \rho\overline{v'v'} & \rho\overline{v'w'} \\ \rho\overline{w'u'} & \rho\overline{w'v'} & \rho\overline{w'w'} \end{pmatrix}$$

- The RANS approach to turbulence modeling requires the Reynolds stresses to be appropriately modeled.
- It is possible to derive its own governing equations (six new equations as the tensor is symmetric), but it is much simpler to model this term.
- Probably this is the most physically sound RANS model (RSM or Reynolds stress model) as it avoids the use of the Boussinesq hypothesis that we will study next.

Incompressible RANS equations

- A common 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,

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

- In the previous equation, $\bar{\mathbf{D}}^R = \frac{1}{2}(\nabla\overline{\mathbf{u}} + \nabla\overline{\mathbf{u}}^{\mathrm{T}})$ denotes the strain-rate tensor.
- $\mathbf{I}$ is the identity matrix.
- μ_T is the turbulent eddy viscosity.
- $k = \frac{1}{2}\overline{\mathbf{u}' \cdot \mathbf{u}'}$ or $k = \frac{1}{2}\left(\overline{u'^2} + \overline{v'^2} + \overline{w'^2}\right)$ is the turbulent kinetic energy.
- <u>At the end of the day we want to determine the turbulent eddy viscosity</u>. Each turbulence model will compute this quantity in a different way.
- The turbulent eddy viscosity is not a fluid property, it is a property needed by the turbulence model.

Incompressible RANS equations

- In the Reynolds decomposition, the quantities of interest can be averaged as follows.
- For steady RANS we can use time averaging as follows,

$$\bar{\phi}(\mathbf{x}) = \lim_{T \to +\infty} \frac{1}{T} \int_t^{t+T} \phi(\mathbf{x}, t)\, \mathrm{d}t \qquad \text{where} \qquad \phi(\mathbf{x}, t) = \bar{\phi}(\mathbf{x}) + \phi'(\mathbf{x}, t)$$

- For unsteady RANS or URANS we can use ensemble averaging as follows,

$$\bar{\phi}(\mathbf{x}, t) = \lim_{\mathcal{N} \to \infty} \frac{1}{\mathcal{N}} \sum_{i=1}^{\mathcal{N}} \phi(\mathbf{x}, t) \qquad \text{where} \qquad \phi(\mathbf{x}, t) = \bar{\phi}(\mathbf{x}, t) + \phi'(\mathbf{x}, t)$$

- For ensemble average the number of experiments of the ensemble must be large enough to eliminate effects of fluctuations. This type of averaging can be applied to any flow (steady or unsteady).

Incompressible RANS equations

- We just outlined the incompressible RANS.
- The compressible RANS equations are similar, to derive them we use Favre average (which can be seen as a mass-weighted averaging).
- If we drop the time derivative in the governing equations we are dealing with steady turbulence.
- On the other hand, if we keep the time derivative we are dealing with unsteady turbulence and we should use ensemble averaging.
- The derivation of the LES equations is very similar, but instead of using averaging we filter the equations in space, and we solve the temporal scales.
- LES/DES models are intrinsically unsteady and three-dimensional.
- We will briefly address filtering and the LES equations.
- Remember, the main goal of the turbulence models is to find the turbulent eddy viscosity to model the Reynolds stress tensor.
- Let us take a look at the governing equations of the $k - \omega$ RANS model.

$k-\omega$ Turbulence model overview

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

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

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

- These are the closure equations of the turbulence problem using Reynolds average.
- These are not physical properties. They kind of represent the generation and destruction of turbulence.
- Recall that $k=\frac{1}{2}\left(\overline{u'^2}+\overline{v'^2}+\overline{w'^2}\right)$
- In the $k-\omega$ model, the turbulent eddy viscosity is computed as follows,

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

$k-\omega$ Turbulence model overview

- In its simplest form, the following equations are applied in the first cell center of the logarithmic layer,

$$U = \frac{u_\tau}{\kappa} ln y + C \qquad k = \frac{u_\tau^2}{\sqrt{\beta^*}} \qquad \omega = \frac{u_\tau}{\sqrt{\beta^*} k y}$$

- These equations are here referred to as the standard wall function for the $k-\omega$ method. They relate the local wall shear stress (through the shear velocity u_τ) to the mean velocity, turbulence kinetic energy k, and rate of dissipation ω.
- A small implicit system must be solved in an iterative way in order to compute the friction velocity u_τ.
- First, we compute the friction velocity, then we compute k, and then we find ω.
- By adopting a mesh where the first cell center is located in the logarithmic layer, it is possible to use the law-of-the-wall to specify the boundary condition for the dependent variables $(u, k, \text{and}\, \omega)$ so that the turbulence model equations are not solved close to the wall.
- Each turbulence model formulation has different wall function equations.

