

# Online Training – Advanced session

## January 2022

---

# Turbulence modeling in OpenFOAM: Theory and applications

# Copyright and disclaimer

This offering is not approved or endorsed by OpenCFD Limited, the producer of the OpenFOAM software and owner of the OPENFOAM® and OpenCFD® trademarks.

© 2014-2022 Wolf Dynamics.

All rights reserved. Unauthorized use, distribution or duplication is prohibited.

Contains proprietary and confidential information of Wolf Dynamics.

Wolf Dynamics makes no warranty, express or implied, about the completeness, accuracy, reliability, suitability, or usefulness of the information disclosed in this training material. This training material is intended to provide general information only. Any reliance the final user place on this training material is therefore strictly at his/her own risk. Under no circumstances and under no legal theory shall Wolf Dynamics be liable for any loss, damage or injury, arising directly or indirectly from the use or misuse of the information contained in this training material.

All trademarks are property of their owners.

Revision 1-2022

**JG**






# Before we begin

## On the training material

- **This training is based on OpenFOAM 9.**
- In the USB key/downloaded files you will find all the training material (tutorials, slides, and lectures notes).
- You can extract the training material wherever you want. From now on, this directory will become:
  - **\$TM**  
(abbreviation of **T**raining **M**aterial)
- To uncompress the tutorials go to the directory where you copied the training material (**\$TM**) and then type in the terminal,
  - `$> tar -zxvf file_name.tar.gz`
- In the case directory of every single tutorial, you will find a few scripts with the extension `.sh`, namely, `run_all.sh`, `run_mesh.sh`, `run_sampling.sh`, `run_solver.sh`, and so on.
- These scripts can be used to run the case automatically by typing in the terminal, for example,
  - `$> sh run_all.sh`
- These scripts are human-readable, and we highly recommend you open them, get familiar with the steps, and type the commands in the terminal. In this way, you will get used with the command line interface and OpenFOAM commands.
- If you are already comfortable with OpenFOAM, run the cases automatically using these scripts.
- In the case directory, you will also find the `README.FIRST` file. In this file, you will find some additional comments.

# Conventions used

## The following typographical conventions are used in this training material

- Text in `Courier new` font indicates Linux commands that should be typed literally by the user in the terminal.
- Text in **`Courier new bold`** font indicates directories.
- Text in *`Courier new italic`* font indicates human readable files or ascii files.
- Text in **Arial bold font** indicates program elements such as variables, function names, classes, statements and so on. It also indicates environment variables, and keywords. They also highlight important information.
- Text in [Arial underline in blue](#) font indicates URLs and email addresses.
- This icon  indicates a warning or a caution.
- This icon  indicates a tip, suggestion, or a general note.
- This icon  indicates a folder or directory.
- This icon  indicates a human readable file (ascii file).
- This icon  indicates that the figure is an animation (animated gif).
- These characters `$>` indicate that a Linux command should be typed literally by the user in the terminal.



# Conventions used

The following typographical conventions are used in this training material

- Large code listing, ascii files listing, and screen outputs can be written in a square box, as follows:

```
1  #include <iostream>
2  using namespace std;
3
4  // main() is where program execution begins. It is the main function.
5  // Every program in c++ must have this main function declared
6
7  int main ()
8  {
9      cout << "Hello world";           //prints Hello world
10     return 0;                        //returns nothing
11 }
```

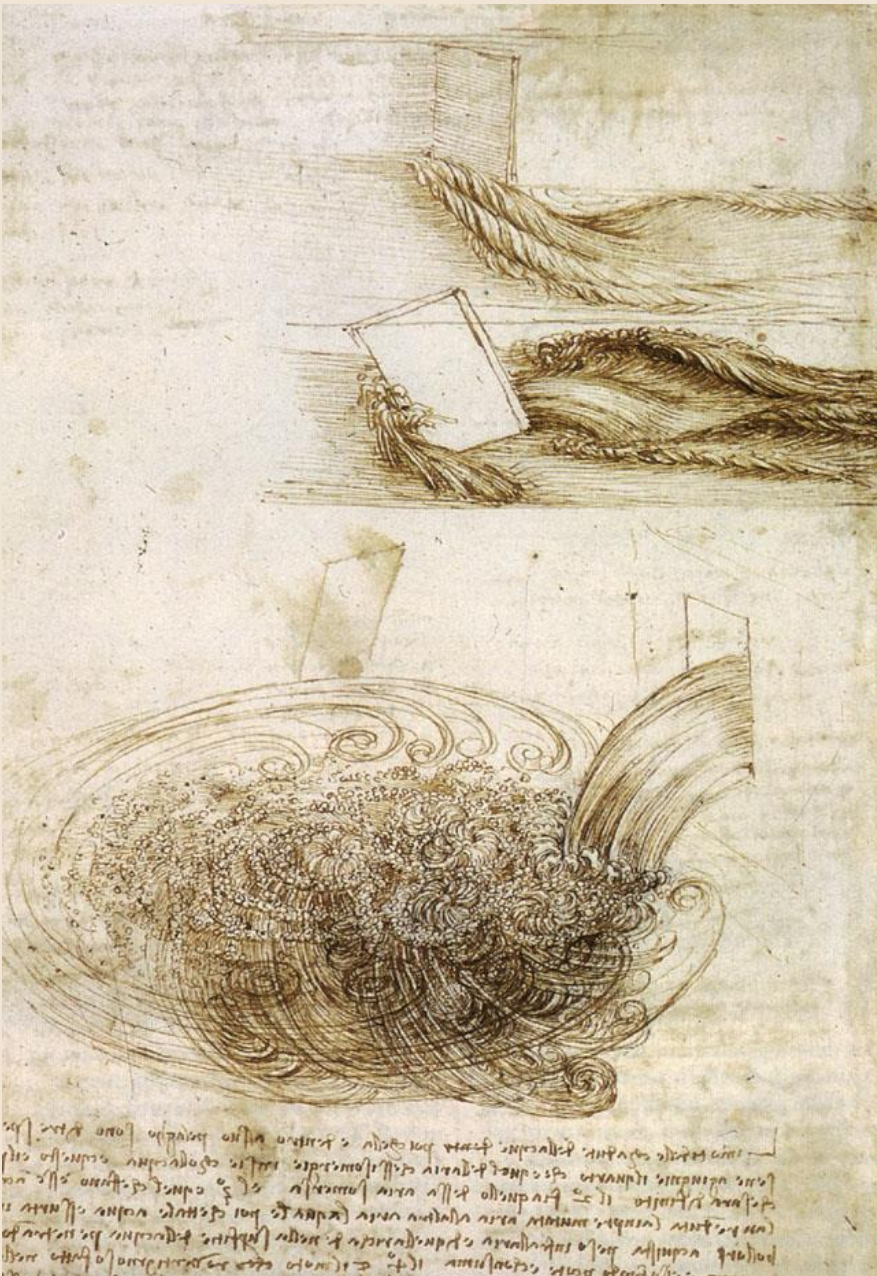
- To improve readability, the text might be colored.
- The font can be `Courier new` or **Arial bold**.
- And when required, the line number will be shown.

# Roadmap

- 1. The turbulent world around us**
- 2. Turbulence, does it matter?**
- 3. Introduction to turbulence modeling**
- 4. Post-processing turbulence simulations**
- 5.  $y^+$  and wall distance units**
- 6. Turbulence modeling in OpenFOAM**
- 7. Some applications using OpenFOAM**
- 8. Final remarks – General guidelines**

# Roadmap

- 1. The turbulent world around us**
2. Turbulence, does it matter?
3. Introduction to turbulence modeling
4. Post-processing turbulence simulations
5.  $y^+$  and wall distance units
6. Turbulence modeling in OpenFOAM
7. Some applications using OpenFOAM
8. Final remarks – General guidelines

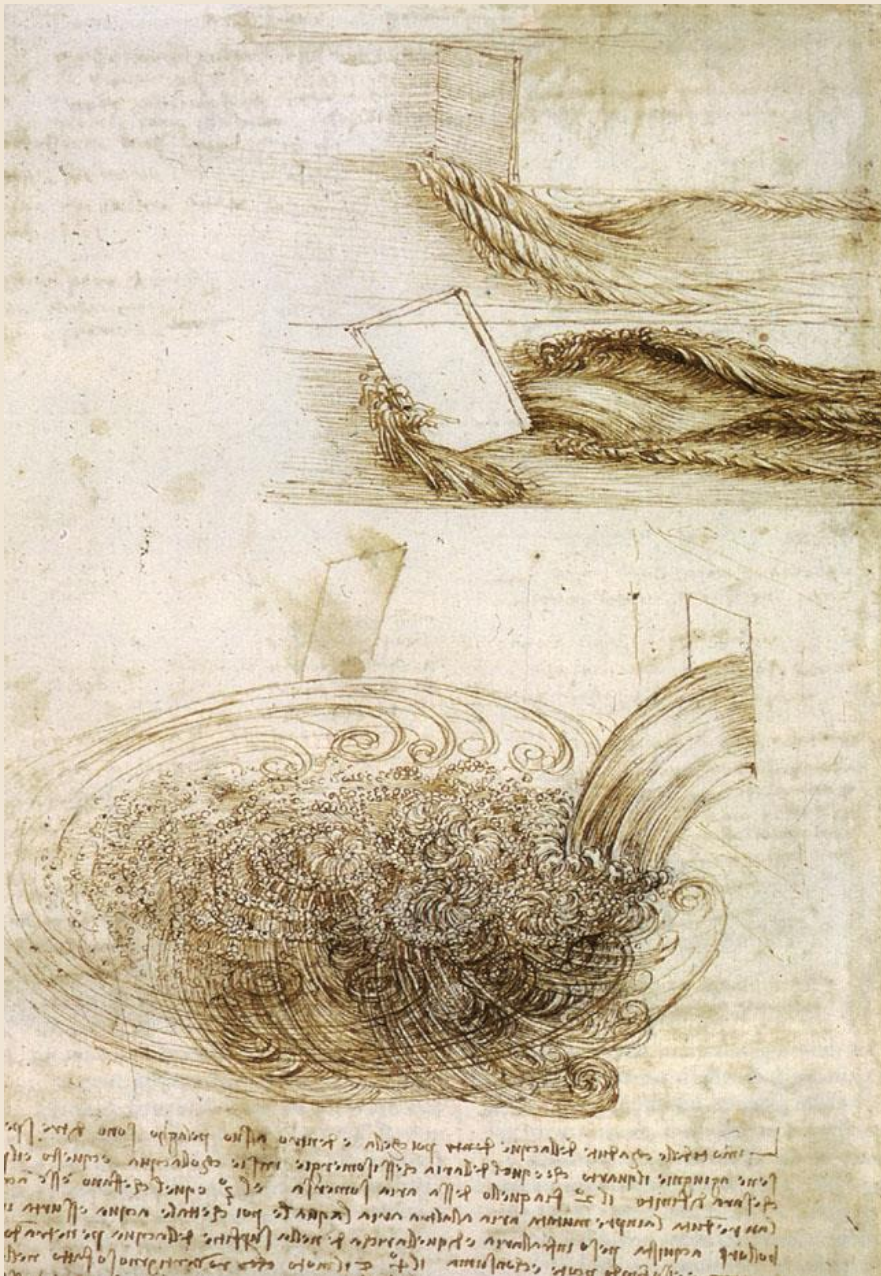


Leonardo da Vinci pioneered the flow visualization genre about 500 years ago. The illustrations to the left (*Studies of water passing obstacles and falling* c. 1508-1509) represents perhaps the world's first use of visualization as a scientific tool to study a turbulent flow.

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.”*



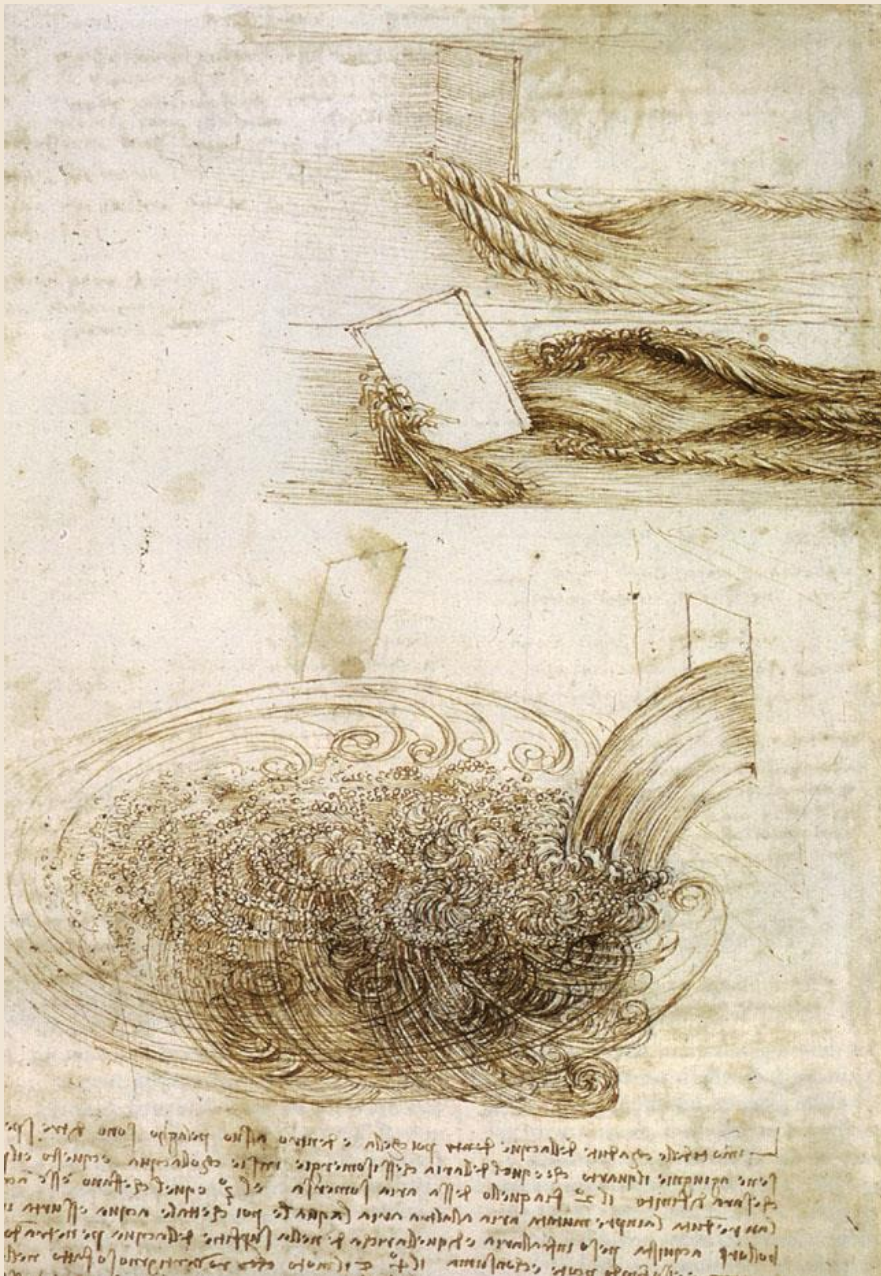


Leonardo da Vinci pioneered the flow visualization genre about 500 years ago. The illustrations to the left (*Studies of water passing obstacles and falling* c. 1508-1509) represents perhaps the world's first use of visualization as a scientific tool to study a turbulent flow.

The following da Vinci's 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.”*





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 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”*

# The turbulent world around us

## What is turbulence?

- According to the Merriam-Webster dictionary, turbulence is defined as,

### turbulence noun



Save Word

tur·bu·lence | \ 'tər-byə-lən(t)s \

#### Definition of *turbulence*

: the quality or state of being turbulent: such as

**a** : great commotion or agitation

// emotional *turbulence*

**b** : irregular atmospheric motion especially when characterized by up-and-down currents

**c** : departure in a fluid from a smooth flow

- Similar definitions follow in different languages.
- Note that this turbulence definition is too general.
- Moving to technical grounds, namely, fluid dynamics, the use of the word **turbulent** to characterize a certain type of flow, namely, the counterpart of a streamline motion, smooth flow, or a laminar flow, is comparatively recent.

# The turbulent world around us

## What is turbulence?

- Not even Osborne Reynolds used the word turbulent in his two principal papers on turbulent flows,
  - O. Reynolds (1883). “An experimental investigation of the circumstances which determine whether the motion of water shall be direct or sinuous, and of the law of resistance in parallel channels.”
  - O. Reynolds (1895). “On the dynamical theory of incompressible viscous fluids and the determination of the criterion.”
- On these two papers, Osborne Reynolds, defined turbulence (or turbulent flows) as eddying or sinuous motion.
- Both papers were published in the Philosophical Transactions of the Royal Society, and essentially provided a marker for the direction of research in Engineering Fluid Mechanics for the next century.
- In fluid dynamics we prefer to use the terminology **turbulent flows** instead of **turbulence**.
- However, both terms are valid when dealing at a technical level.
- Hereafter, we will use both terminologies interchangeably.



# The turbulent world around us

## 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 slides).
  - 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”*

# The turbulent world around us

## What is turbulence?

- Due to the extremely complex nature of turbulence and its incomplete understanding, there is not a single accepted definition of turbulence.
  - 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”*

- T. von Karman [2] who is known for his studies about Fluid Dynamics, quotes G. I. Taylor with the following definition of turbulence in 1937,

*“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] L. F. Richardson. “Weather Prediction by Numerical Process”. Cambridge University Press, 1922.

[2] J. O. Hinze. “Turbulence”. McGraw-Hill, New York, 1959.

# The turbulent world around us

## What is turbulence?

- Due to the extremely complex nature of turbulence and its incomplete understanding, there is not a single accepted definition of turbulence.
  - J.O. Hinze offers yet another definition for turbulence in 1959 [1],

*“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.”*

- Monin and Yaglom [2], in 1971, gave another definition of laminar and turbulent flows.

*“It is known that all flows of liquids and gases may be divided into two sharply different types; the quiet smooth flows known as “laminar” flows, and their opposite, “turbulent” flows in which the velocity, pressure, temperature and other fluid mechanical quantities fluctuate in a disordered manner with extremely sharp and irregular space- and time-variations.”*

[1] J. O. Hinze. “Turbulence”. McGraw-Hill, New York, 1959.

[2] A. S. Monin and A. M. Yaglom. Statistical Fluid Mechanics: The Mechanics of Turbulence, volume 1. M. I. T. Press, 1971.

# The turbulent world around us

## 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”. Dwoyer et al.(eds), Springer-Verlag, New York, pp. 19–49, 1985.

[2] S. Rodriguez. “Applied Computational Fluid Dynamics and Turbulence modeling”. Springer, 2019.

# The turbulent world around us

## 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.

# The turbulent world around us

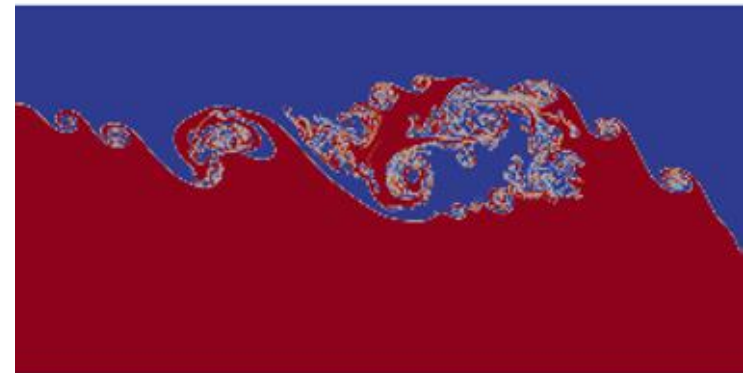
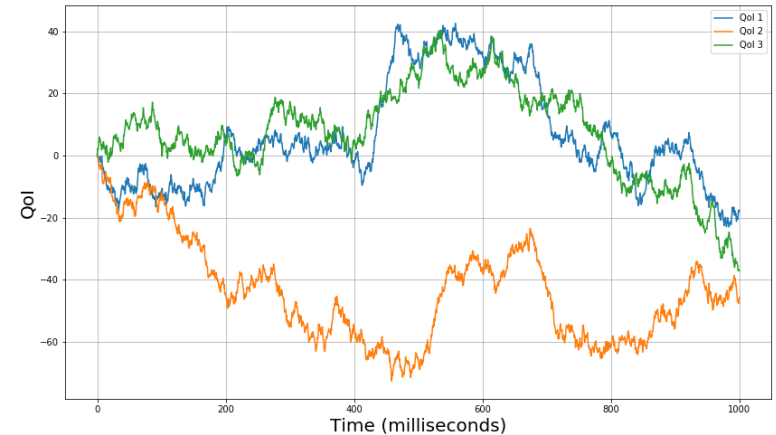
## Turbulent flows have the following characteristics

- 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 statistically.
- 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.

# The turbulent world around us

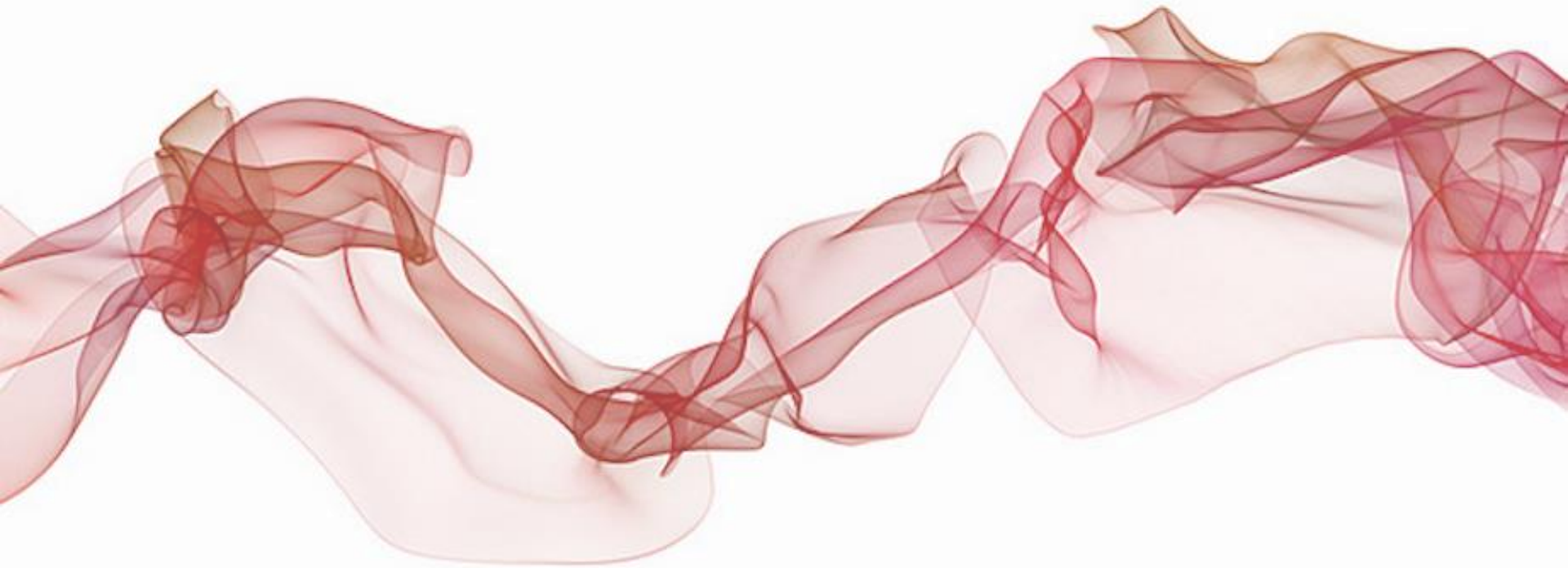
## ... so, what is turbulence?

- Turbulent flows share all the previous characteristics.
- For the purposes of this training, let us define turbulence as follows,
  - 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 contain a wide range of eddy sizes (scales):
    - Large eddies derive 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.



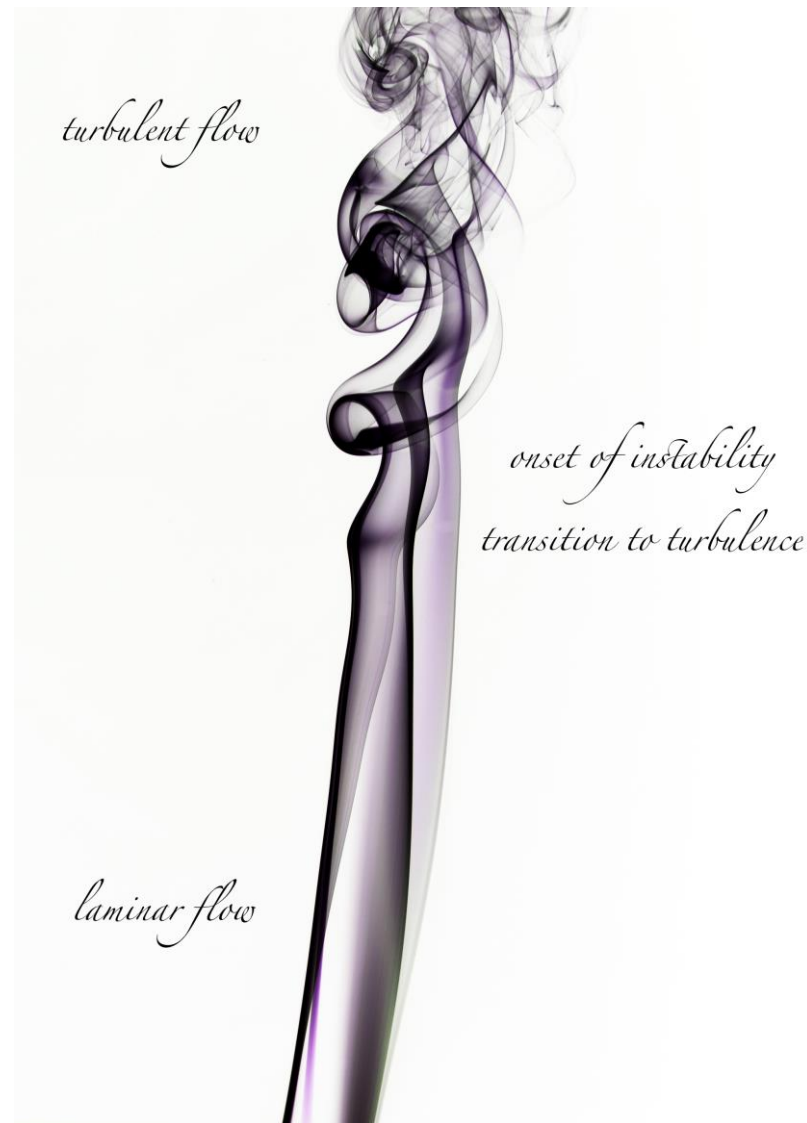
# The turbulent world around us

Before continuing, we want to share a few amazing images that show the beauty and complexity of turbulence in nature and engineering applications.





# The turbulent world around us

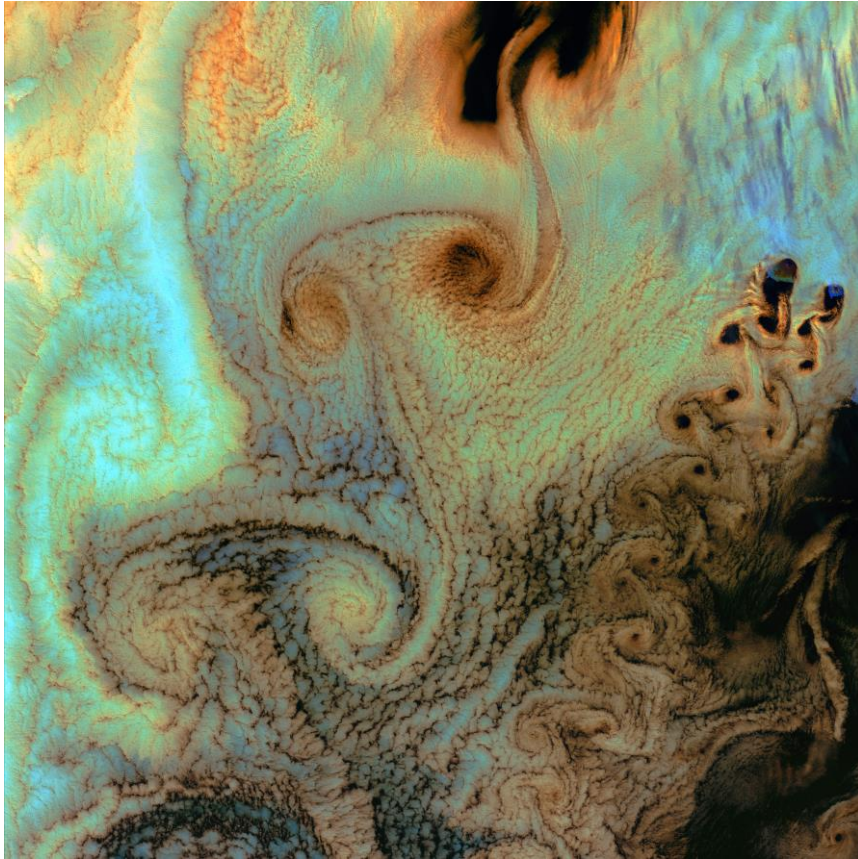


**Buoyant plume of smoke rising from a stick of incense**

Photo credit: <https://www.flickr.com/photos/jlhoppgood/>

This work is licensed under a Creative Commons License (CC BY-NC-ND 2.0)

# 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.  
Copyright on the images is held by the contributors. Apart from Fair Use, permission must be sought for any other purpose.



**Von Karman Vortex Streets in the northern Pacific Photographed from the International Space Station**

Photo credit: NASA

Copyright on the images is held by the contributors. Apart from Fair Use, permission must be sought for any other purpose.



# The turbulent world around us



## **Turbulent waters**

Photo credit: <https://www.flickr.com/photos/thepaegan>

This work is licensed under a Creative Commons License (CC BY-NC-ND 2.0)



## **Cirrus clouds - Kelvin-Helmholtz instability**

Photo credit: <https://www.pinterest.it/pin/8514686771867411/>

Copyright on the images is held by the contributors. Apart from Fair Use, permission must be sought for any other purpose.

# The turbulent world around us



**Tugboat riding on the turbulent wake of a ship**

Photo credit: <https://www.flickr.com/photos/oneeighteen/>

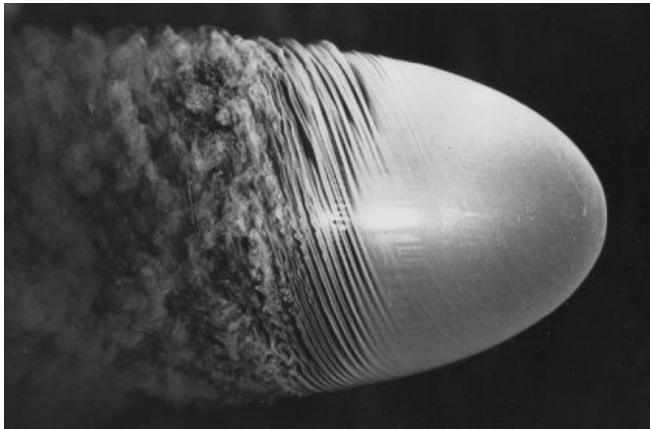
This work is licensed under a Creative Commons License (CC BY-NC 2.0)



**Trailing vortices**

Photo credit: Steve Morris. AirTeamImages.

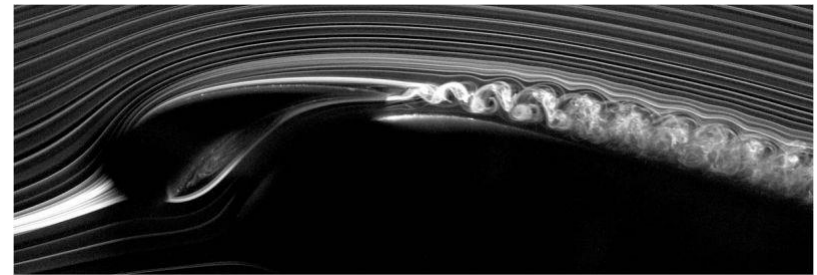
Copyright on the images is held by the contributors. Apart from Fair Use, permission must be sought for any other purpose.



**Flow visualization over a spinning spheroid**

Photo credit: Y. Kohama.

Copyright on the images is held by the contributors. Apart from Fair Use, permission must be sought for any other purpose.

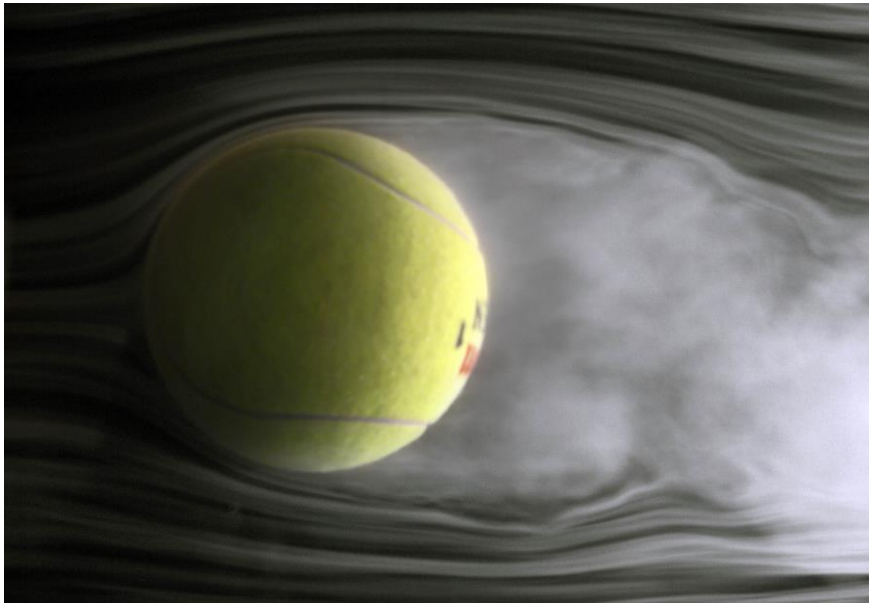


**Flow around an airfoil with a leading-edge slat**

Photo credit: S. Makiya et al.

Copyright on the images is held by the contributors. Apart from Fair Use, permission must be sought for any other purpose.

# The turbulent world around us



## Wind Tunnel Test of New Tennis Ball

Photo credit: NASA

<http://tennisclub.gsfc.nasa.gov/tennis.windtunnelballs.html>

Copyright on the images is held by the contributors. Apart from Fair Use, permission must be sought for any other purpose.



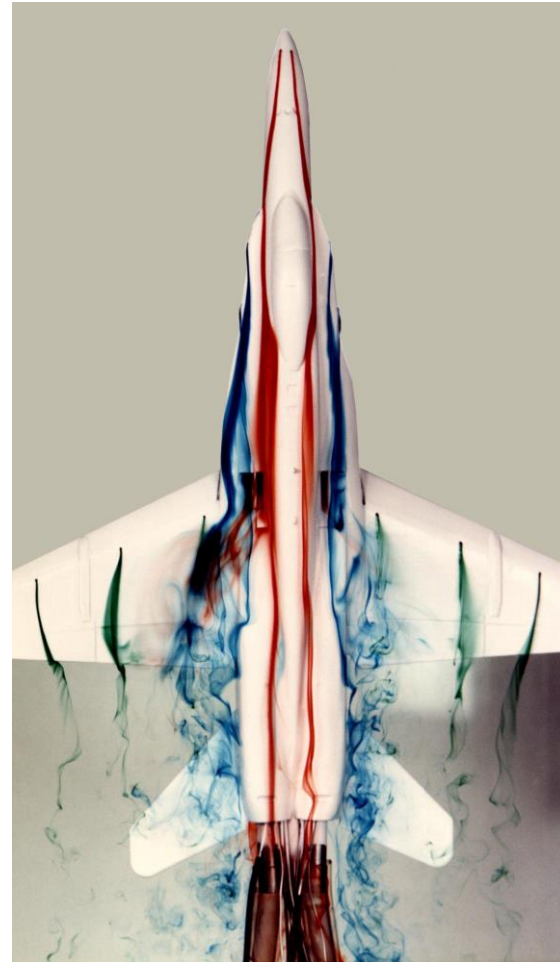
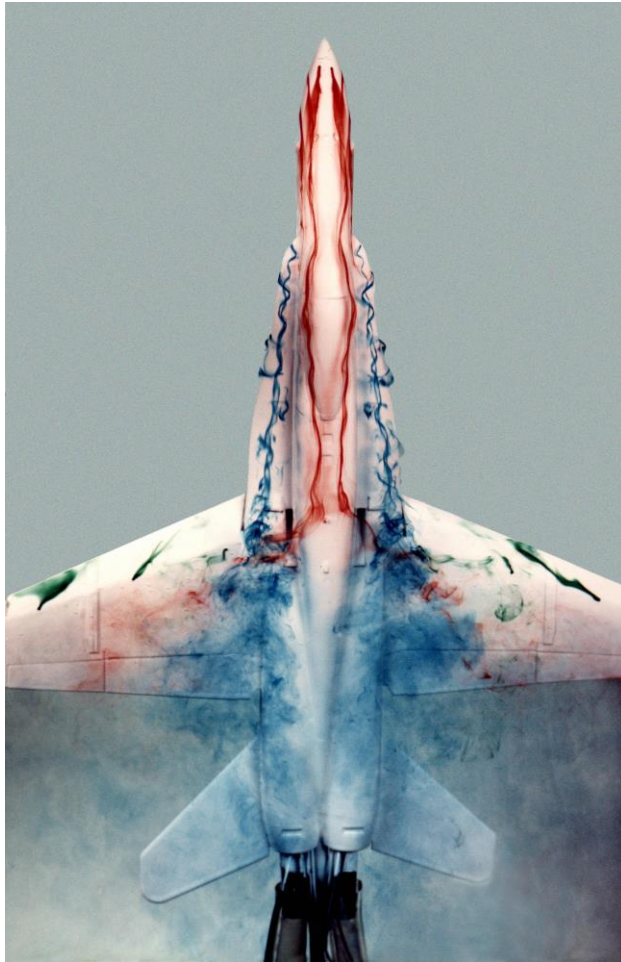
## Wake turbulence behind individual wind turbines

Photo credit: NREL's wind energy research group.

Copyright on the images is held by the contributors. Apart from Fair Use, permission must be sought for any other purpose.



# 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>  
Copyright on the images is held by the contributors. Apart from Fair Use, permission must be sought for any other purpose.

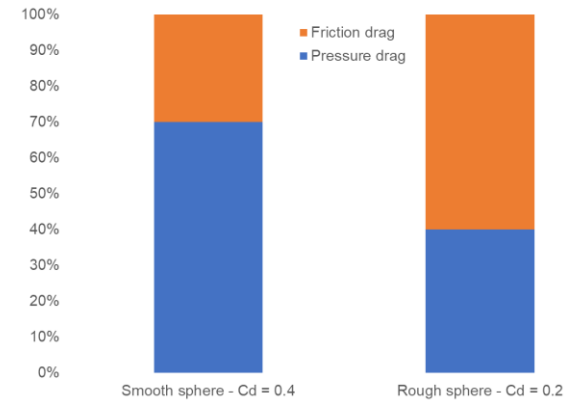
# The turbulent world around us



(a)  $C_D \approx 0.4$



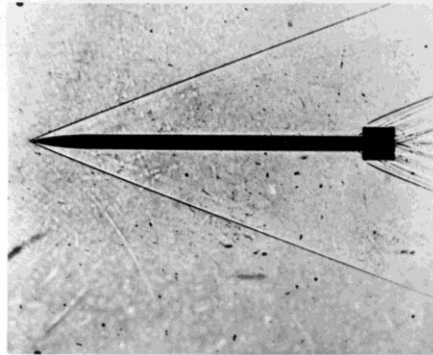
(b)  $C_D \approx 0.2$



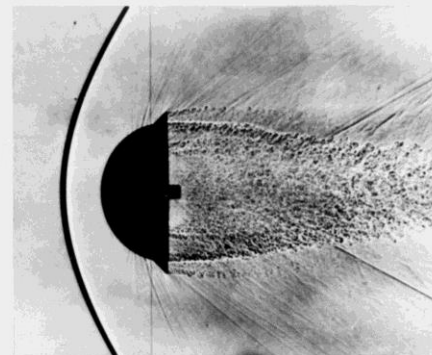
**Abstract representation of the drag decomposition**

# The turbulent world around us

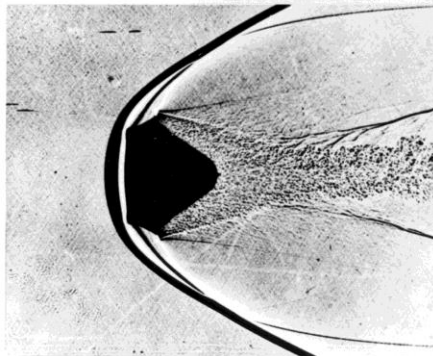
## RESEARCH CONTRIBUTING TO PROJECT MERCURY



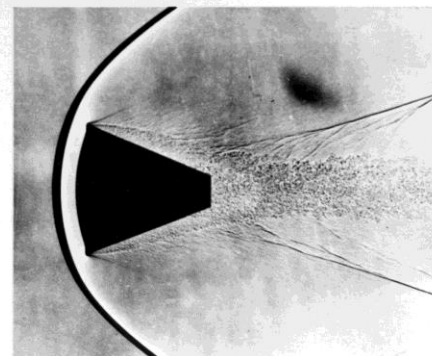
INITIAL CONCEPT



BLUNT BODY CONCEPT 1953



MISSILE NOSE CONES 1953-1957



MANNED CAPSULE CONCEPT 1957

### Shadowgraph Images of Re-entry Vehicles

Photo credit: NASA on the Commons. <https://www.flickr.com/photos/nasacommons/>

Copyright on the images is held by the contributors. Apart from Fair Use, permission must be sought for any other purpose.



# The turbulent world around us

## Astrophysical, plasma, planetary and quantum turbulence

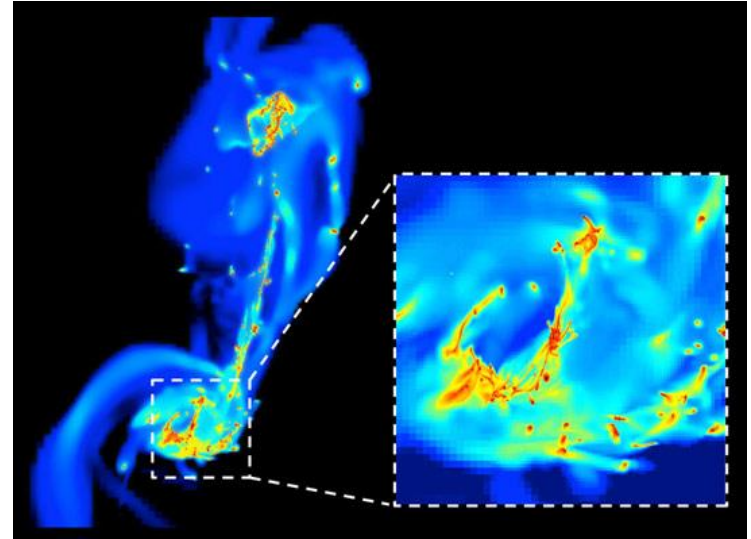


**M8: The Lagoon Nebula**

Photo credit: Steve Mazlin, Jack Harvey, Rick Gilbert, and Daniel Verschate.

Star Shadows Remote Observatory, PROMPT, CTIO

Copyright on the images is held by the contributors. Apart from Fair Use, permission must be sought for any other purpose



**A frame from the simulation of the two colliding Antennae galaxies.**

Photo credit: F. Renaud / CEA-Sap.