$k-\omega$ Turbulence model overview

- The boundary and initial conditions recommended for this model are,

$$\frac{U_\infty}{L} < \omega_{farfield} < \frac{10U_\infty}{L} \qquad \frac{10^{-5}U_\infty^2}{Re_L} < k_{farfield} < \frac{0.1U_\infty^2}{Re_L}$$

where *L* is the Length of the domain.

- Alternatively, you can use the following equations:

$$\omega_{farfield} = \frac{\rho k}{\mu}\left(\frac{\mu_t}{\mu}\right)^{-1} \qquad k_{farfield} = \frac{3}{2}(UI)^2$$

where μ_t/μ is the viscosity ratio and $I = u'/\bar{u}$ is the turbulence intensity (recommended values in the next slide).

$k - \omega$ Turbulence model overview

- If you are totally lost, you can use these reference values. They work most of the times, but it is a good idea to have some experimental data or a better initial estimate.

	Low	Medium	High
I	1.0 %	5.0 %	10.0 %
μ_t/μ	1	10	100

where μ_t/μ is the viscosity ratio and $I = u'/\bar{u}$ is the turbulence intensity.

- By the way, use these guidelines for external aerodynamics only.
- Similar estimates are available for internal flows.

$k - \omega$ Turbulence model free-stream initial conditions

- At the walls, you can use the following values

$$\omega_{wall} = 10 \frac{6\nu}{\beta y^2} \qquad \qquad k_{wall} = 0$$

where $\beta = 0.075$ and y is the distance to the first cell center normal to the wall.

- The values are the same stated in the original reference.
- If you use a different turbulence model, just refer to the original reference or visit the following site:

  https://turbmodels.larc.nasa.gov/

- You can also visit our website and use the calculator to estimate turbulence quantities,

  http://www.wolfdynamics.com/tools.html?id=110

$k-\omega$ Turbulence model boundary conditions at the walls

- As for the free-stream boundary conditions, you need to give the boundary conditions for the near-wall treatment and domain boundaries.
- When it comes to near-wall treatment, you have three options:
  - Use wall functions (values up to 600 are acceptable if they do not cover more that 20% of the surface and are not located in critical regions):

$$30 \leq y^+ \leq 300$$

  - Use y^+ insensitive wall functions, this only applies with the $k-\omega$ SST model:

$$1 \leq y^+ \leq 300$$

  - Resolve the boundary layer (no wall functions):

$$y^+ \leq 6$$

$k-\omega$ Turbulence model boundary conditions at the walls

- If you are planning to use wall functions ($30 \leq y^+ \leq 300$), the following values are good choices.
- For k you can use the wall function **kqRWallFunction** with the following value:

$$k_{wall} = k \qquad \text{or} \qquad k_{wall} = 1e-10$$

- For ω you can use the wall function **omegaWallFunction** with the following value:

$$\omega_{wall} = 10 \frac{6\nu}{\beta y^2} \qquad \text{or the free-stream value}$$

where $\beta = 0.075$ and y is the distance to the first cell center normal to the wall.

- To avoid potential erroneous arithmetic operations, it is recommended to use a small value instead of 0.

$k - \omega$ Turbulence model boundary conditions at the walls

- If you are planning to use y+ insensitive wall functions ($1 \leq y^+ \leq 300$), the following values are good choices.
- For k you should use the wall function **kqRWallFunction** or **kLowReWallFunction** with the following value:

$$k_{wall} = k \qquad \text{or} \qquad k_{wall} = 1e-10$$

- For ω you should use the wall function **omegaWallFunction** with the following value:

$$\omega_{wall} = 10 \frac{6\nu}{\beta y^2} \qquad \text{or the free-stream value}$$

where $\beta = 0.075$ and y is the distance to the first cell center normal to the wall.

- To avoid potential erroneous arithmetic operations, it is recommended to use a small value instead of 0.