Copyright on the images is held by the contributors. Apart from Fair Use, permission must be sought for any other purpose



**Jupiter photo taken by Juno's cam.**

Photo credit: NASA / JPL / SwRI / MSSS / David Marriott

Copyright on the images is held by the contributors. Apart from Fair Use, permission must be sought for any other purpose

# Roadmap

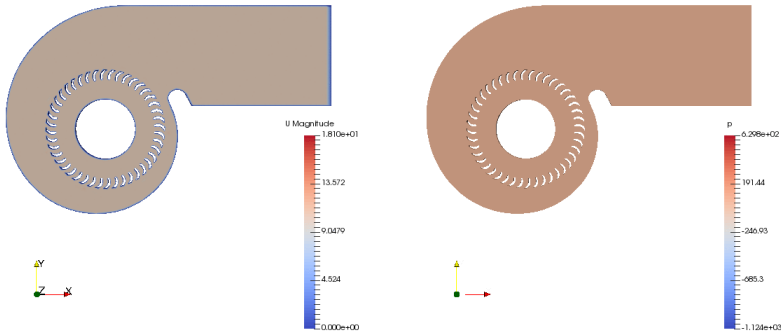
- ~~1. The turbulent world around us~~
- 2. Turbulence, does it matter?**
- ~~3. Introduction to turbulence modeling~~
- ~~4. Post-processing turbulence simulations~~
- ~~5.  $y^+$  and wall distance units~~
- ~~6. Turbulence modeling in OpenFOAM~~
- ~~7. Some applications using OpenFOAM~~
- ~~8. Final remarks – General guidelines~~

# Turbulence, does it matter?

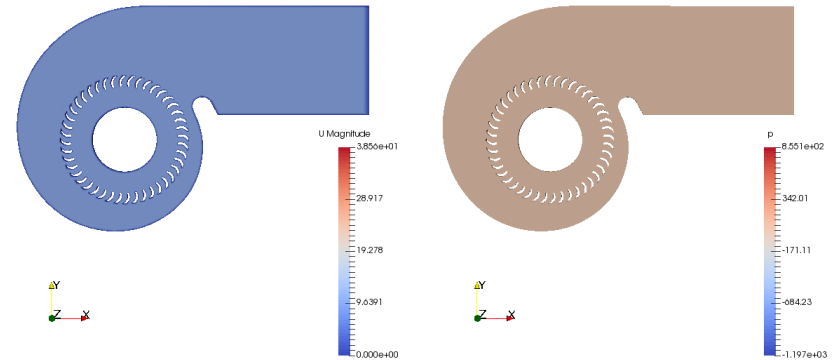
## Blower simulation using sliding grids



Time: 0.000000



Time: 0.000000



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

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

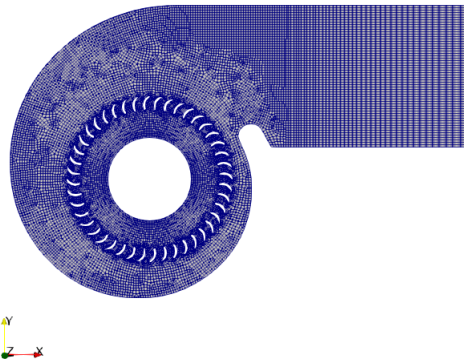
**K-epsilon turbulence model**

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

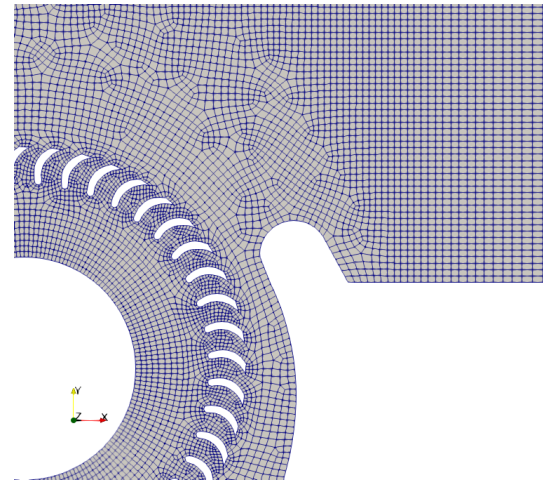
# Turbulence, does it matter?

## Blower simulation using sliding grids

- In this case, and even though the mesh is coarse, thanks to the help of the turbulence model, we managed to capture the right physics.



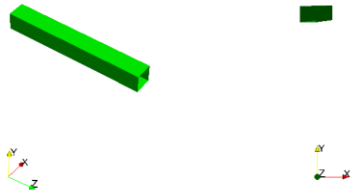
<http://www.wolfdynamics.com/training/turbulence/blower1.gif>



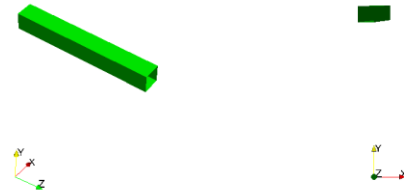
<http://www.wolfdynamics.com/training/turbulence/blower2.gif>

# Turbulence, does it matter?

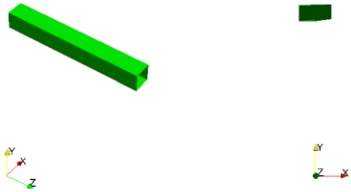
## Vortex shedding past square cylinder



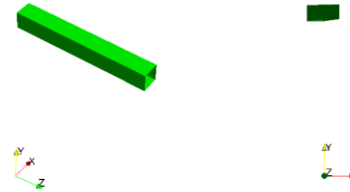
**URANS (K-Omega SST with no wall functions) – Vortices visualized by Q-criterion**  
[www.wolfdynamics.com/wiki/squarecil/urans2.gif](http://www.wolfdynamics.com/wiki/squarecil/urans2.gif)



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



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



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

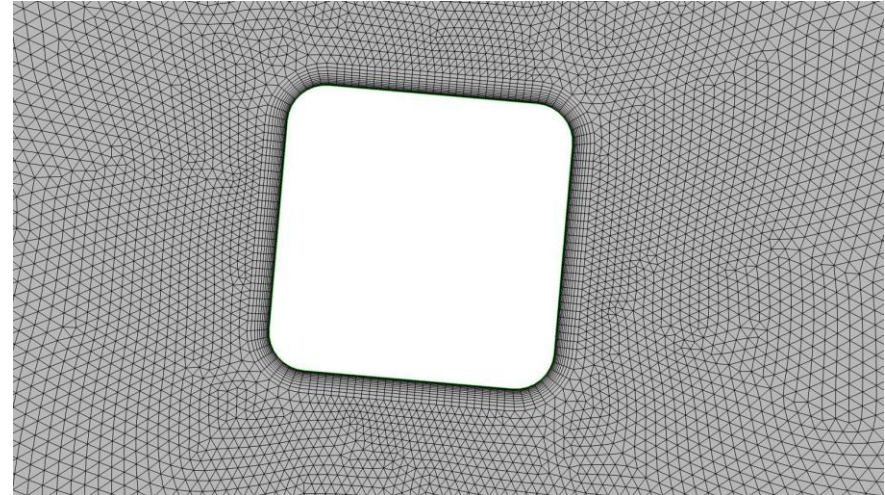
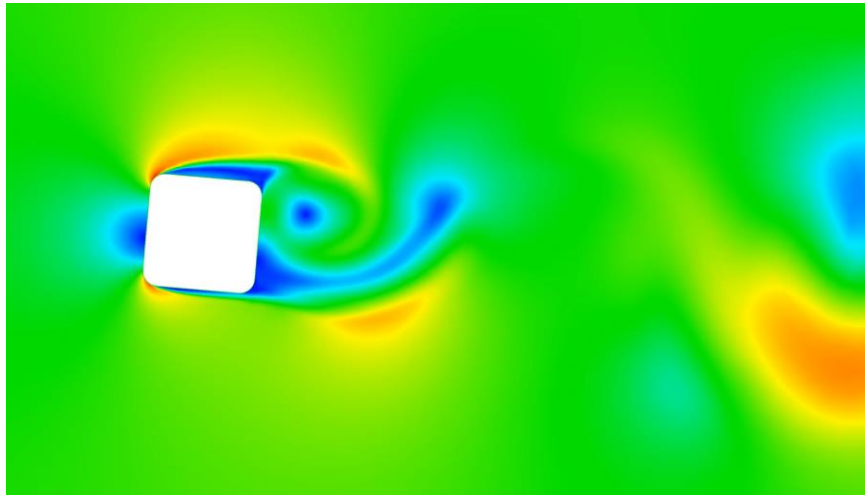
**Note:** all simulations were run using 4 cores.

### 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

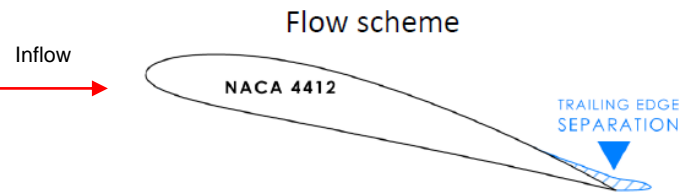
[www.wolfdynamics.com/wiki/turb/media1.mp4](http://www.wolfdynamics.com/wiki/turb/media1.mp4)

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



# Turbulence, does it matter?

## Separated flow around a NACA-4412 airfoil



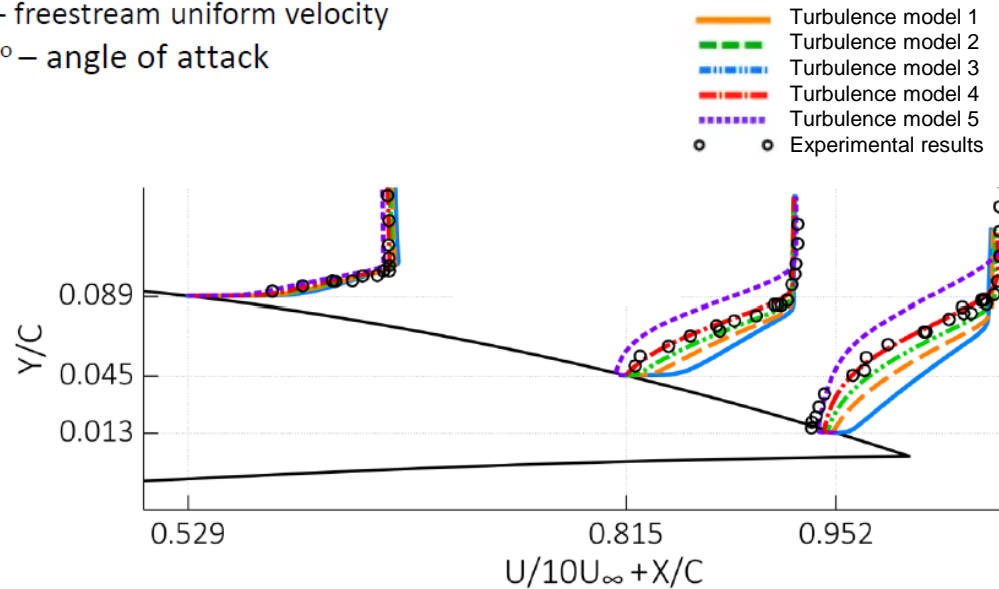
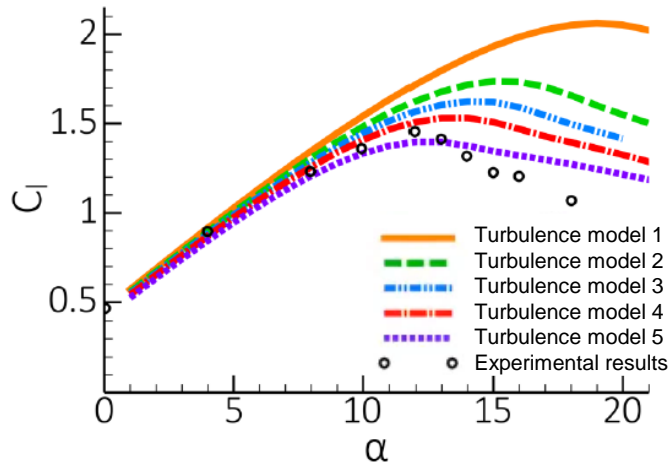
Incompressible flow

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

$C$  - airfoil chord

$U_{\infty}$  - freestream uniform velocity

$\alpha = 12^\circ$  - angle of attack



- Turbulence model 1
- Turbulence model 2
- Turbulence model 3
- Turbulence model 4
- Turbulence model 5
- Experimental results

- 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.

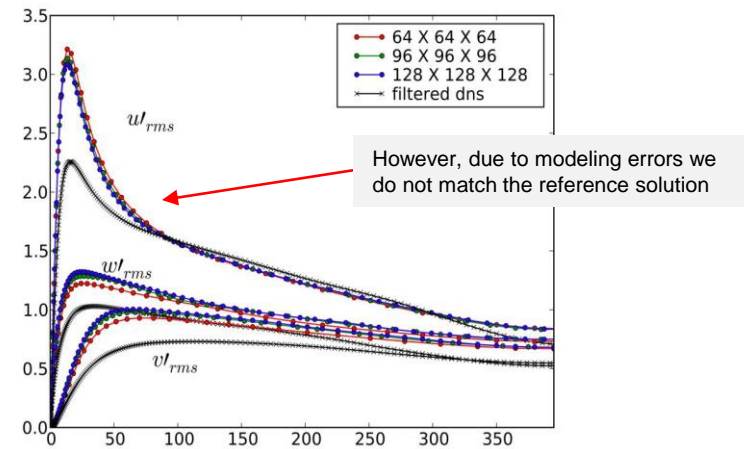
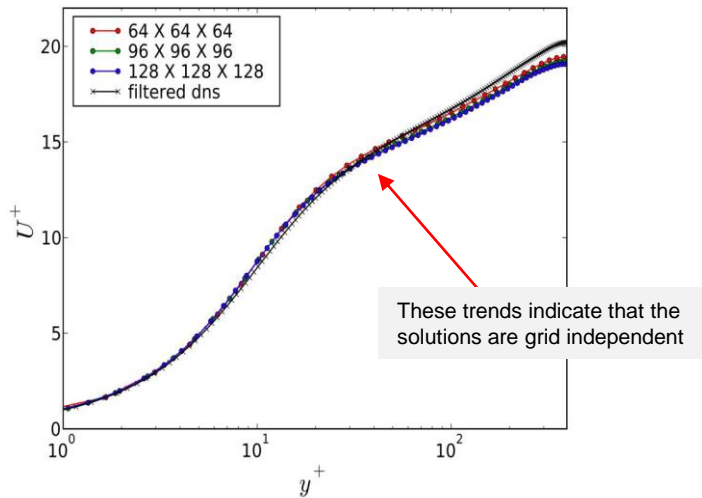
### References:

- 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?

## Grid independent solutions and modeling errors



- CFD has been around since the late 1970s. Since then, a lot of progress has been done in hardware, software, algorithms and turbulence models.
- But many times, even if we have grid-independent solutions, we fail to get a good match with the experiments (not that the experiments are always right) or a reference solution due to modeling errors or numerical dissipation.

*"The multiscale nature of turbulence creates unique challenges for numerical simulations. Discretization methods must preserve the physical processes, reducing or eliminating artificial dissipation and dispersion ...  
... How do you establish confidence in the numerical simulations of turbulent flows?"*

### References:

P. Moin. "Turbulence: V&V and UQ Analysis of a Multi-scale Complex System". Center for Turbulence Research Stanford University

# Turbulence, does it matter?

## Turbulence is not a trivial problem – Some quotes

*“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

*“Turbulence is the graveyard of theories”*

Hans W. Liepmann

# Turbulence, does it matter?

## Turbulence is not a trivial problem – Some quotes

- This is probably my favorite quote, as it covers the largest elephants in CFD, mesh and turbulence.

*“Geometry modeling is to meshing what turbulence modeling is to computational fluid dynamics (CFD) – a mathematically complex model of something important that we try to treat as the proverbial black box.”*

John Chawner - Pointwise

# Roadmap

- ~~1. The turbulent world around us~~
- ~~2. Turbulence, does it matter?~~
- 3. Introduction to turbulence modeling**
- ~~4. Post-processing turbulence simulations~~
- ~~5.  $y^+$  and wall distance units~~
- ~~6. Turbulence modeling in OpenFOAM~~
- ~~7. Some applications using OpenFOAM~~
- ~~8. Final remarks – General guidelines~~

# Introduction to turbulence modeling

- Turbulence modeling is a very complex field.
- Hereafter, we will address the most important concepts.
- We will also address the best standard practices.
- If you want to dig deeper into the theory, you are invited to review the literature.
- In particular, we recommend the following references:
  - S. Pope. Turbulent Flows. Cambridge University Press, 2000.
  - D. Wilcox. Turbulence Modeling for CFD. DCW Industries ,Inc., 2010.
  - M. Leschziner. Statistical Turbulence modelling for Fluid Dynamics. Imperial College Press, 2016.
  - P. Bernard. Turbulent Fluid Flow. Wiley, 2019.
  - S. Rodriguez. Applied Computational Fluid Dynamics and Turbulence Modeling. Springer, 2020.
  - Turbulence Modelling by Lars Davidson [http://www.tfd.chalmers.se/~lada/comp\\_turb\\_model/](http://www.tfd.chalmers.se/~lada/comp_turb_model/)
  - NASA turbulence modeling resources - <https://turbmodels.larc.nasa.gov/>
  - Turbulence and CFD models by J. Guerrero [http://www3.dicca.unige.it/guerrero/teaching\\_turbulence.html](http://www3.dicca.unige.it/guerrero/teaching_turbulence.html)

# Introduction to turbulence modeling

*“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”*.

# Introduction to turbulence modeling

## Turbulence modeling in engineering

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, from smallest to system scales, as a function of time.



# Introduction to turbulence modeling

## Turbulence modeling in engineering

- Most natural and engineering flows are turbulent, hence the necessity of modeling turbulence.
- Turbulence can be:
  - Wall bounded → It develops at the walls.
  - Free shear → Mixing layers and jets.
- 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 treatment, and choosing runtime parameters and numerics.

# Introduction to turbulence modeling

## 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.
- Contain a wide range of scales (eddies), ranging from very small scales to system scale.
- Therefore, in order to accurately model/resolve turbulent flows, the simulations must be three-dimensional, time-accurate, and with meshes fine enough such that all spatial and temporal scales are properly captured.
- Additional physics that makes turbulence modeling even harder:
  - Buoyancy, compressibility effects, heat transfer, multiphase flows, transition to turbulence, surface finish (roughness), combustion, adverse pressure gradients, high speed flows, shock waves, and so on.

# Introduction to turbulence modeling

## Reynolds number and Rayleigh number

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

$$\begin{array}{l} \text{Convective effects} \longrightarrow \\ \text{Viscous effects} \longrightarrow \end{array} Re_L = \frac{\rho U L}{\mu} = \frac{U L}{\nu}$$
$$\nu = \frac{\mu}{\rho}$$

- Where  $U$  is a characteristic velocity, e.g., free-stream velocity.
- And  $L$  is a representative length scale (or system scale), e.g., length, height, diameter, etc.

# Introduction to turbulence modeling

## Reynolds number and Rayleigh number

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

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

Specific heat  $\downarrow$   $c_p$   
 Thermal expansion coefficient  $\swarrow$   $\beta$   
 Thermal conductivity  $\nwarrow$   $k$

Momentum diffusivity  $\longrightarrow$   $Pr = \frac{\nu}{\alpha} = \frac{\mu c_p}{k}$   
 Thermal diffusivity  $\longrightarrow$

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

# Introduction to turbulence modeling

## Reynolds number and Rayleigh number

- Turbulent flow occurs at large Reynolds number.

- For external flows,

$$Re_x \geq 500000 \quad \text{Around slender/streamlined bodies (aerodynamic bodies)}$$

$$Re_d \geq 20000 \quad \text{Around an obstacle (bluff bodies)}$$

- For internal flows,

$$Re_{d_h} \geq 2300$$

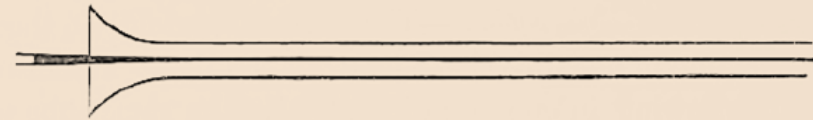
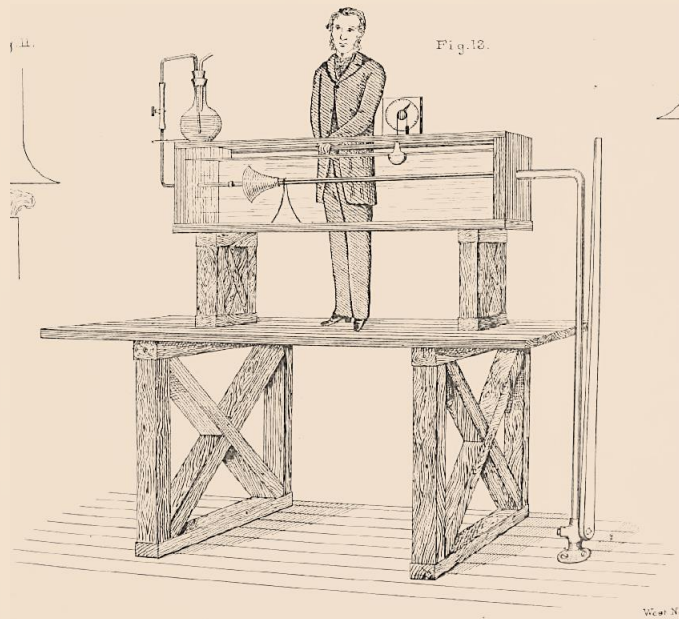
- Notice that other factors such as free-stream turbulence, surface conditions, transpiration (blowing/suction), roughness and other disturbances, may cause transition to turbulence at lower Reynolds number.
- If you are dealing with natural convection and buoyancy, turbulent flows occur when,

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

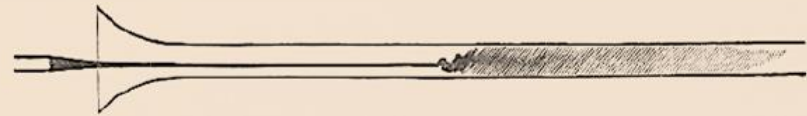


# Introduction to turbulence modeling

## Historical note – Osborne Reynolds experiment



Laminar flow

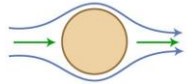


Turbulent flow – Sinuous flow

- 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.

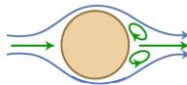
# Introduction to turbulence modeling

## What happens when we increase the Reynolds number?



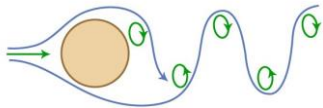
**Creeping flow (no separation)**  
Steady flow

$$Re < 5$$



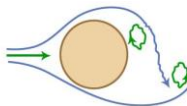
**A pair of stable vortices  
in the wake**  
Steady flow

$$5 < Re < 40 - 46$$



**Laminar vortex street  
(Von Karman street)**  
Unsteady flow

$$40 - 46 < Re < 150$$

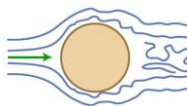


**Laminar boundary layer up to  
the separation point, turbulent  
wake**  
Unsteady flow

$$150 < Re < 300$$

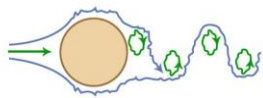
**Transition to turbulence**

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



**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 physics.

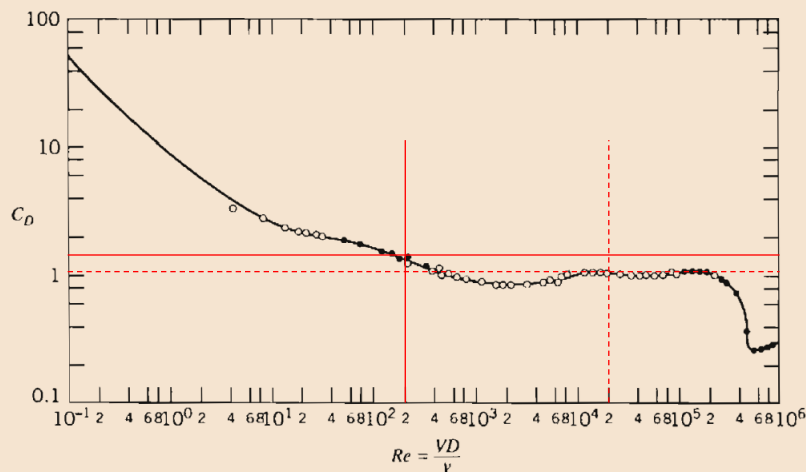
- Relatively easy to simulate.
- It becomes more challenging when the boundary layer transition to turbulent.
- Unsteady physics.

- Very challenging to simulate due to turbulent boundary layer and wake, massive separation, and strong unsteadiness.
- Unsteady physics.

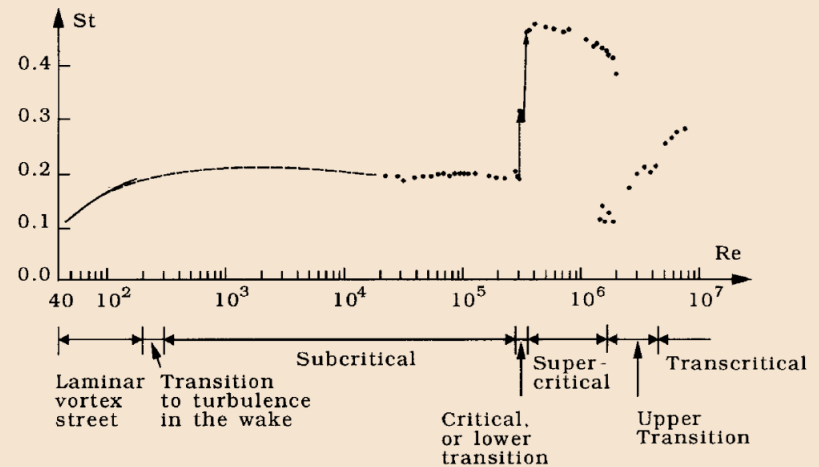
Vortex shedding behind a cylinder and Reynolds number

# Introduction to turbulence modeling

## What happens when we increase the Reynolds number?



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



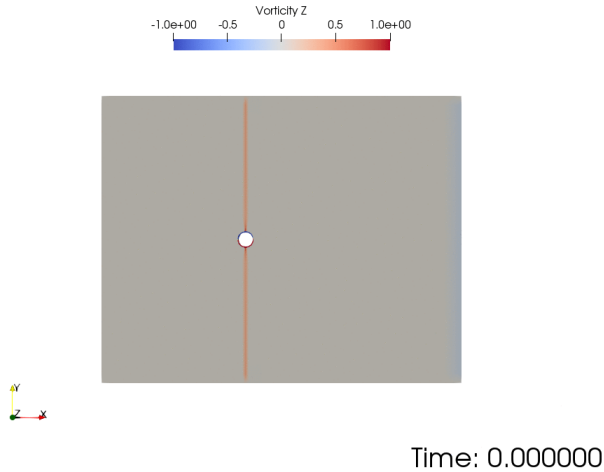
Strouhal number  $St = \frac{fL}{U}$  for a smooth cylinder [2]

## 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 Scientific, 2006

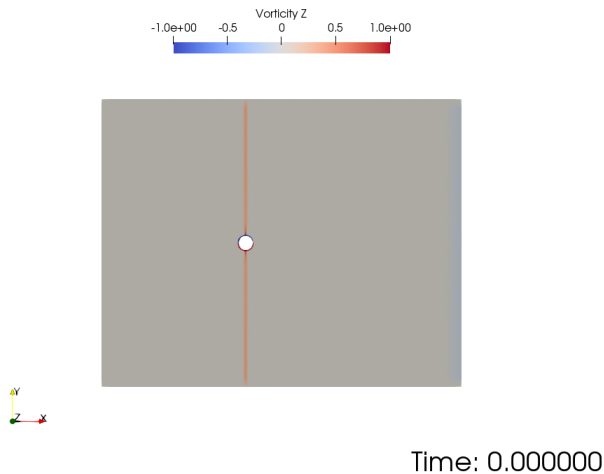
# Introduction to turbulence modeling

## What happens when we increase the Reynolds number?



**Reynolds = 20 – Non-uniform initialization**  
**Laminar flow with separation**

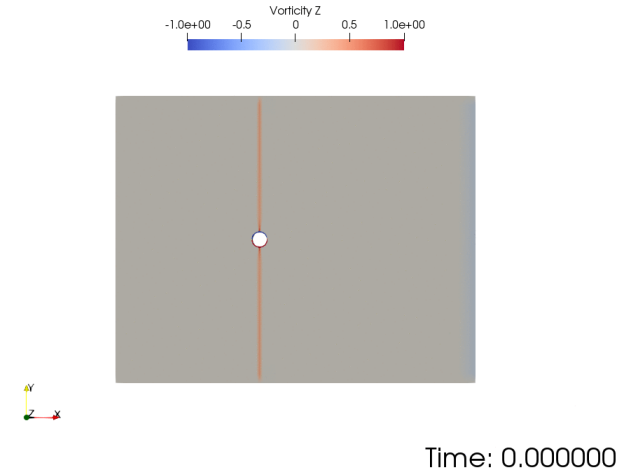
<http://www.wolfdynamics.com/training/turbulence/unscyl1.gif>



**Reynolds = 50 – Non-uniform initialization**

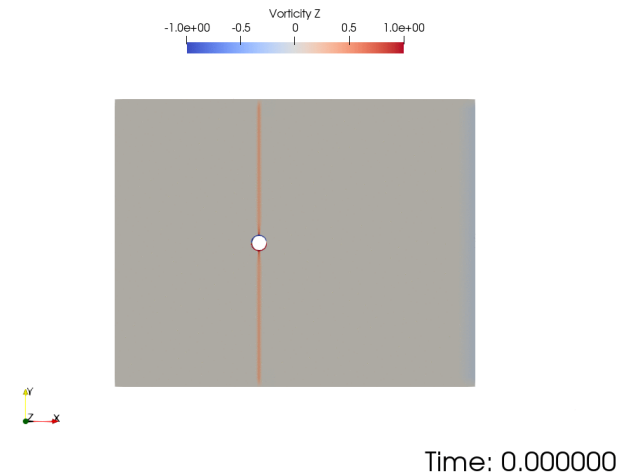
**Laminar flow with vortex shedding (maybe on the limit of the onset of the Von Karman street.**

<http://www.wolfdynamics.com/training/turbulence/unscyl2.gif>



**Reynolds = 200 – Non-uniform initialization**  
**Laminar flow with vortex shedding**

<http://www.wolfdynamics.com/training/turbulence/unscyl3.gif>



**Reynolds = 20000 – Non-uniform initialization**

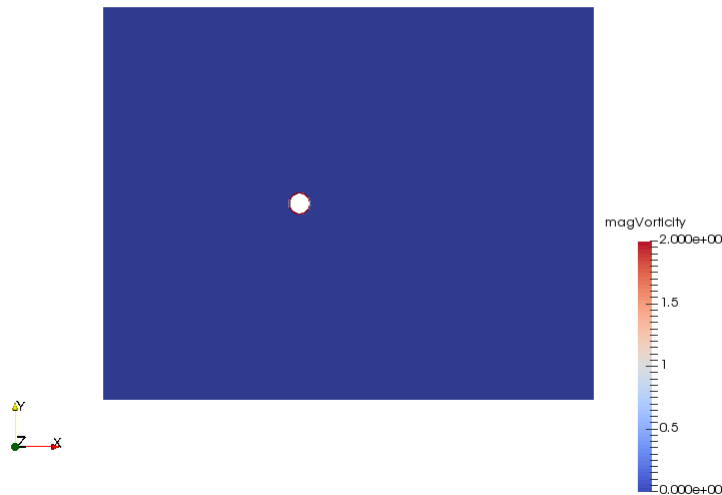
**Turbulent flow with vortex shedding (turbulence model enable)**

<http://www.wolfdynamics.com/training/turbulence/unscyl4.gif>

# Introduction to turbulence modeling

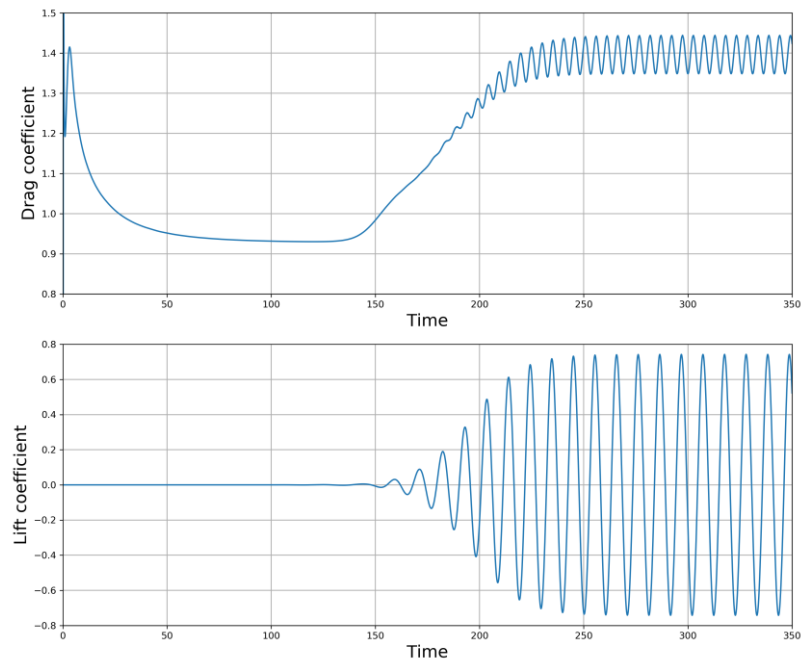
## Vorticity does not always mean turbulence

Time: 0.000000



### Instantaneous vorticity magnitude field

<http://www.wolfdynamics.com/training/turbulence/image6.gif>

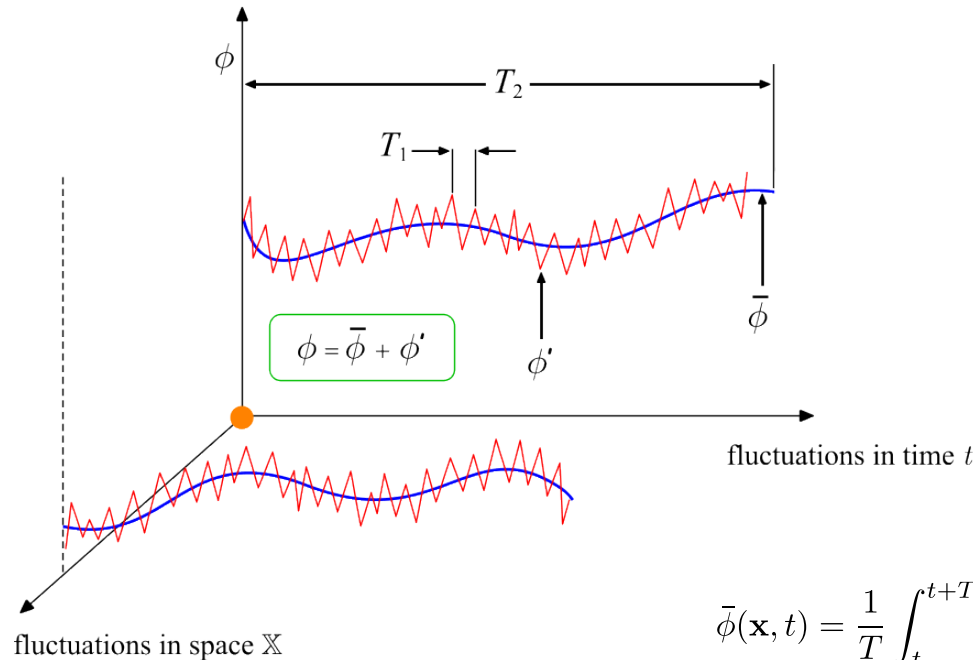


- The Reynolds number in this case is 100, for these conditions, the flow still is laminar.
- We are in the presence of the Von Karman vortex street, which is the periodic shedding of vortices caused by the unsteady separation of the fluid around blunt bodies.
- Vorticity is not a direct indication of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



# Introduction to turbulence modeling

## Turbulence modeling – Fluctuations of transported quantities

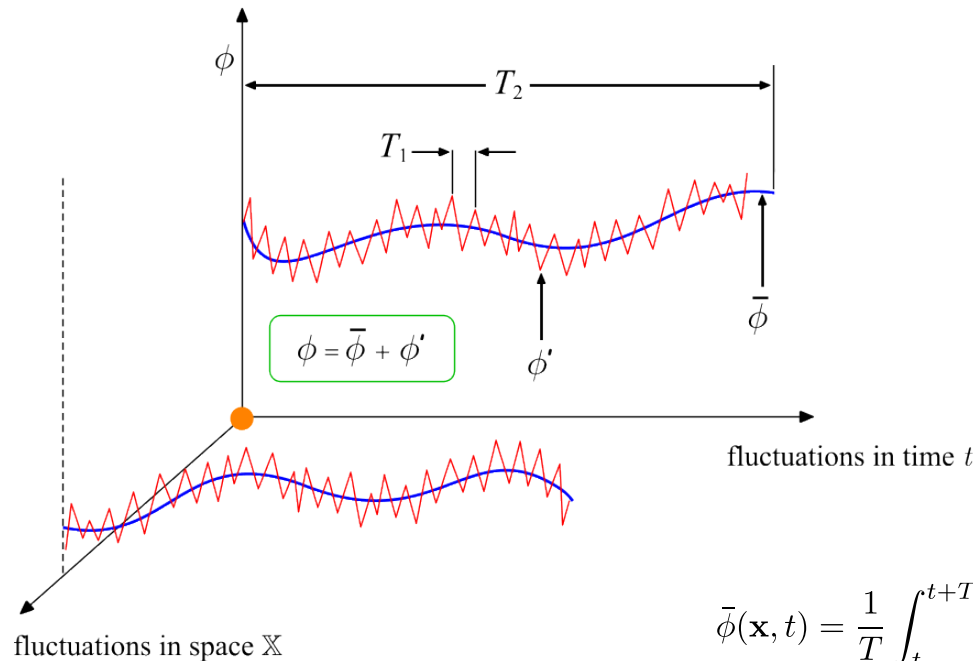


$$\bar{\phi}(\mathbf{x}, t) = \frac{1}{T} \int_t^{t+T} \phi(\mathbf{x}, t) dt, \quad T_1 \ll T \ll T_2$$

- 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, avoid, or filter the small scales by using models.

# Introduction to turbulence modeling

## Turbulence modeling – Fluctuations of transported quantities

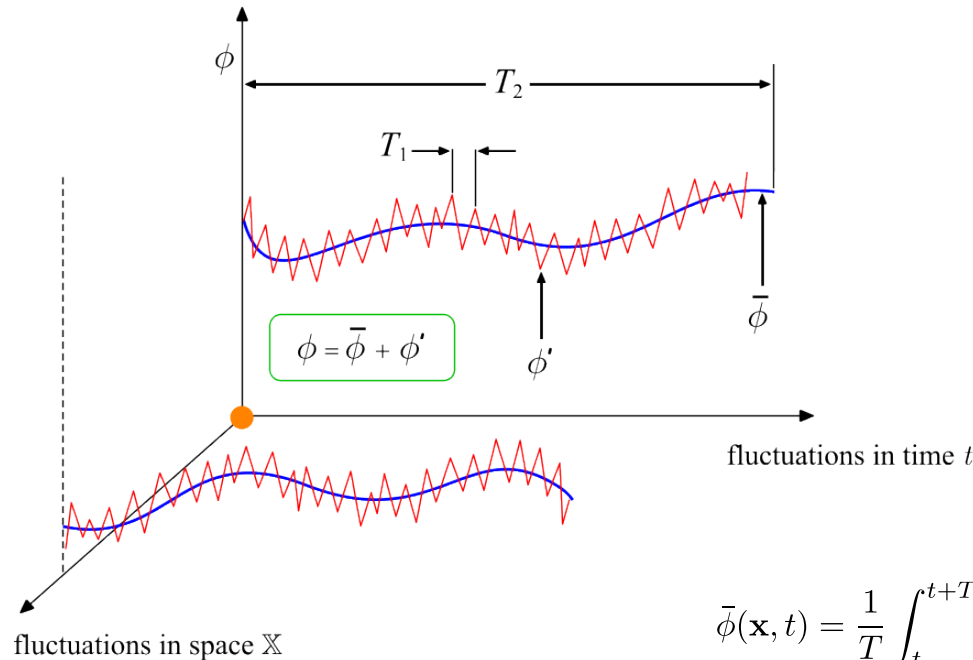


$$\bar{\phi}(\mathbf{x}, t) = \frac{1}{T} \int_t^{t+T} \phi(\mathbf{x}, t) dt, \quad T_1 \ll T \ll T_2$$

- To remove, avoid, or filter the instantaneous fluctuations (or small scales), two methods can be used: **Reynolds averaging** and **filtering** the governing equations.
- Both methods introduce additional terms that must be modeled for closure of the governing equations, this is the fundamental problem of Turbulence Modeling.
- We are going to talk about closure methods later.

# Introduction to turbulence modeling

## Turbulence modeling – Fluctuations of transported quantities

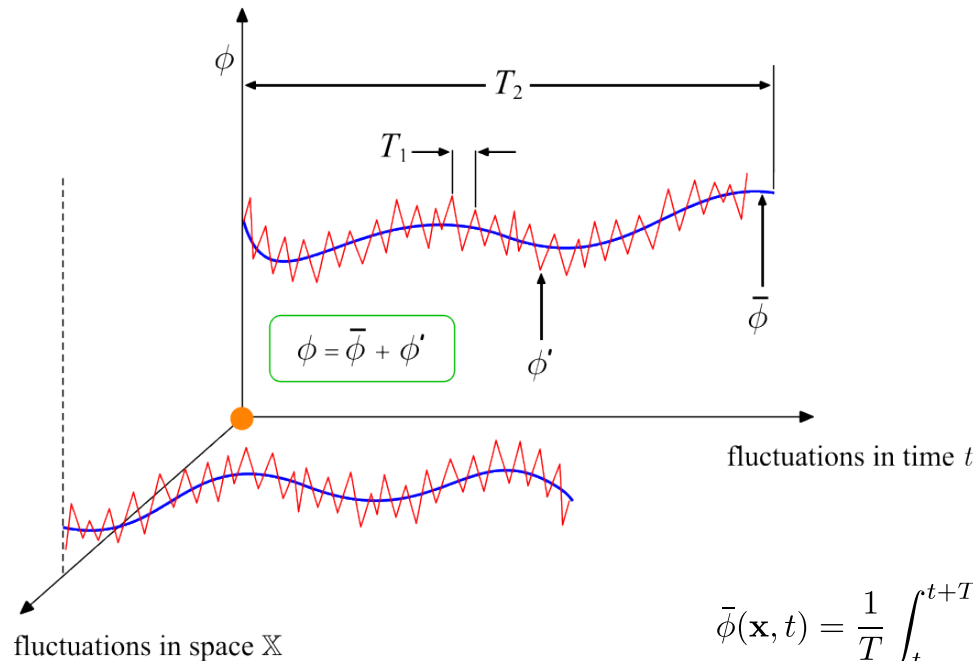


$$\bar{\phi}(\mathbf{x}, t) = \frac{1}{T} \int_t^{t+T} \phi(\mathbf{x}, t) dt, \quad T_1 \ll T \ll T_2$$

- The objective of turbulence modeling is to develop equations that predict the time averaged primitive fields (velocity, pressure, temperature, concentration, and so on), without calculating the complete turbulent flow pattern, from smallest to system scales, as a function of time.