$k - \omega$ Turbulence model boundary conditions at the walls

- If you are planning not to use wall functions ($y^+ \leq 6$), the following values are good choices.
- For k you should use **fixedValue** or **kLowReWallFunction** with the following value:

$$k_{wall} = 1e - 10$$

- For ω you should use **fixedValue** with the following value:

$$\omega_{wall} = 10 \frac{6\nu}{\beta y^2}$$

- You can also use **fixedValue** with a large value.
- To properly resolve the boundary layer, it is recommended to cluster at least 10 layers close to the walls.
- To avoid potential erroneous arithmetic operations, it is recommended to use a small value instead of 0.

Typical wall functions boundary conditions

- Remember, you can only use wall functions if the primitive patch (the patch type defined in the `boundary` dictionary), is of type **wall**.

Field	Wall functions – High RE	Resolved BL – Low RE
nut	**nut(–)WallFunction*** or **nutUSpaldingWallFunction**** (with 0 or a small number)	**nutUSpaldingWallFunction**** or **nutLowReWallFunction** or **fixedValue** (with 0 or a small number)
k, q, R	**kqRWallFunction** or **kLowReWallFunction** (with inlet value, 0, or a small number)	**kLowReWallFunction** or **fixedValue** (with inlet value, 0, or a small number)
epsilon	**epsilonWallFunction** (with inlet value)	**epsilonWallFunction** (with inlet value) or **zeroGradient** or **fixedValue** (with 0 or a small number)
omega	**omegaWallFunction** (with a large number)	**omegaWallFunction**** or **fixedValue** (both with a large number)
nuTilda	–	**fixedValue** (with 0 or a small number)

* `$WM_PROJECT_DIR/src/TurbulenceModels/turbulenceModels/derivedFvPatchFields/wallFunctions/nutWallFunctions`

** For y^+ insensitive wall functions (continuous wall functions)

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.

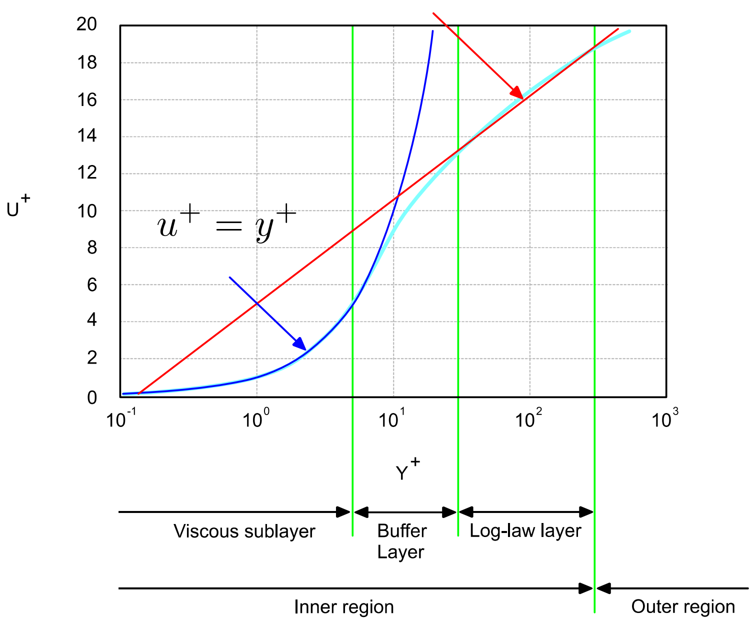
Final remarks on RANS turbulence models

- In terms of meshing requirements, RANS simulations are not computational expensive.
- Therefore, it is recommended to use a RANS approach for steady simulations.
- If you are conducting URANS simulations (unsteady runs), low-RE meshes can impose strict time-step restrictions. Therefore, high-RE meshes are recommended.
- Nevertheless, you can use large CFL numbers in URANS simulations.
- If you are dealing with thermal boundary layers, it is recommended to use a low-RE approach.
- Remember, avoid as much as possible the buffer layer $10 < y^+ < 30$
- Use RANS simulations as starting point for LES/DES.

Turbulence near the wall - Law of the wall

- Let us revisit the definition of y^+.
- Remember, y^+ or wall distance units normal to the wall is a very important concept when dealing with wall bounded turbulence modeling, remember this definition as it is used a lot.