# Introduction to turbulence modeling

## Turbulence modeling – Fluctuations of transported quantities

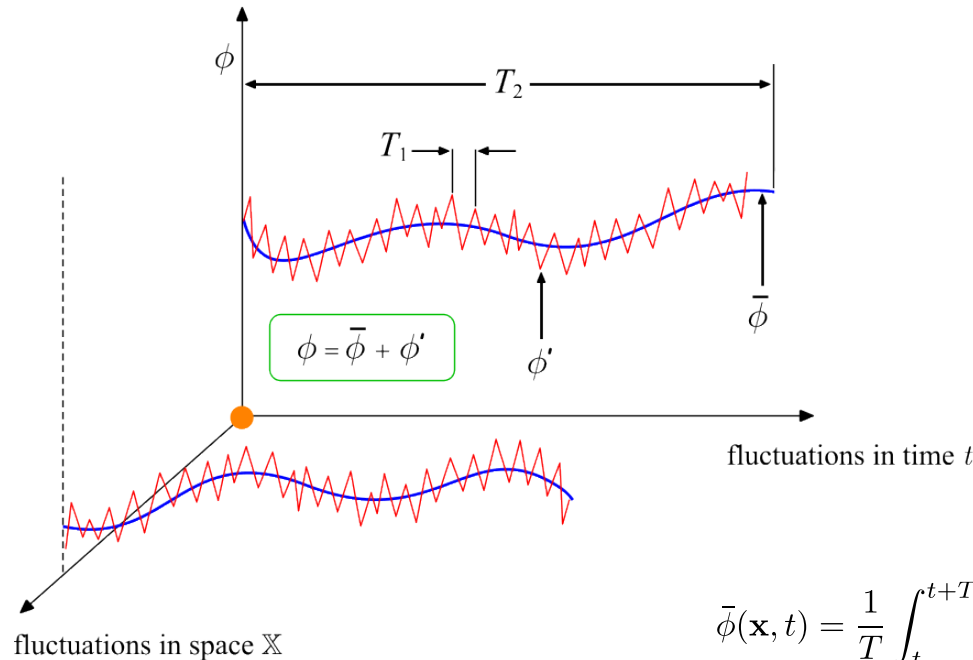


$$\bar{\phi}(\mathbf{x}, t) = \frac{1}{T} \int_t^{t+T} \phi(\mathbf{x}, t) dt, \quad T_1 \ll T \ll T_2$$

- In other words, we do not want to resolve all the space and time scales.
  - This reduces the computational time and resources.
  - Turbulence modeling is very reliable if you know what are you doing.

# Introduction to turbulence modeling

## Turbulence modeling – Fluctuations of transported quantities

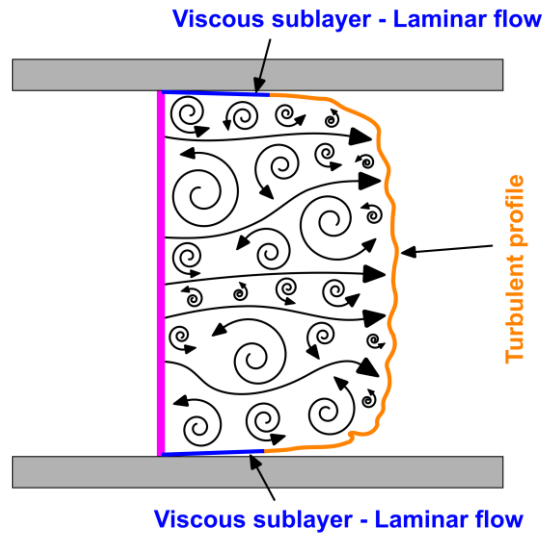


$$\bar{\phi}(\mathbf{x}, t) = \frac{1}{T} \int_t^{t+T} \phi(\mathbf{x}, t) dt, \quad T_1 \ll T \ll T_2$$

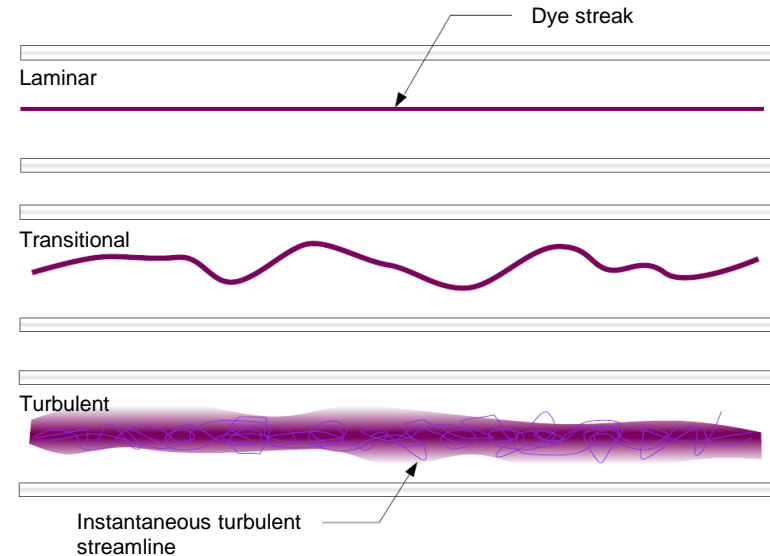
- Important to understand:
  - The time averaged flow pattern is a statistical property of the flow.
  - It is not the actual flow pattern.
  - The flow pattern changes from instant to instant.
  - In engineering applications, most of the time it is enough to know the average value.
  - However, we can also compute instantaneous values and calculate other statistical properties (e.g., RMS, two point-correlations, PDF, and so on).
- Later we are going to make a distinction between averaging stationary turbulence, averaging nonstationary turbulence, and statistically steady turbulence.

# Introduction to turbulence modeling

## Turbulence modeling – Wall bounded and shear free turbulence



Wall bounded turbulence



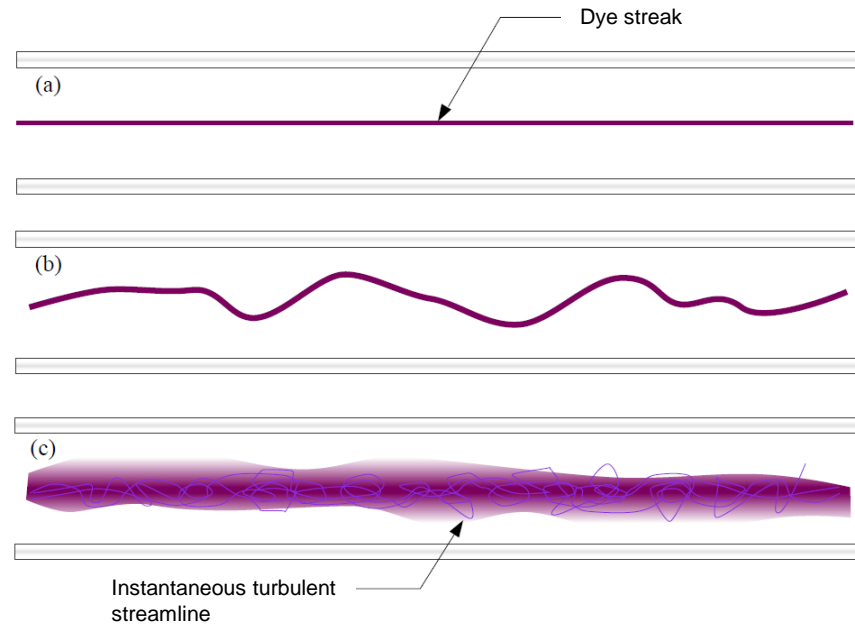
Shear free turbulence

- **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, mixing layers).
- But indifferently of the type of turbulence, that is, wall bounded or shear free, turbulent motion has a direct effect on the velocity profiles and mixing of the transported quantities (heat transfer rate, species concentration, mass transfer, and so on).



# Introduction to turbulence modeling

## Turbulence modeling – Wall bounded and shear free turbulence

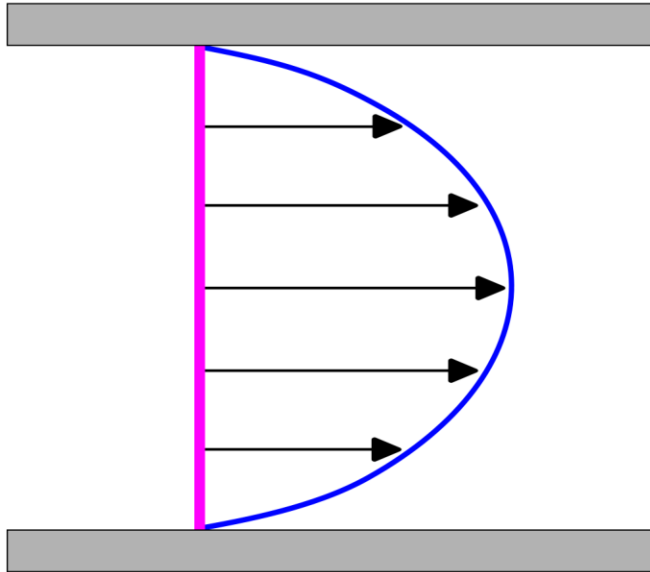


Flow in a pipe. (a) Laminar, (b) Transitional, (c) Turbulent

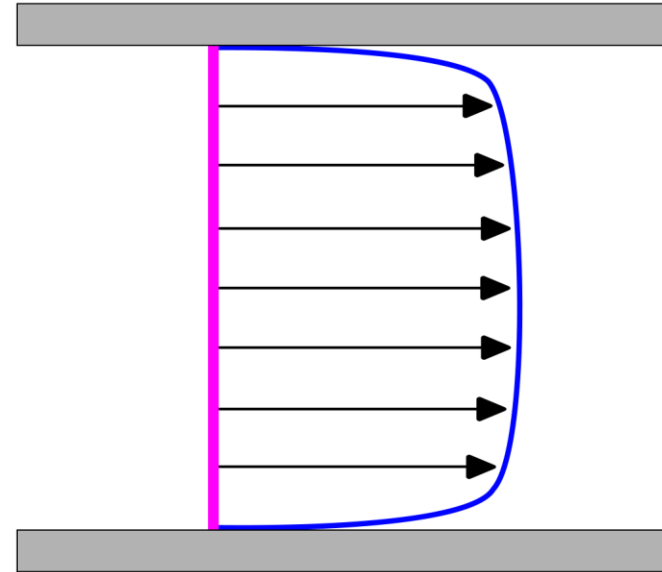
- Turbulence has a direct effect on the velocity profiles and mixing of transported quantities.
  - **Case (a)** correspond to a laminar flow, where the dye can mix with the main flow only via molecular diffusion, this kind of mixing can take very long times.
  - **Case (b)** shows a transitional state 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.
- This image give us an idea of what happens at the core of the flow, but what about the walls?

# Introduction to turbulence modeling

## Turbulence modeling – Wall bounded and shear free turbulence



Laminar flow in a pipe

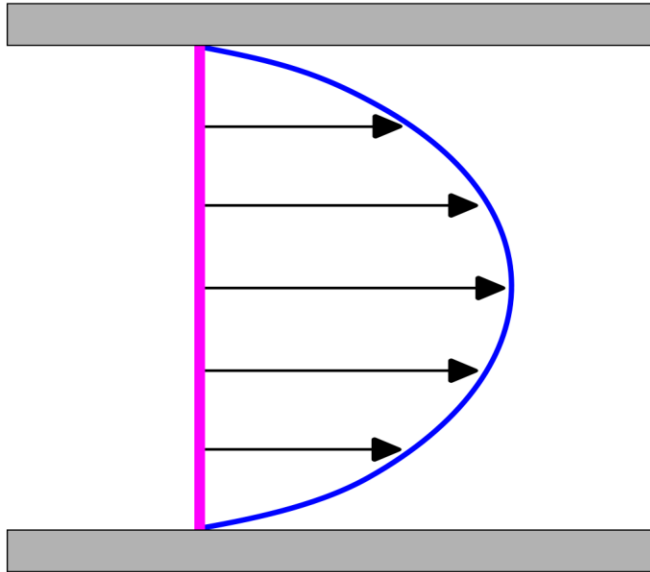


Turbulent flow in a pipe

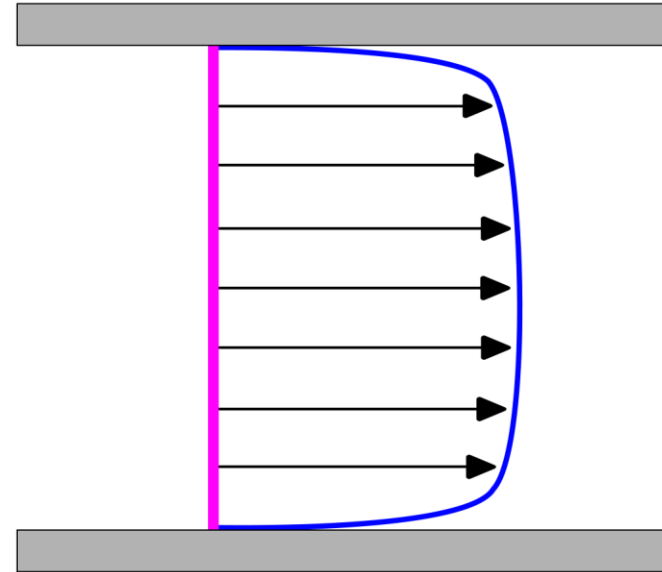
- Turbulence has a direct effect on the velocity profiles and mixing of transported quantities.
  - In the **laminar case**, the velocity gradient close to the walls is small (therefore the shear stresses are lower).
  - The **turbulent case** shows two regions. One thin region close to the walls with 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, random fluctuations of the velocity field and transported quantities are present.

# Introduction to turbulence modeling

## Turbulence modeling – Wall bounded and shear free turbulence



Laminar flow in a pipe



Turbulent flow in a pipe

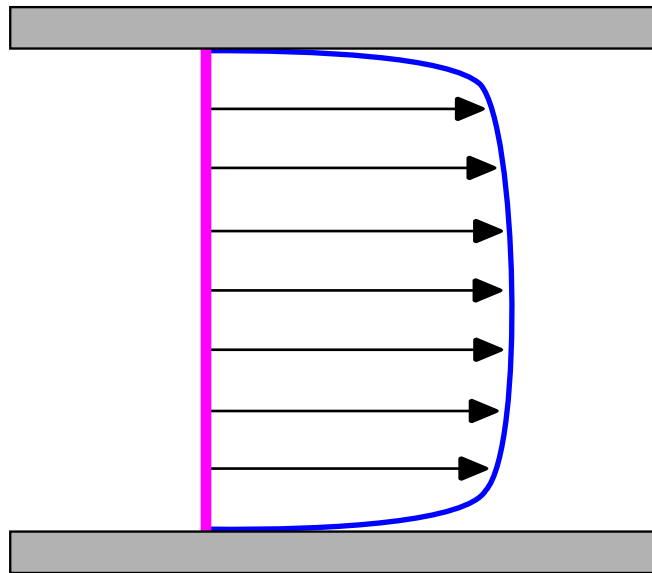
- Turbulence has a direct effect on the velocity profiles and mixing of transported quantities.
  - The larger the velocity gradient close to the walls, the larger the wall shear stresses will be.
  - Recall that the wall shear stress  $\tau_{wall}$  for laminar flows or in the viscous sub-layer\* is equal to:

$$\tau_{wall} = \mu \left. \frac{\partial u}{\partial y} \right|_{y=0}$$

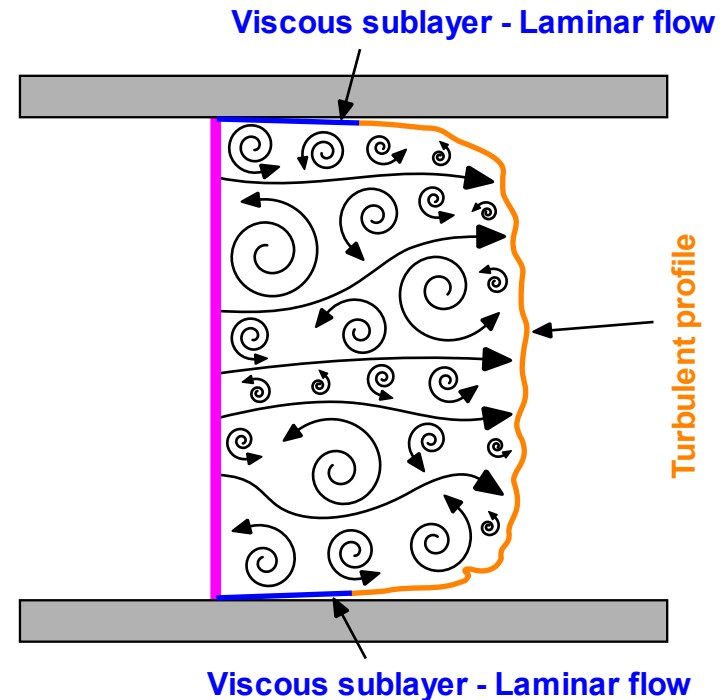
\* The viscous sub-layer is region of the turbulent boundary layer very close to the wall where the flow is laminar.

# Introduction to turbulence modeling

## Turbulence modeling – Wall bounded and shear free turbulence



Averaged turbulent flow

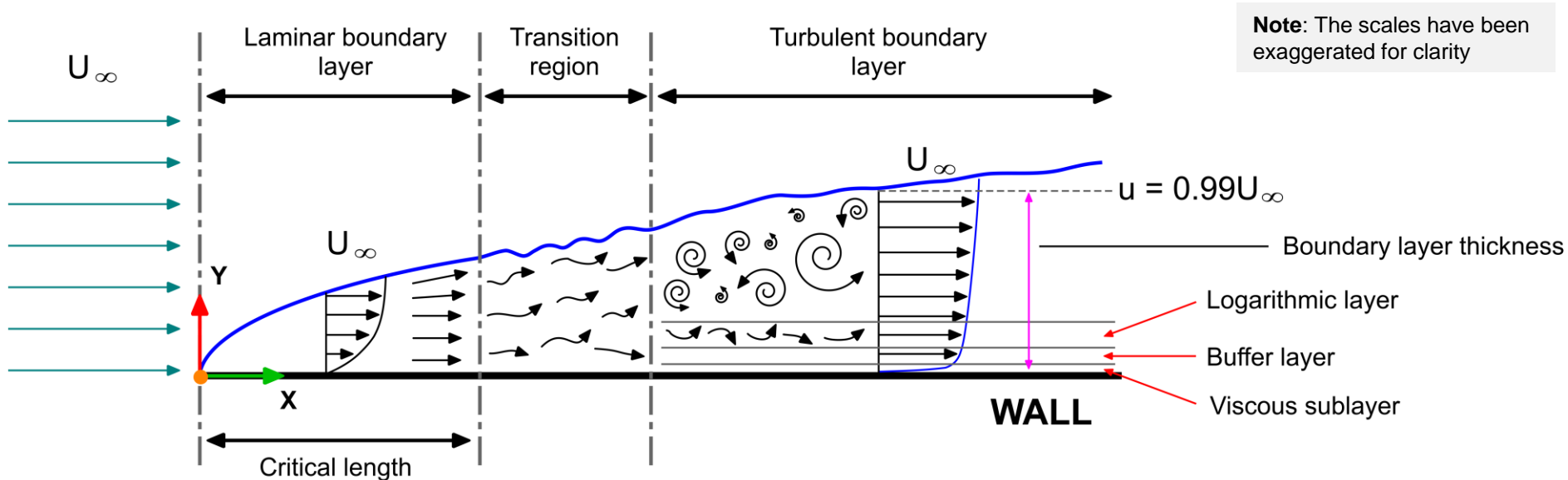


Instantaneous turbulent flow

- 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 increases the wall shear stresses and enhances mixing.

# Introduction to turbulence modeling

## 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 gradients 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.

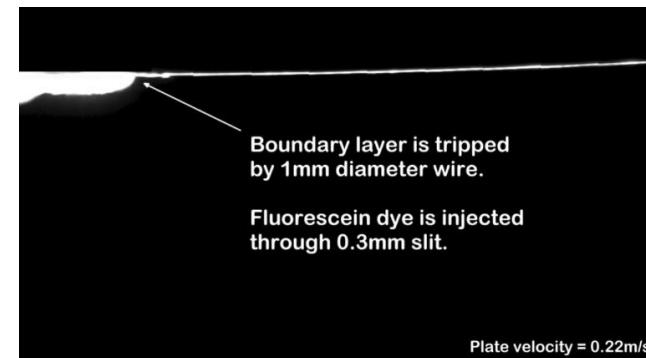
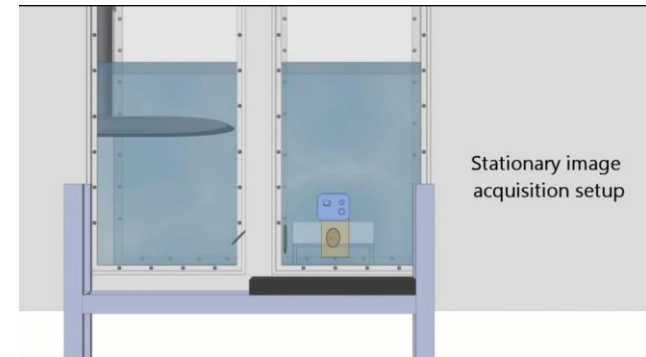
# Introduction to turbulence modeling

## Turbulence modeling – Boundary layer and fluctuations of transported quantities



Turbulent boundary layer on a flat plate

[www.wolfdynamics.com/training/turbulence/video-bl1.mp4](http://www.wolfdynamics.com/training/turbulence/video-bl1.mp4)



**Video credit:** <https://www.youtube.com/watch?v=e1TbkLIDWys>

**arXiv:** <https://arxiv.org/abs/1210.3881>

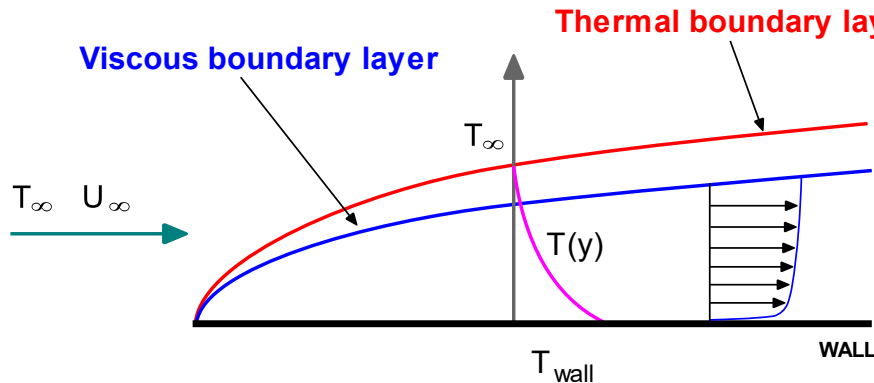
This video has been edited to fit this presentation.

Copyright on the images is held by the contributors. Apart from Fair Use, permission must be sought for any other purpose.

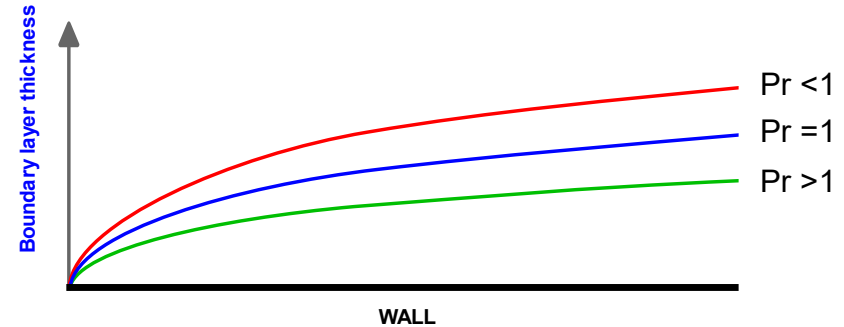


# Introduction to turbulence modeling

## Turbulence modeling – Thermal boundary layer



Thermal boundary layer vs. Viscous boundary layer  
Forced convection



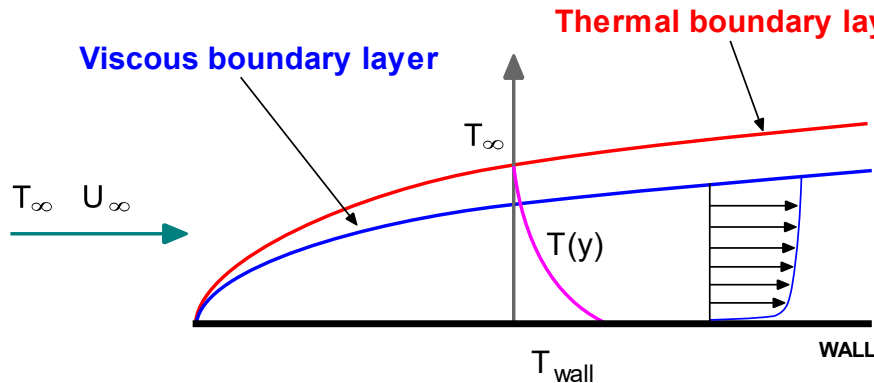
Thermal boundary layer in function of Prandtl number (Pr)

## Momentum and thermal boundary layer

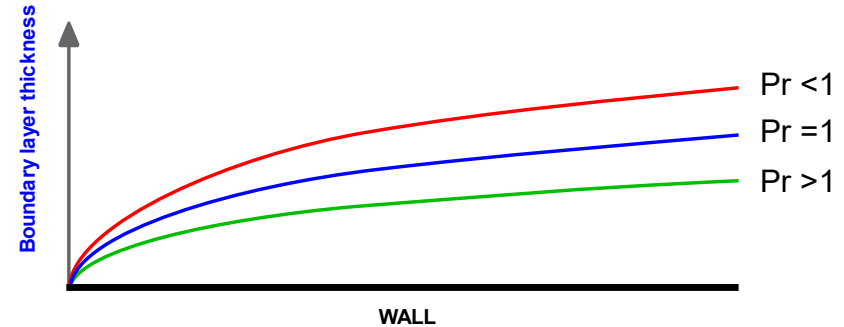
- Just as there is a viscous boundary layer in the velocity distribution (or momentum), there is also a thermal boundary layer.
- Thermal boundary layer thickness is different from the thickness of the viscous sublayer (momentum), and it is fluid dependent.
- The thickness of the thermal sublayer for 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.

# Introduction to turbulence modeling

## Turbulence modeling – Thermal boundary layer



Thermal boundary layer vs. Viscous boundary layer  
Forced convection



Thermal boundary layer in function of Prandtl number (Pr)

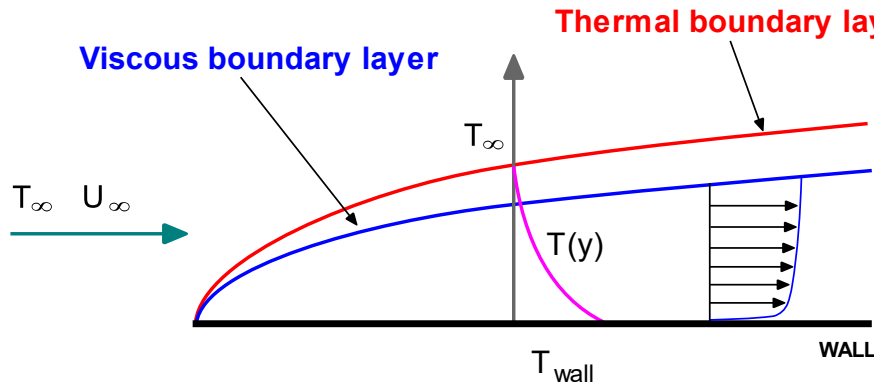
## Momentum and thermal boundary layer

- Note that the thickness of the thermal sublayer for a high Prandtl number fluid (e.g., water) is much less than the momentum sublayer thickness.
- Therefore, the mesh requirements close to the walls need to be corrected for the thinner thermal boundary layer.
- Using the following relationships, the distance from to wall to the first mesh node normal to the wall (in function of  $y^+$ ) can be corrected as follows,

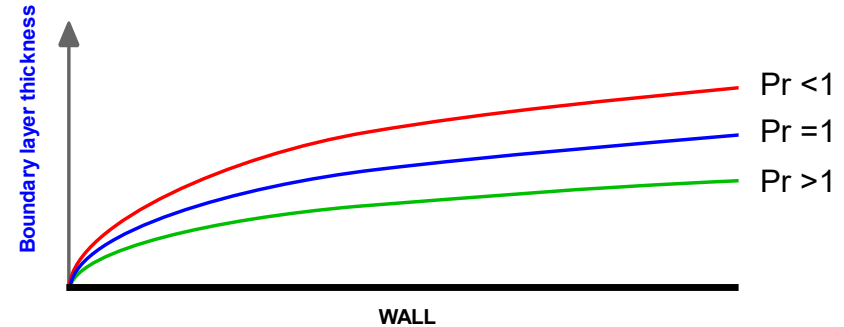
$$\frac{\delta_{\text{hydrodynamic b.l.}}}{\delta_{\text{thermal b.l.}}} \approx \frac{y_{\text{viscous s.l.}}}{y_{\text{thermal s.l.}}} \approx \sqrt{Pr} \quad \text{therefore} \quad y^+ \approx \frac{1}{\sqrt{Pr}}$$

# Introduction to turbulence modeling

## Turbulence modeling – Thermal boundary layer



Thermal boundary layer vs. Viscous boundary layer  
Forced convection



Thermal boundary layer in function of Prandtl number (Pr)

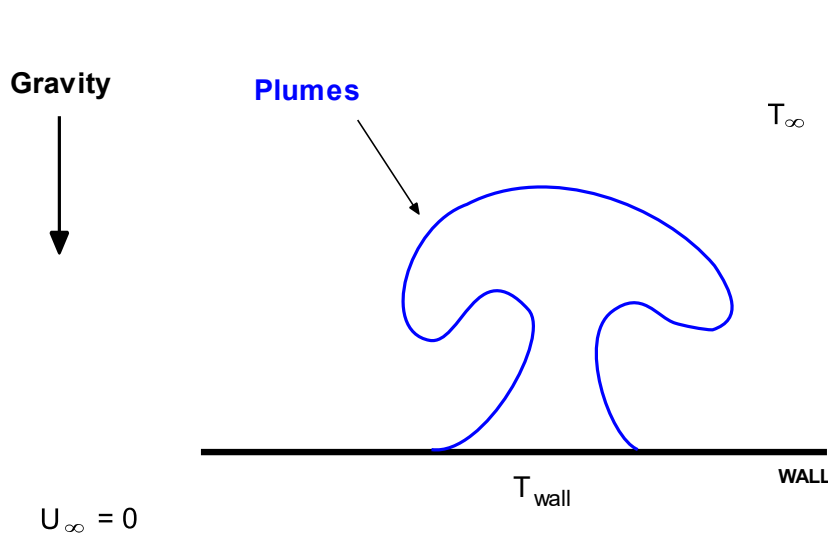
## Momentum and thermal boundary layer

- A similar situation exists when working with multispecies transport (e.g., combustion) and when using the Schmidt number (Sc).
- If the Schmidt number is considerably larger than unity, then the thickness of the thermal diffusion sublayer is much less than the momentum sublayer thickness.
- Using the Schmidt number, the distance from to wall to the first mesh node normal to the wall (in function of  $y^+$ ) can be corrected as follows,

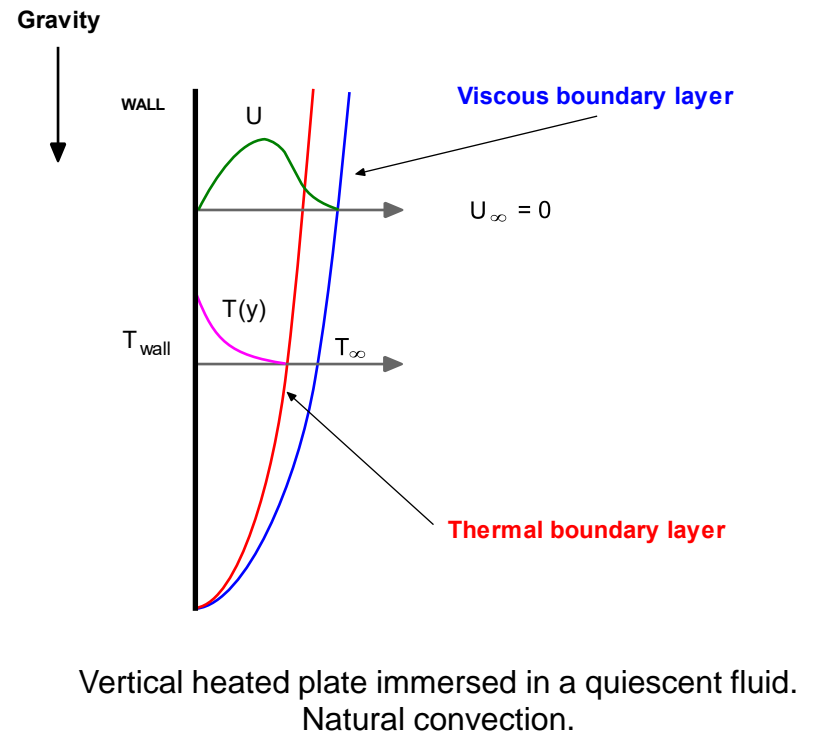
$$y^+ \approx \frac{1}{\sqrt[3]{Sc}}$$

# Introduction to turbulence modeling

## Turbulence modeling – Thermal boundary layer



Horizontal heated plate immersed in a quiescent fluid.  
Natural convection



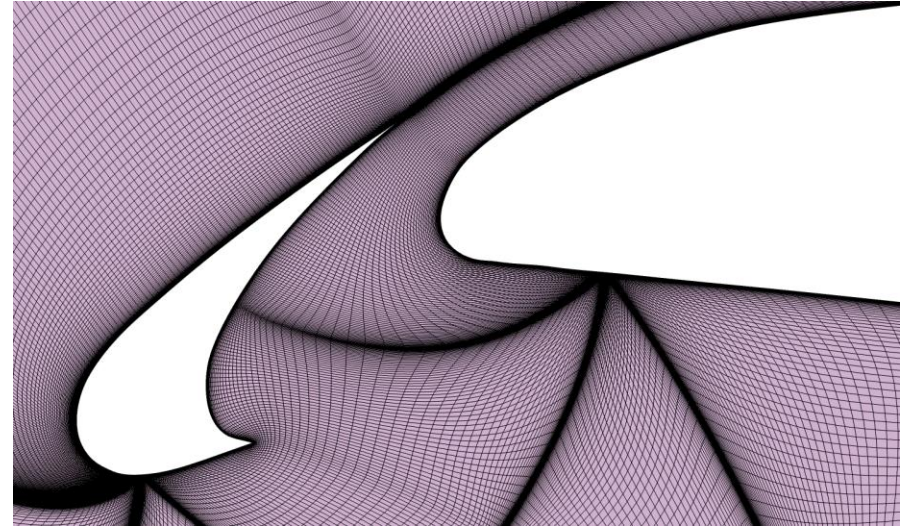
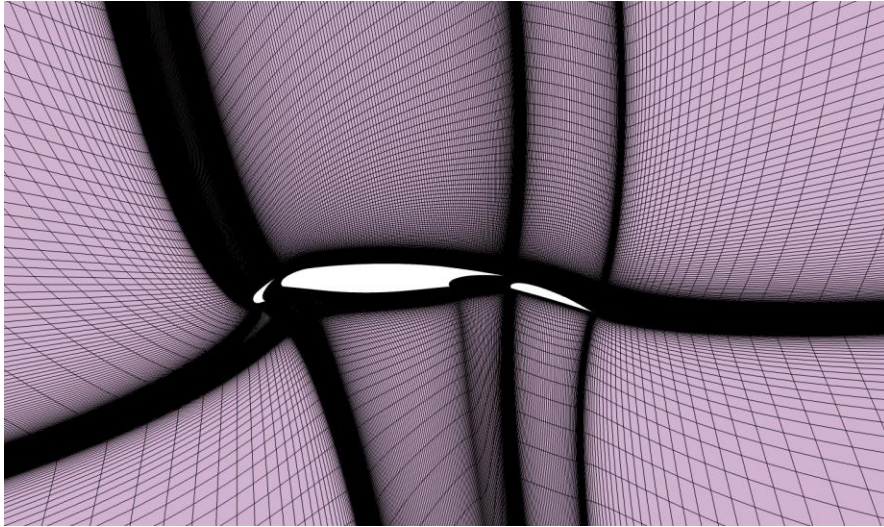
Vertical heated plate immersed in a quiescent fluid.  
Natural convection.

### 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.

# Introduction to turbulence modeling

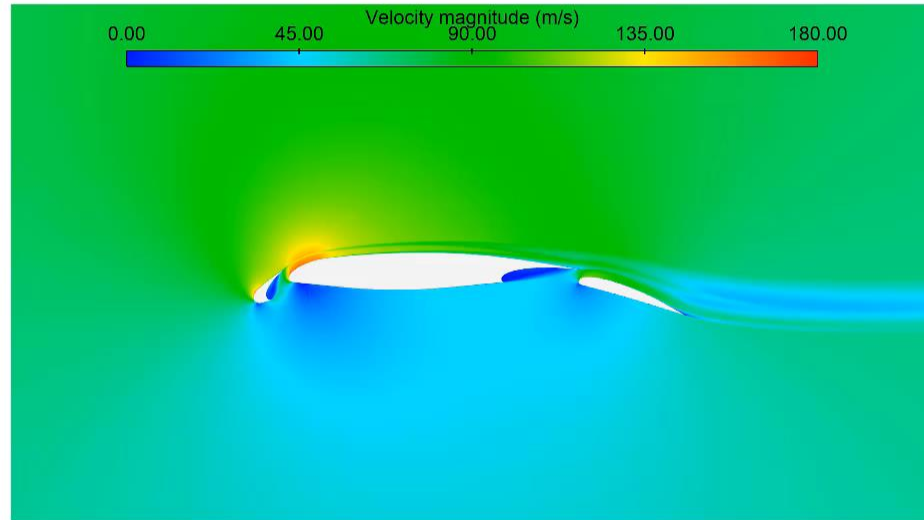
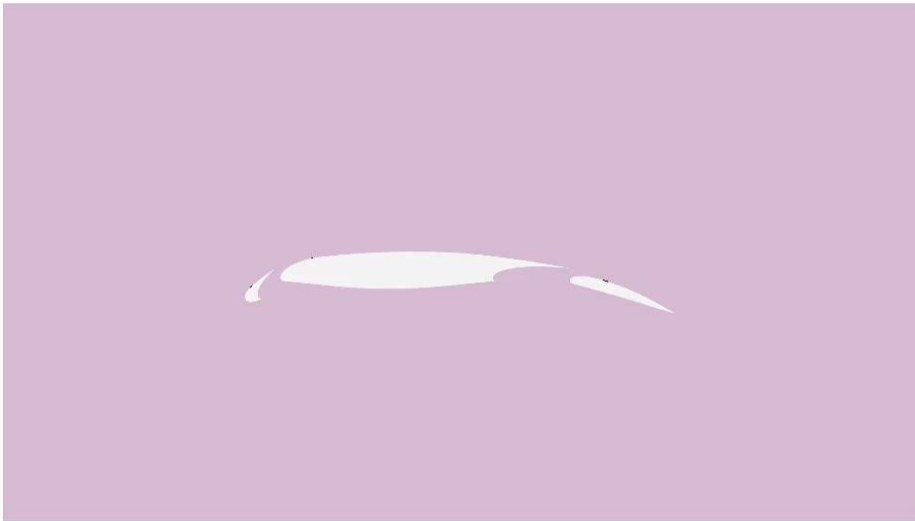
## Turbulence modeling – Laminar separation bubbles and transition to turbulence



- Maybe, the most challenging topic of turbulence modeling is the prediction of transition to turbulence.
- Trying to predict transition to turbulence in CFD requires very fine meshes and well calibrated models.
- Traditional turbulence models assume that the boundary layer is turbulent in all its extension.
- But assuming that the boundary layer is entirely turbulent might not be a good assumption, as in some regions the boundary layer might still be laminar, so we may be overpredicting drag forces or predicting wrong separation points.
- In many applications, transition to turbulence is preceded by laminar separation bubbles (LSB), which are laminar recirculation areas that separate from the wall and reattach in a very short distance and are very sensitive to disturbances.
- After the LSB, the flow becomes turbulent.

# Introduction to turbulence modeling

## Turbulence modeling – Laminar separation bubbles and transition to turbulence



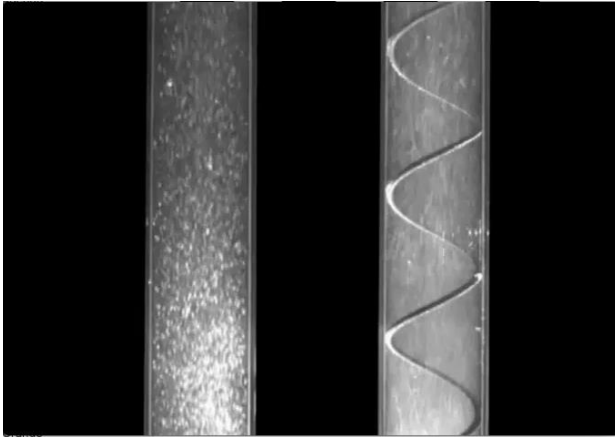
Laminar separation bubbles

<http://www.wolfdynamics.com/images/airfoil.mp4>

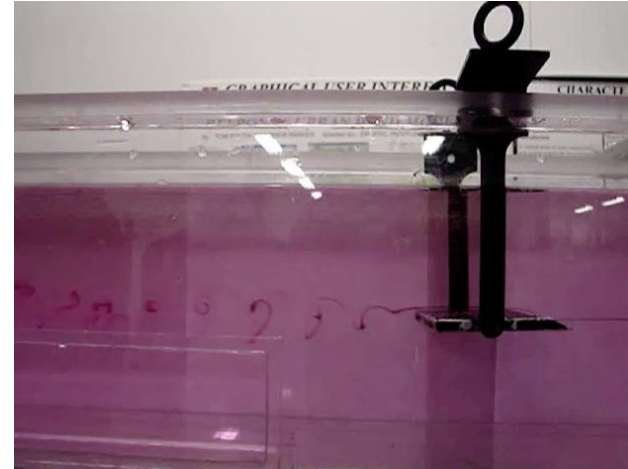


# Introduction to turbulence modeling

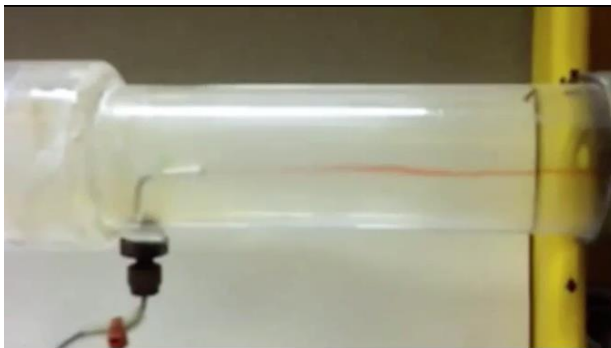
## Turbulence flows videos



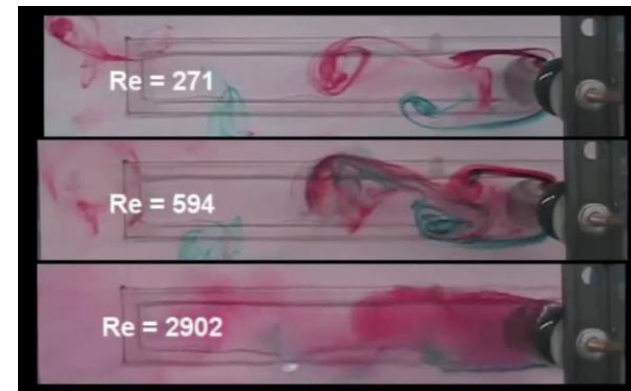
Laminar-Turbulent flow in a pipe  
Smooth vs corrugated tube



Flow over a flat plate  
Attached and separated boundary layer



Various types of flow in a pipe  
Laminar-Transitional-Turbulent



Laminar-Turbulent vortex shedding

# Introduction to turbulence modeling

## On the use of dimensional analysis in turbulence modeling

- Historically, dimensional analysis has been one of the most powerful tools available for deducing and correlating properties of turbulent flows.
- As we will see later, eddy viscosity and length scales are all related on the basis of dimensional arguments.
- However, we should always be aware that while dimensional analysis is extremely useful, it unveils nothing about the physics underlying its implied relationships.
- The physics is in the choice of the variables and the correctness of the hypotheses taken.
- Later, we will enumerate a few important relations.
- Many of these relations are based on dimensional analysis.
- So, we invite you to dust your notes.
- At this point, let us remind you a few base and derived quantities that we will use.