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



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

$$U_\tau = \sqrt{\frac{\tau_w}{\rho}} \longleftarrow \text{wall shear stresses}$$

$$u^+ = \frac{U}{U_\tau}$$

Where y is the distance normal to the wall, U_τ is the shear velocity, and u^+ relates the mean velocity to the shear velocity

y^+ and wall distance units

y^+ wall distance units normal to the wall

- We never know a priori the y^+ value (because we do not know the friction velocity).
- What we usually do is to run the simulation for a few time-steps or iterations, and then we get an estimate of the y^+ value.
- After determining where we are in the boundary layer (viscous sub-layer, buffer layer or log-law layer), we take the mesh as a good one or we modify it if is deemed necessary.
- It is an iterative process and it can be very time consuming, as it might require remeshing and rerunning the simulation.
- Have in mind that it is quite difficult to get a uniform y^+ value at the walls.
- Try to get a y^+ <u>mean</u> value as close as possible to your target.
- Also, check that you do not get very high maximum values of y^+ (more than a 1000)
- Values up to 300 are fine. Values larger that 300 and up to a 1000 are acceptable is they do not covert a large surface (no more than 10% of the total wall area), or they are not located in critical zones.
- Use common sense when accessing y^+ value.

y^+ and wall distance units

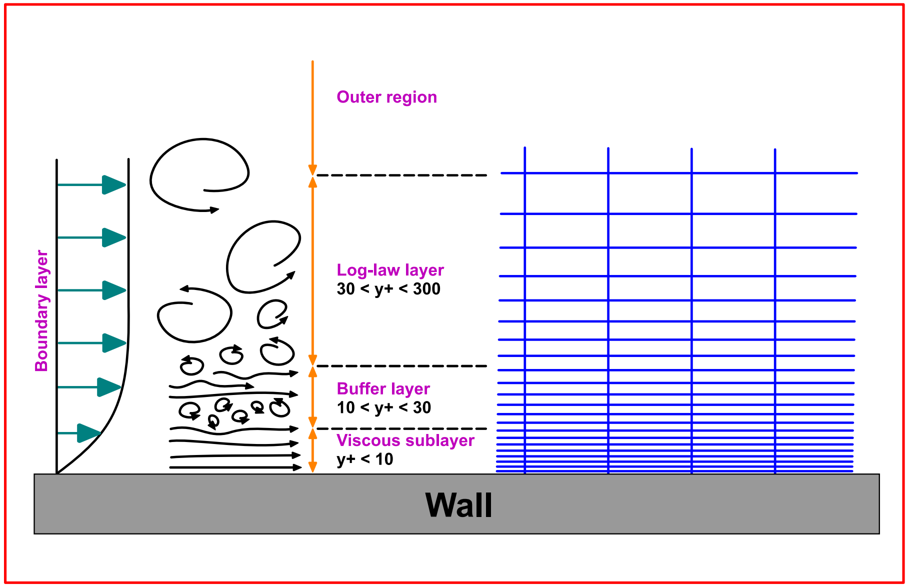
Estimating normal wall distance

- At meshing time, to estimate the normal wall distance to the first cell center, we use the well know y^+ definition,

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

- Where we set a target y^+ value and then we isolate the quantity y (normal wall distance to the first center). This will be distance that we will use when generating the boundary layer mesh.

- So if you choose a low y^+, you will have a mesh that is clustered towards the wall. And if you choose a large y^+ value, you will have a coarse mesh towards the walls.



Estimating normal wall distance

- At meshing time, to estimate the normal wall distance to the first cell center, we use the well know y^+ definition,

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

- The problem is that at meshing time we do not know the value of the shear velocity,

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

- So, how do we get an initial estimate of this quantity?

Estimating normal wall distance

- To get an initial estimate of the distance from the wall to the first cell center y, without recurring to a precursor simulation, you can proceed as follows,

1. $$Re = \frac{\rho \times U \times L}{\mu}$$

2. $$C_f = 0.058 \times Re^{-0.2}$$ (Skin friction coefficient of a flat plate, there are similar correlations for pipes)