# Introduction to turbulence modeling

## On the use of dimensional analysis in turbulence modeling

Base quantity	Symbol	Dimensional units	SI units
Length	-	L	m
Mass	-	M	kg
Time	-	T	s

Derived quantity	Symbol	Dimensional units	SI units
Velocity	-	$LT^{-1}$	m/s
Density	$\rho$	$ML^{-3}$	kg/m <sup>3</sup>
Kinematic viscosity	$\nu$	$L^2T^{-1}$	m <sup>2</sup> /s
Dynamic viscosity	$\mu$	$ML^{-1}T^{-1}$	kg/m-s
Energy dissipation rate per unit mass	$\epsilon$	$L^2T^{-3}$	m <sup>2</sup> /s <sup>3</sup>
Turbulent kinetic energy per unit mass	$k$	$L^2T^{-2}$	m <sup>2</sup> /s <sup>2</sup>
Wavenumber	$\kappa$	$L^{-1}$	1/m
Energy spectral density per wavenumber	$E(\kappa)$	$L^3/T^2$	m <sup>3</sup> /s <sup>2</sup>
Stress	$\mathcal{T}$	$ML^{-1}T^{-2}$	kg/(m-s <sup>2</sup> ) or N/m <sup>2</sup> or Pa

# Introduction to turbulence modeling

## Turbulence modeling – Scales of turbulence

- According to Kolmogorov's universal equilibrium theory (K41), the motion at the smallest scales should depend only upon:

- The rate at which the larger eddies supply energy,

Turbulent dissipation rate  $\longrightarrow \varepsilon = \frac{-dk}{dt}$   $\longleftarrow$  Turbulent kinetic energy

- The kinematic viscosity  $\nu$ .

- By using dimensional analysis and similarity hypothesis (and a lot of intuition), Kolmogorov derived the turbulence microscales:

$$\eta = \left( \frac{\nu^3}{\varepsilon} \right)^{1/4}$$

Length scale

$$\tau_\eta = \left( \frac{\nu}{\varepsilon} \right)^{1/2}$$

Time scale

$$u_\eta = (\nu \varepsilon)^{1/4}$$

Velocity scale

- These scales are indicative of the smallest eddies, that is, the scale at which the energy is dissipated.

For a complete derivation please refer to:

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

S. Pope. Turbulent Flows, Cambridge University Press, 2000.

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

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

# Introduction to turbulence modeling

## Turbulence modeling – Scales of turbulence

- From the previous equations, we can derive the following relations

$$\frac{u_\eta}{\eta} = \frac{1}{\tau_\eta} \quad \text{and} \quad Re_\eta = \frac{\eta u_\eta}{\nu} = 1$$

- Where  $Re_\eta$  is the Kolmogorov Reynolds number.
- The fact that the Kolmogorov Reynolds number is equal to 1, is consistent with the notion that the energy cascade proceeds to smaller and smaller scales until the Reynolds number is small enough for dissipation due to viscosity to be effective.
- From the Kolmogorov turbulence microscales, it can be seen that for large Reynolds number the length, time, and velocity scales of the smallest eddies are small compared to those of the largest eddies.

Largest eddies	→	$l_0 \sim Re_T^{3/4}$	$\frac{\tau_0}{\tau_\eta} \sim Re_T^{1/2}$	$\frac{u_0}{u_\eta} \sim Re_T^{1/4}$
Smallest eddies	→	$\eta$		

- Where  $l_0 = \frac{k^{3/2}}{\varepsilon}$ ,  $\tau_0 = t = \frac{k}{\varepsilon}$ , and  $Re_T \equiv \frac{k^{1/2} l_0}{\nu}$  is the turbulence Reynolds number

# Introduction to turbulence modeling

## Turbulence modeling – Implications of scales

- The previous relationships indicate that the range of turbulent scales may span orders of magnitude for high Reynolds number flows.
- Since the turbulent length scales  $\eta$  are much smaller than the large or integral scales (*i.e.*, wing chord, channel height, blockage ratio) associated with the flow of interest, it is easy to see that a large amount of grid points/cells would be required to fully simulate high Reynolds number flows.
- For example, in a direct numerical simulation (DNS) where all scales of turbulence are resolved, the number of grid points/cells in one direction is directly proportional to  $Re_T^{3/4}$ .
- Thus, in a DNS simulation the gridding requirements scale proportional to  $Re_T^{9/4}$  or approximately proportional to  $Re_T^3$  for a single time step.
- And the number of time steps to satisfy a Courant condition of  $CFL < 1$ , is on the order of  $O(Re_T^{3/4})$ .
- As you can see, resolving all turbulence scales (or DNS simulations) requires a formidable amount of computational power in CFD.
- Therefore, the importance of using turbulence models to alleviate the incredible requirements of resolving all turbulence scales.

# Introduction to turbulence modeling

## Turbulence modeling – Grid requirements

- The grid requirements of turbulence modeling in CFD, can be summarize as follows:
  - DNS simulations requires no modeling, but it demands resolution from the large scales all the way through at least the beginning of the dissipation scales. This results in a grid scaling proportional to  $Re_T^3$ , or worse.
  - LES simulations requires modeling of part of the inertial subrange and into the beginning of the dissipation scales. The amount of required modeling is set by the grid resolution but is unlikely that the grid will scale worse than  $Re_T^2$ . Even if this requirement appears to be high, LES simulations are starting to become affordable in modern supercomputers.
  - RANS simulations require modeling of everything, from the integral scales into the dissipation range. As a consequence, the grid scaling is a weak function of the system Reynolds number.
- As you can see, RANS simulations are very affordable (steady and unsteady).
- RANS simulations are the workhorse of turbulence modeling in industrial applications.
- Nevertheless, LES and DES are becoming more affordable.
- DNS is only use in research and for low Reynolds number.



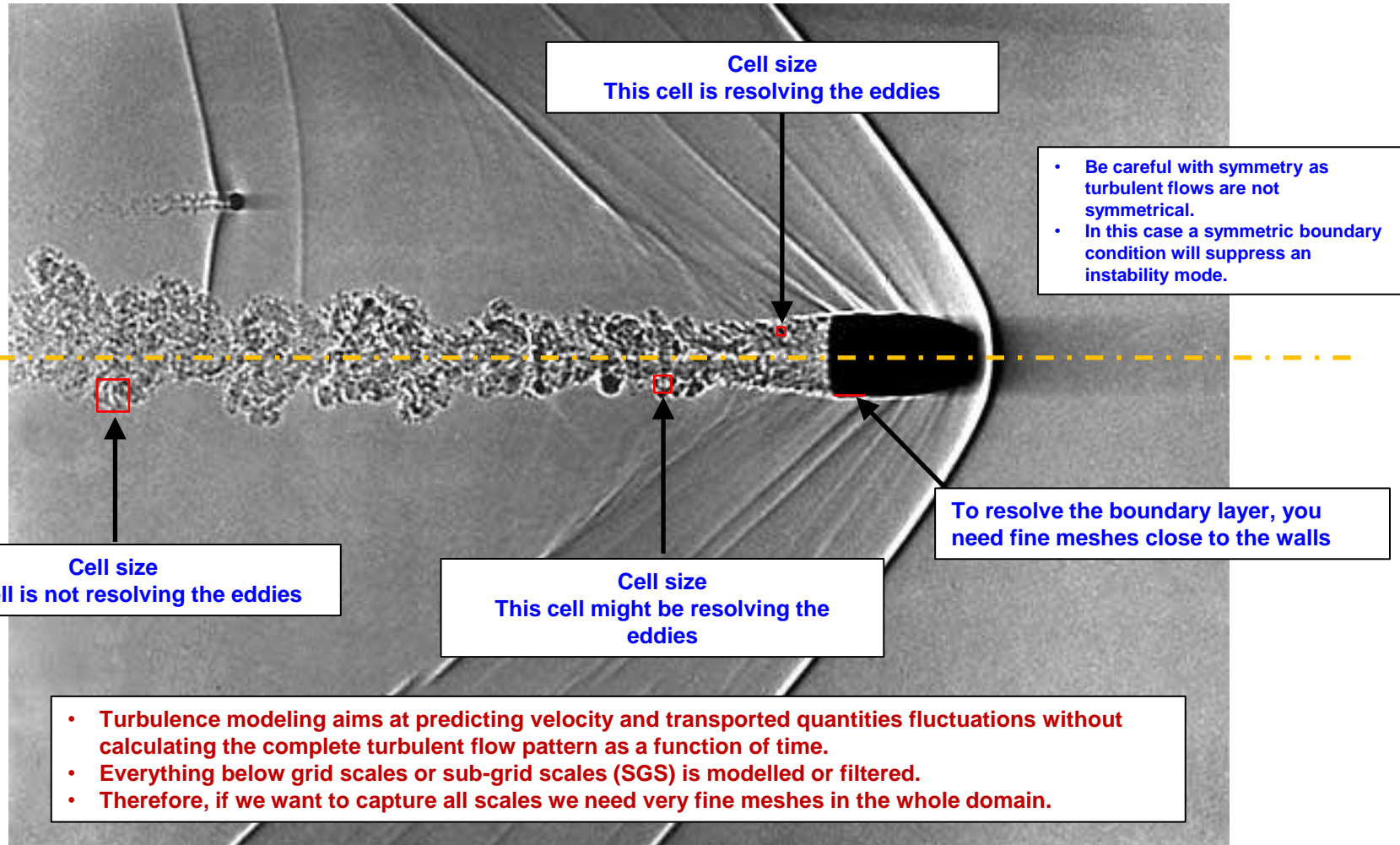
# Introduction to turbulence modeling

## Turbulence modeling – Grid requirements

- As you can see, resolving all turbulence scales in CFD (in space and time) requires a formidable amount of computational power and user time.
- To this, you need to add the IO overhead, storage requirements, and the qualitative and quantitative post-processing, which can be as expensive as the simulations.
- Therefore, the importance of using turbulence models to alleviate the incredible requirements of resolving all turbulence scales.

# Introduction to turbulence modeling

## Turbulence modeling – Grid requirements



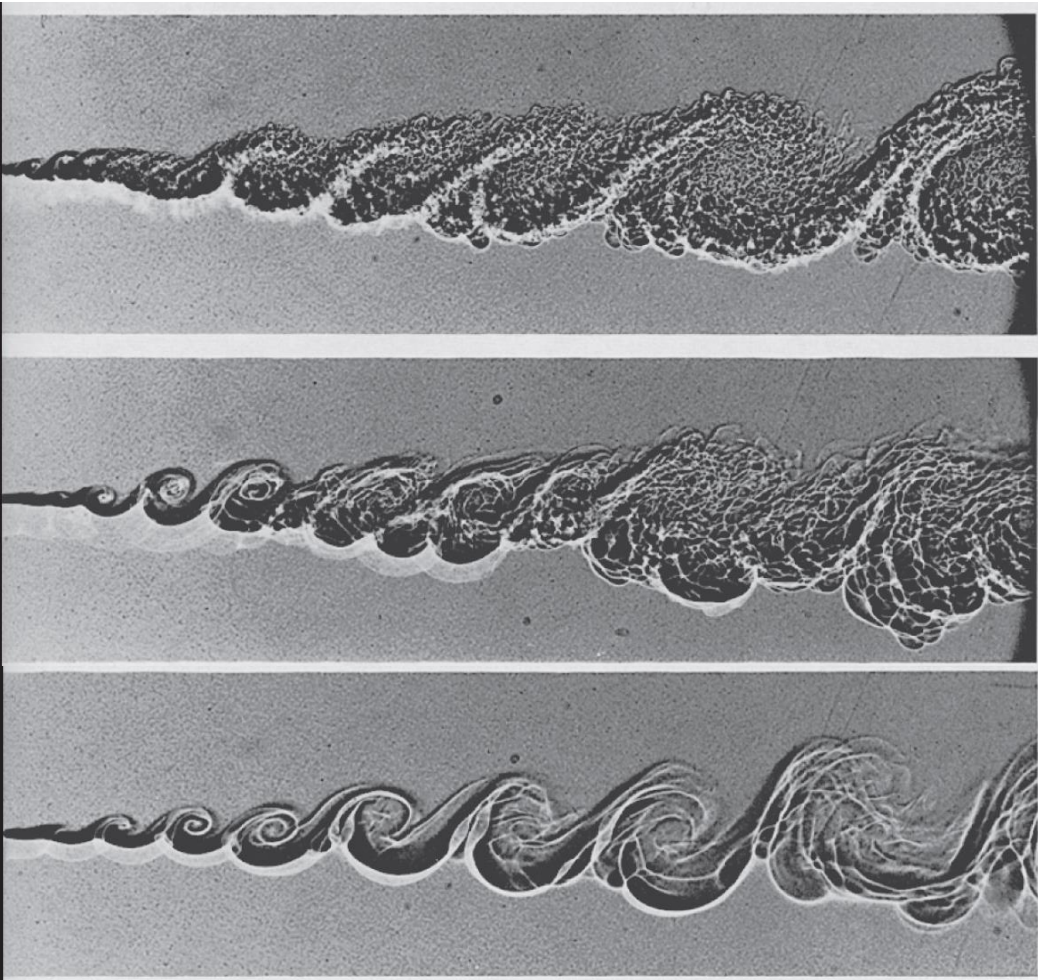
**Bullet at Mach 1.5**

Photo credit: Andrew Davidhazy. Rochester Institute of Technology.

Copyright on the images is held by the contributors. Apart from Fair Use, permission must be sought for any other purpose.

# Introduction to turbulence modeling

## Turbulence modeling – Grid scales

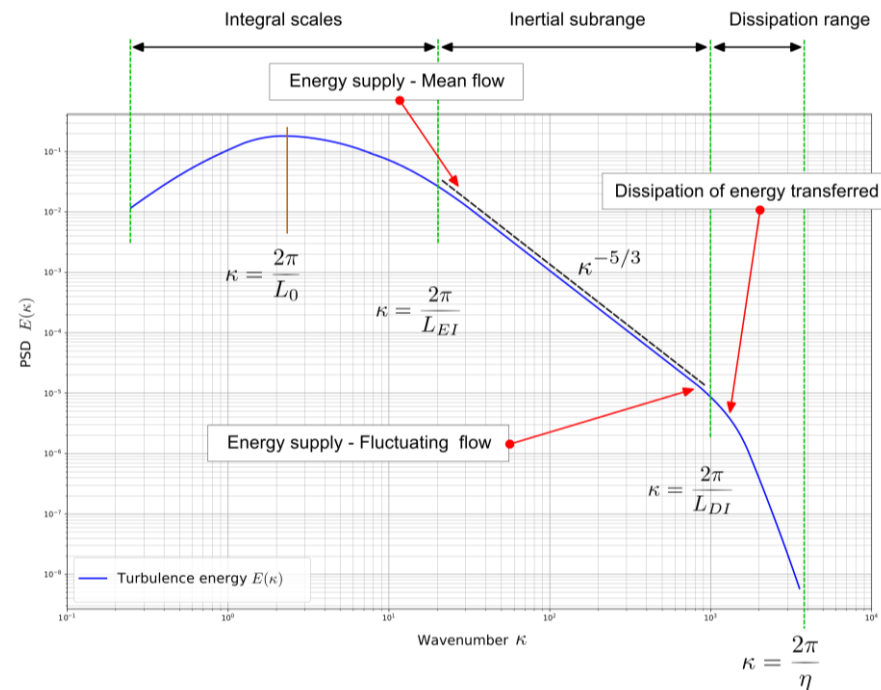


Increasing Reynolds number

- The large integral scales are roughly the same as the Reynolds number increases.
- However, at larger Reynolds number there are more microscales.
- To resolve the large integral scales in all images, we can use the same mesh.
- However, to resolve the smallest microscales at large Reynolds number, we need very fine meshes.
- As it can be seen, at large Reynolds number, the ratio between the largest and smallest scale is very large.

# Introduction to turbulence modeling

## Energy spectrum and energy cascade



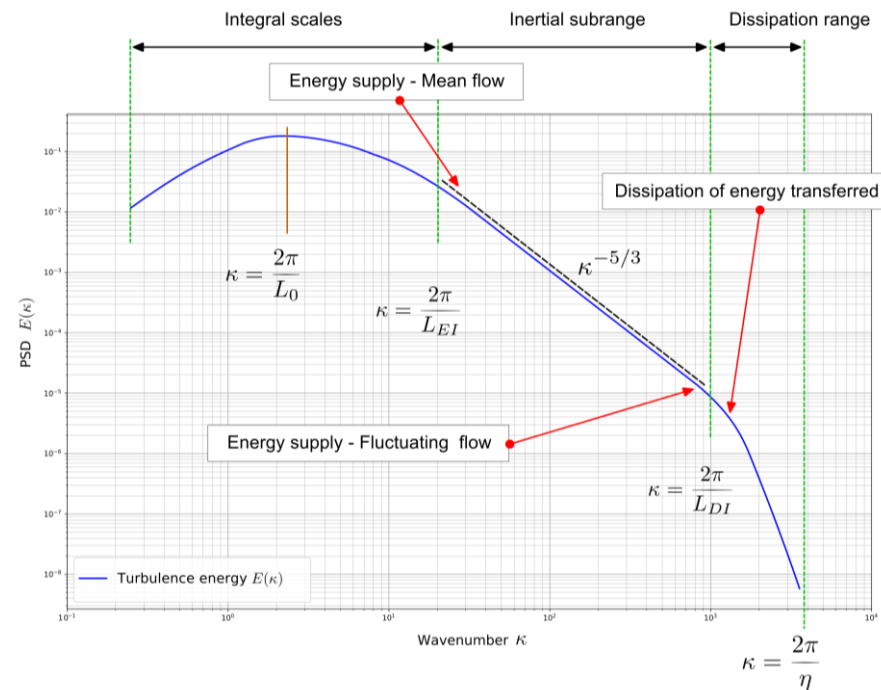
- The existence of a wide separation of scales is a central assumption Kolmogorov made as part of his universal equilibrium theory.
- Kolmogorov hypothesized that there is a range of eddy sizes between the largest and smallest for which the cascade process is independent of the statistics of the energy containing eddies and of the effect of viscosity.
- As a consequence, a range of wavenumbers exists in which the energy transferred by inertial effects, wherefore  $E(\kappa)$  depends only upon  $\epsilon$  and  $\kappa$ .
- On dimensional grounds, Kolmogorov concluded that,

$$E(\kappa) = C_K \epsilon^{2/3} \kappa^{-5/3}$$

- This relationship is sometimes known as the Kolmogorov -5/3 law.

# Introduction to turbulence modeling

## Energy spectrum and energy cascade

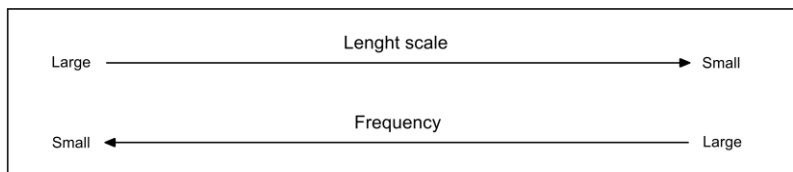
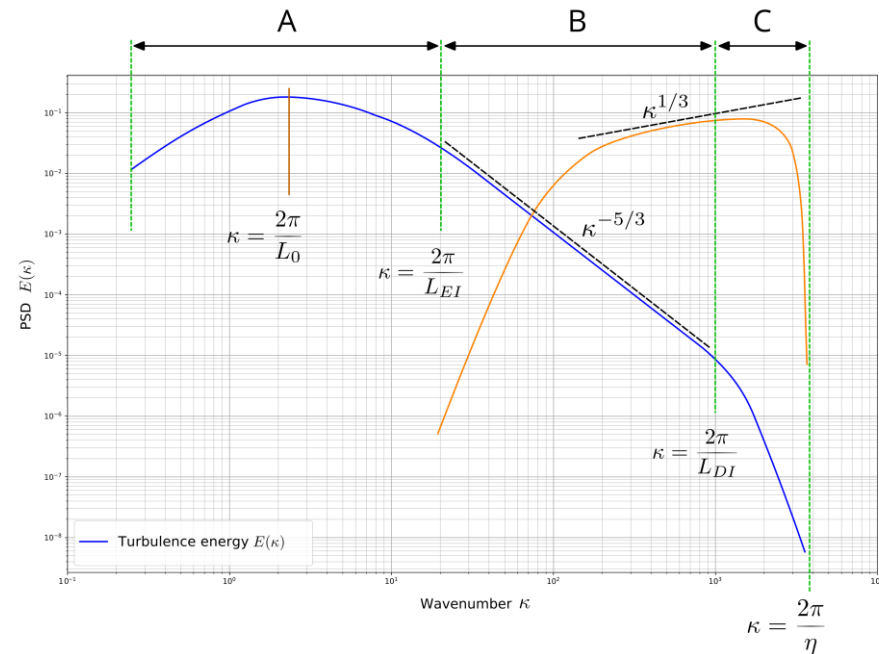
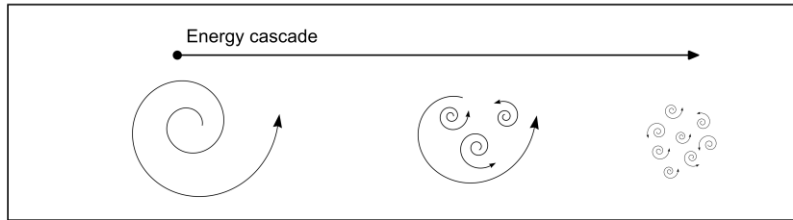


- Let us talk about the mean kinetic energy (that of the mean flow) and turbulent kinetic energy (that related to the velocity fluctuations or small scales).
- Large scales (mean flow), are very energetic. They supply energy to the flow. Therefore, large scales loss energy.
- This energy is transferred at a constant rate in the inertial subrange.
- The energy loss of the mean flow is an energy gain of the turbulent kinetic energy.
- At the end of the inertial subrange, the turbulent kinetic energy is dissipated.
- It is worth noting that we have assumed the energy transfer follows one direction.
- However, the flow of energy from small scales to large scales has been observed and documented. This is known as backscatter.