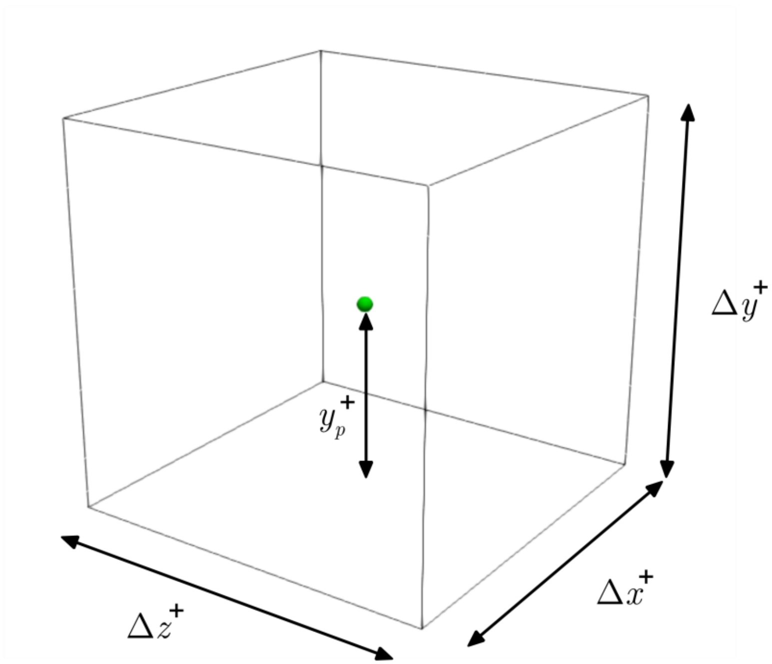
3. $$\tau_w = \frac{1}{2} \times C_f \times \rho \times U_\infty^2$$

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

5. $$y = \frac{\mu \times y^+}{\rho \times U_\tau}$$ Your desired value

- You will find a simple calculator for the wall distance estimation in the following link: http://www.wolfdynamics.com/tools.html?id=2

Wall distance units



- Similar to y^+, the wall distance units can be computed in the stream-wise (Δx^+) and span-wise (Δz^+) directions.
- The wall distance units in the stream-wise and span-wise directions can be computed as follows:

$$\Delta x^+ = \frac{U_\tau \Delta x}{\nu} \qquad \Delta z^+ = \frac{U_\tau \Delta z}{\nu}$$

- And recall that y^+ is computed at the cell center, therefore:

$$\Delta y^+ = 2 \times y^+$$

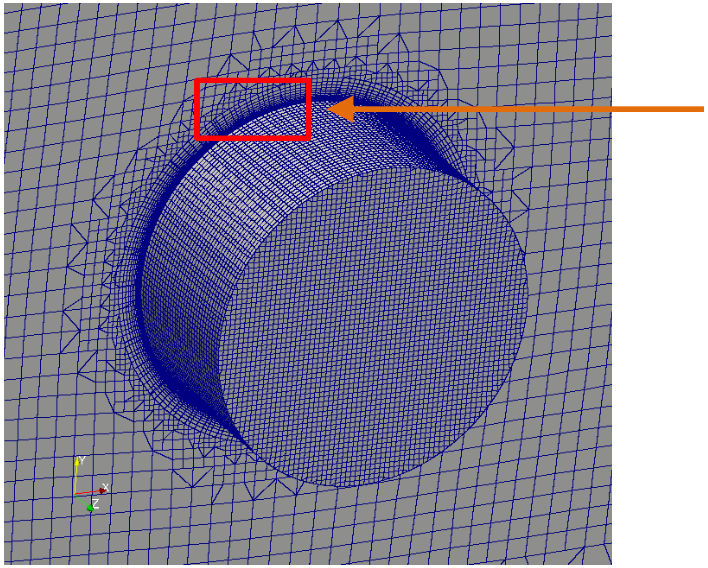
$$(\Delta x^+, \Delta y^+, \Delta z^+) = \left(\frac{x}{l_\tau}, \frac{y}{l_\tau}, \frac{z}{l_\tau} \right)$$

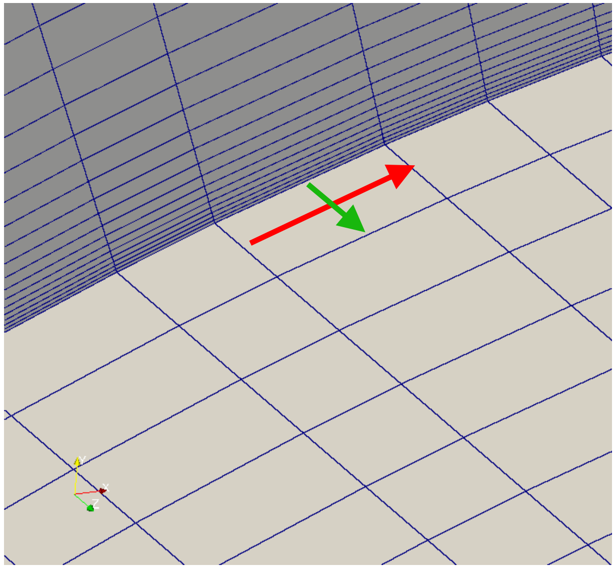
where

$$l_\tau = \frac{\nu}{U_\tau}$$

Viscous length

Wall distance units and some rough estimates





- Similar to y^+, the wall distance units can be computed in the span-wise (Δz^+) and stream-wise (Δx^+) directions.
- Typical requirements for LES are (these are approximations based on different references):

$$\Delta x^+ < 40, \;\; \Delta z^+ < 40 \quad \text{for} \quad y^+ < 10$$

Wall resolving

$$\Delta x^+ < 4\Delta y^+, \;\; \Delta z^+ < 4\Delta y^+ \quad \text{for} \quad 10 \leq y^+ \leq 300$$

Wall modeling

Wall distance units and some rough estimates

- DES and RANS simulations do not have stream-wise and span-wise wall distance units requirements as in LES simulations. Therefore, they are more affordable.
- If you are conducting DES simulations it is highly recommend to resolved the boundary layer, but you can also use wall functions.
- LES wall functions are valid across the whole boundary layer, even in the buffer layer).
- The upper limit of y^+ for LES and DES simulations should be less than 300 ($y^+ < 300$).
- Remember, it is strongly recommended to use wall functions with LES simulations. Otherwise your meshing requirements will be close to those of DNS.
- If you are doing DNS, y^+ should be less than 1. The spanwise and streamwise values should be less than 10, but ideally close to 1.

Wall distance units and some rough estimates

- If you are using wall functions with RANS models, you should avoid the buffer layer.
- In RANS, the typical range values for wall functions is between $30 \leq y^+ \leq 300$
- But values of y^+ up to 600 are also acceptable (depending on the Reynolds number).
- If you want to resolve the viscous layer of the boundary layer using RANS, $y^+ < 10$, you should cluster at least 10 cells in viscous region.
- If you want to use y^+ insensitive wall functions with RANS models, $1 \leq y^+ \leq 300$
- Remember the y^+ insensitive wall functions are only valid with y^+ insensitive turbulence models ($k - \omega$ SST).
- There are no strict requirements when it comes to the span-wise and stream-wise wall distance units in RANS, but as a general rule you can use Δx^+ and Δz^+ values as high as 300 times the value of Δy^+

A few mesh resolution guidelines and rough estimates

- The mesh is everything in CFD, and when it comes to turbulence modeling it is extremely important to have meshes with good quality and acceptable resolution.
- Some general guidelines for meshes to be used with RANS/DES/LES:
  - Resolve well the curvature.
  - Allow a smooth transition between cell of different sizes (at least 3 cells).
  - Identify the integral scales and try to cluster at least 5 cells in the domain regions where you expect to find the integral scales.

- Some guidelines specific to RANS meshes:
  - When it comes to RANS, the most important metric for mesh resolution is the y^+ value. Identify your wall treatment a-priory and mesh your domain according to this requirement.
  - If you are doing 3D simulations, there are no strict requirements when it comes to the span-wise and stream-wise directions, but as a general rule you can use Δx^+ and Δz^+ values as high as 300 the value of Δy^+.

y^+ and wall distance units

A few mesh resolution guidelines and rough estimates

- Some guidelines specific to DES meshes:
  - The mesh requirements are very similar to those of RANS meshes.
  - It is extremely important to resolve well the integral length scales.
  - DES simulations are intrinsically 3D.
  - Try to avoid the use of symmetry (axial and planar).
- Some guidelines specific to LES meshes:
  - When it comes to LES meshes, it is recommended to use wall functions. Otherwise the meshing requirements are similar to those of DNS.
  - It is recommended to use values in the range of $10 < y^+ < 60$. LES uses wall functions that can deal with the buffer layer.
  - In LES, it is extremely important to resolve well the stream-wise and span-wise directions. Recommended values are: $\Delta x^+ < 1000$ and $\Delta z^+ < 1000$
  - LES simulations are intrinsically 3D.
  - Try to avoid the use of symmetry (axial and planar).
  - Use hexahedral meshes.

References

- Turbulent Flows
  S. B. Pope
- Turbulence Modeling for CFD
  D. C. Wilcox
- Turbulence: An Introduction for Scientists and Engineers
  P. A. Davidson
- Large Eddy Simulation for Incompressible Flows
  P. Sagaut
- A First Course in Turbulence
  H. Tennekes and J. L. Lumley
- Boundary-Layer Theory
  H. Schlichting
- Turbulence Modelling - http://www.tfd.chalmers.se/~lada/comp_turb_model/
  Lars Davidson
- https://turbmodels.larc.nasa.gov/