# Introduction to turbulence modeling

## Energy spectrum and energy cascade



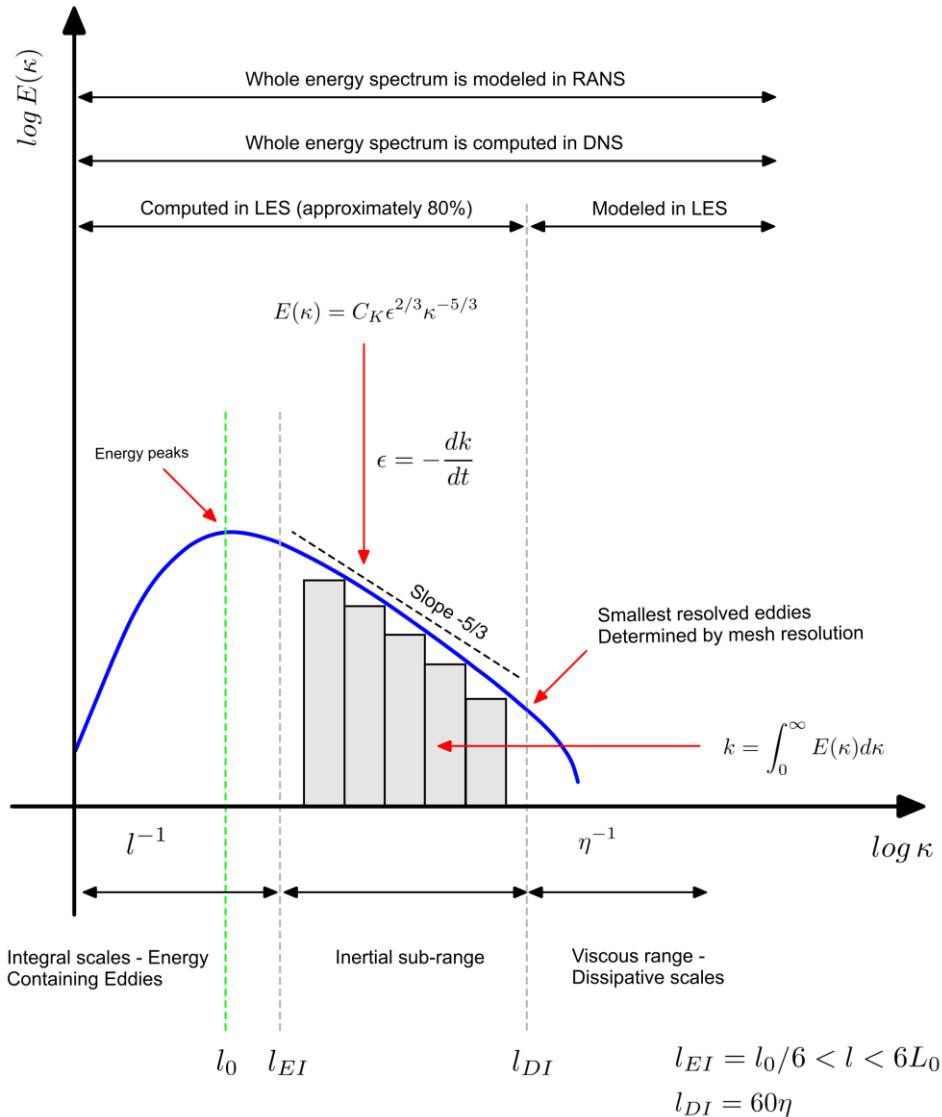
- In the same manner as  $E(\kappa)$  describe the wavenumber dependence of the turbulent kinetic energy, another distribution,  $D(\kappa)$ , describes the dependence of the dissipation rate on the wavenumber in the subrange C (refer to the figure).
- In this range, viscosity plays a central role.
- Again, by using dimensional analysis, we can derive an expression for the dissipation spectrum  $D(\kappa)$ ,

$$D(\kappa) = C_\epsilon \nu \epsilon^{2/3} \kappa^{1/3} \quad \text{with} \quad C_\epsilon \approx 2C_k$$

- In the figure, the variation of  $D(\kappa)$  in function of the wavenumber is illustrated (orange line).
- As it can be seen, the dissipation starts to rise in the inertial subrange (B in the figure), and then declines rapidly (in the subrange C in the figure).
- Similar observations to those of the  $E(\kappa)$  spectrum apply to the  $D(\kappa)$  spectrum, namely, slope, determination of the constant, experimental observations, and so on.

# Introduction to turbulence modeling

## 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,

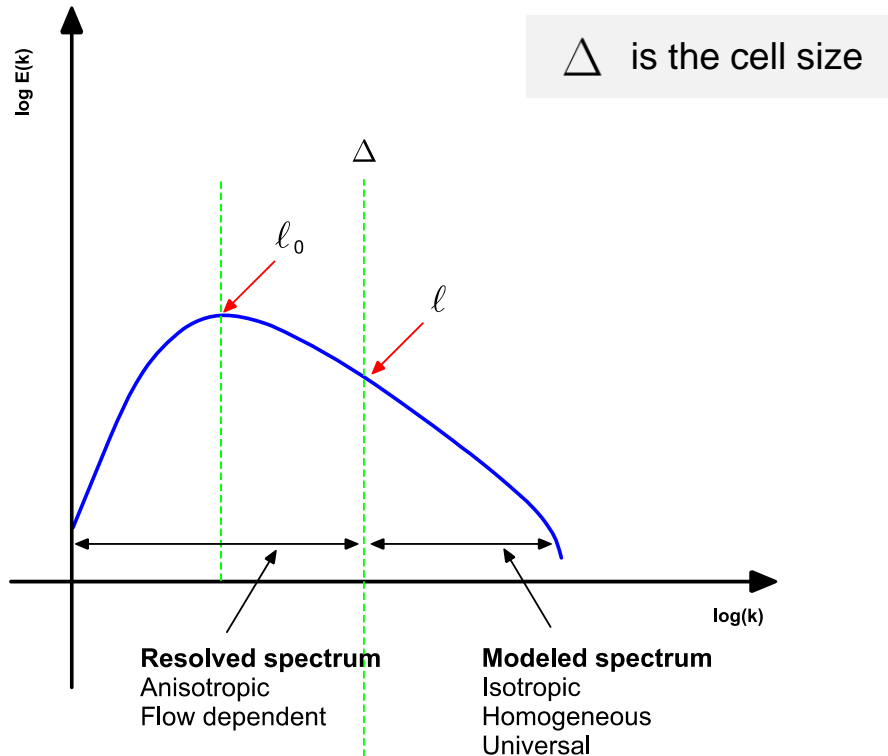
$$\kappa = \frac{2\pi}{l} = \frac{2\pi f}{U}$$

- The energy spectrum density or energy spectrum,  $E(\kappa)$  is related to the Fourier transform of  $k$ .
- The turbulent power spectrum represents the distribution of the turbulent kinetic energy  $k$  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.
- Remember, in CFD eddies cannot be resolved down to the molecular dissipation limit.

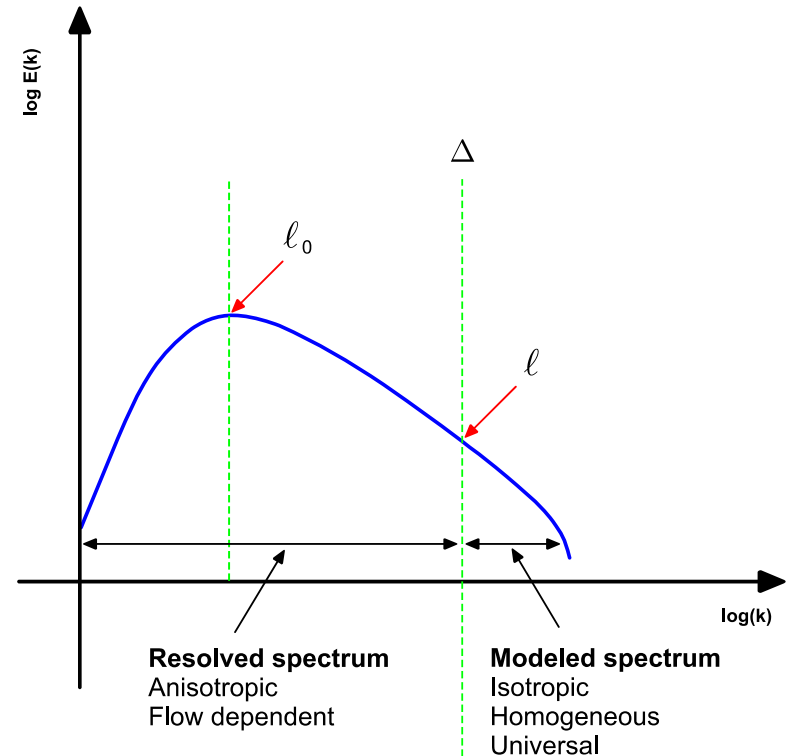


# Introduction to turbulence modeling

## Energy spectrum and energy cascade



Coarse mesh – More energy is modeled

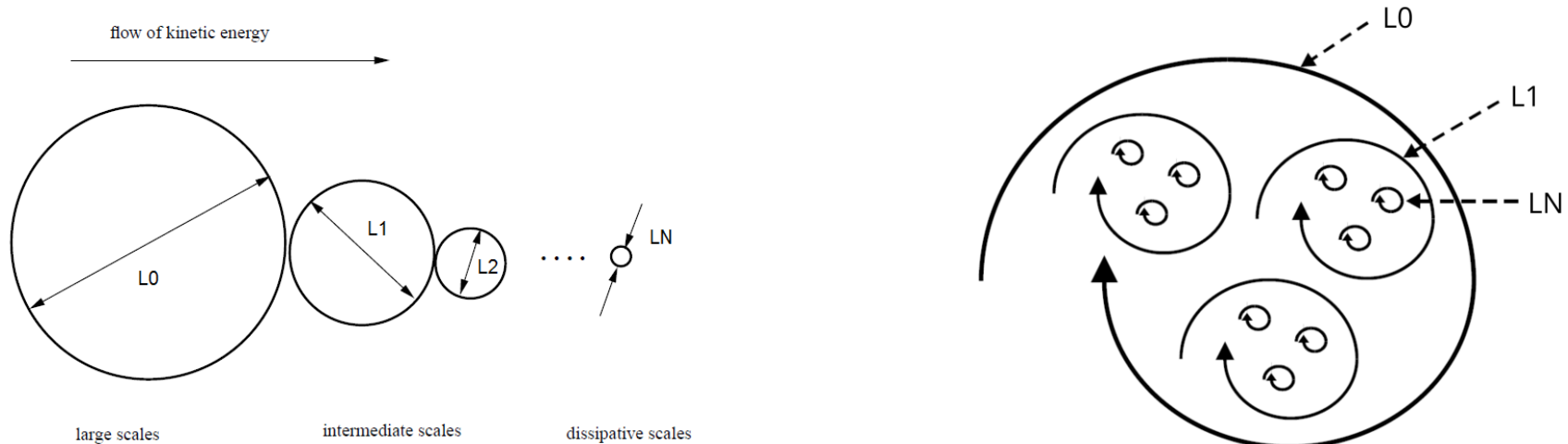


Fine mesh – Less energy is modeled

- The finer the mesh the less energy that is being modeled.
- Turbulent kinetic energy peaks at integral length scale  $l_0$ .
- In SRS simulations, this scale must be sufficiently resolved.

# Introduction to turbulence modeling

## Energy spectrum and energy cascade

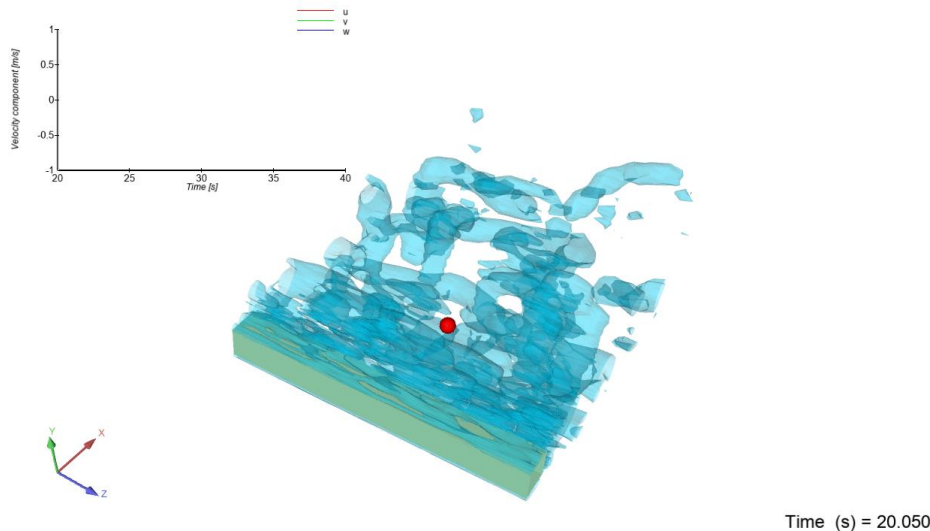
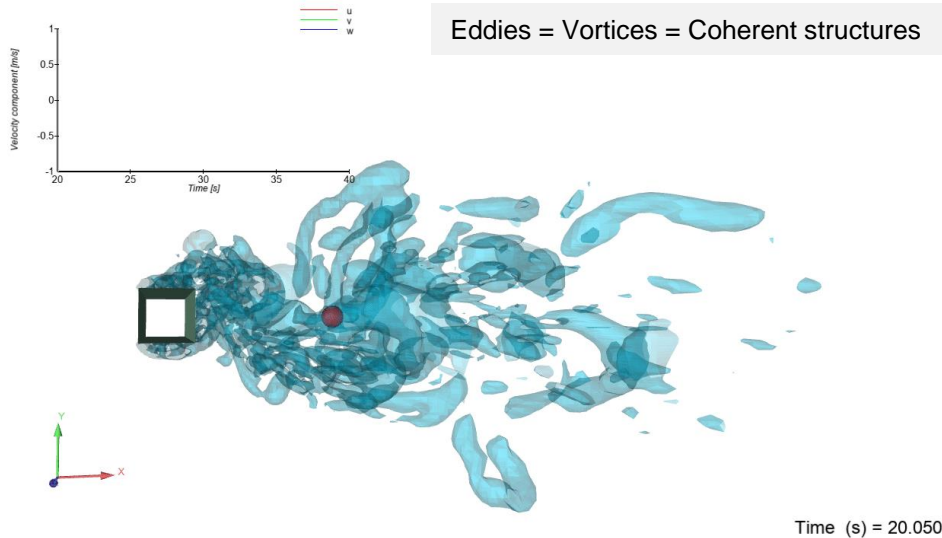


- The energy-containing eddies are denoted by  $L_0$ .
- $L_1$  and  $L_2$  denotes the size of the eddies in the inertial sub-range such that  $L_2 < L_1 < L_0$ .
- $L_N$  is the size of the dissipative eddies.
- The large, energy containing eddies transfer energy to smaller eddies via vortex stretching.
- 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.
- Large eddies ( $L_0$ ) are anisotropic; whereas small eddies ( $L_N$ ) are isotropic.
- The eddies in the inertial sub-range become more isotropic as energy is transferred from large eddies to small (dissipative) eddies.

# Introduction to turbulence modeling



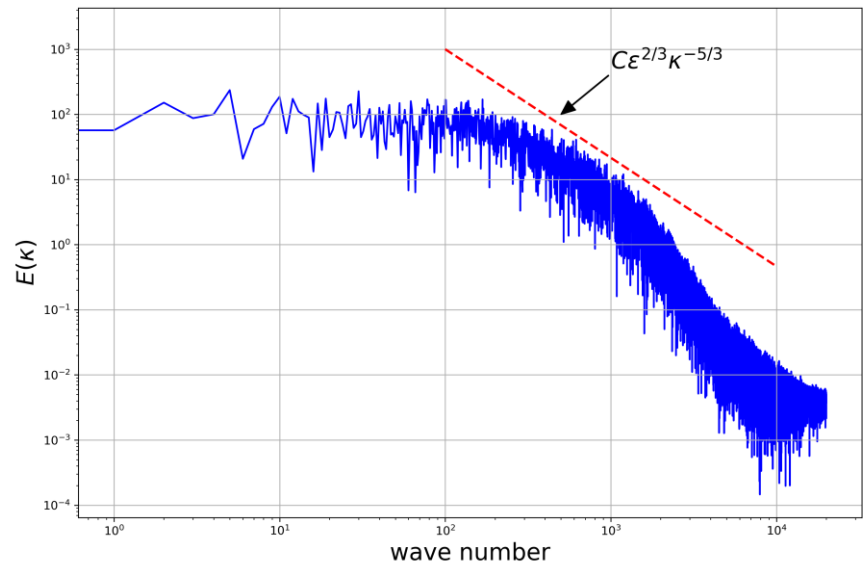
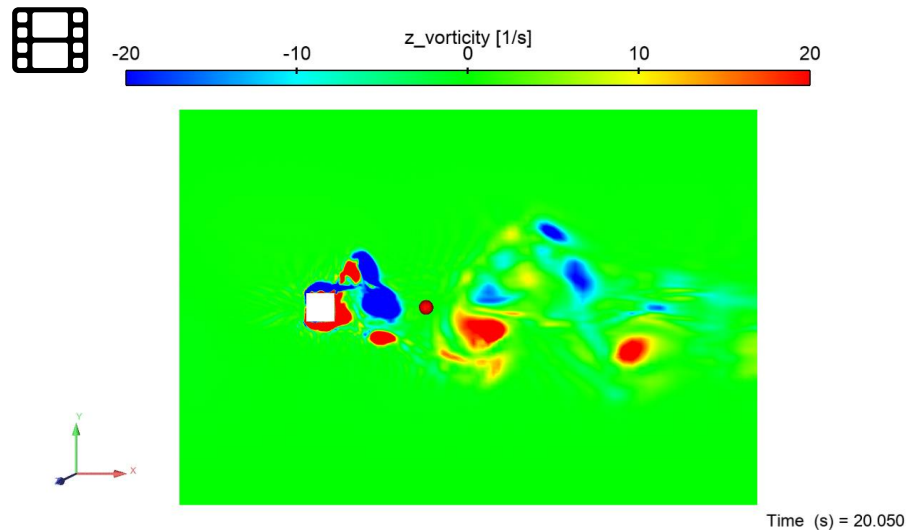
## Energy cascade in action



- To plot the energy spectrum, we need to sample the velocity field in a location behind the wake (the red sphere in this case).
- Then, by using signal processing (FFT), we can convert physical space into frequency space.
- We will address this type of post-processing later.
- **From these animations, we can notice that:**
- Large eddies:
  - They are energy-containing and extract their energy from the mean flow.
  - Their velocity is on the order of the mean flow.
  - Their size is on the order of the mean flow or the obstacle the flow over.
  - They are anisotropic and unstable. They break-up into smaller eddies.
  - Their frequency is low compared to the small eddies.
- Small eddies:
  - Smallest eddies convert kinetic energy into thermal energy via viscous dissipation.
  - Their behavior is more universal in nature.
  - Their frequency is high.

# Introduction to turbulence modeling

## Energy cascade in action



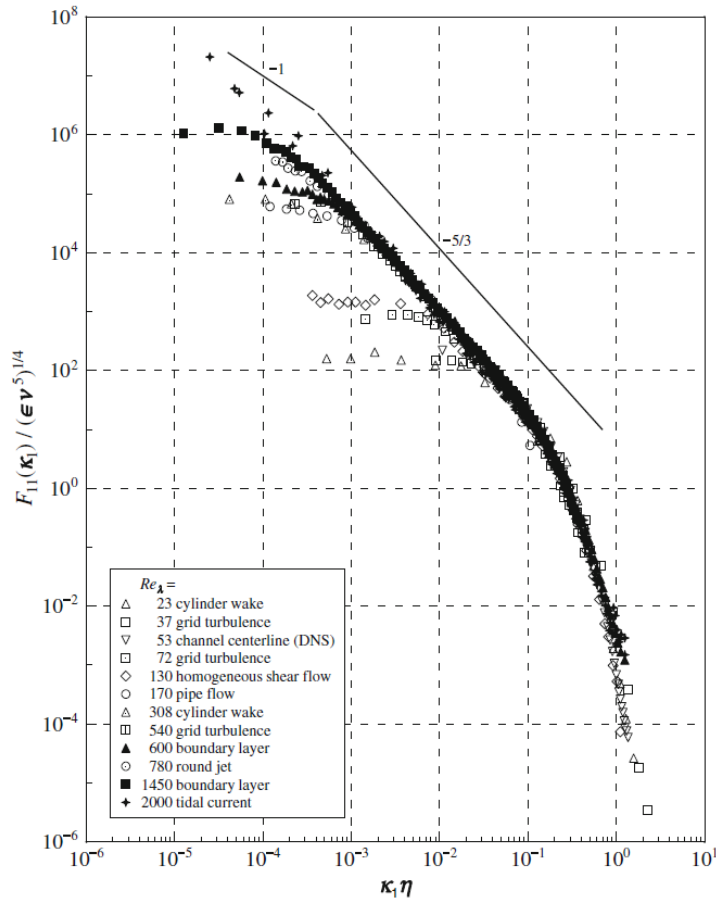
Vorticity contours – The velocity field is being sampled at the red sphere.

<http://www.wolfdynamics.com/training/turbulence/image21.gif>

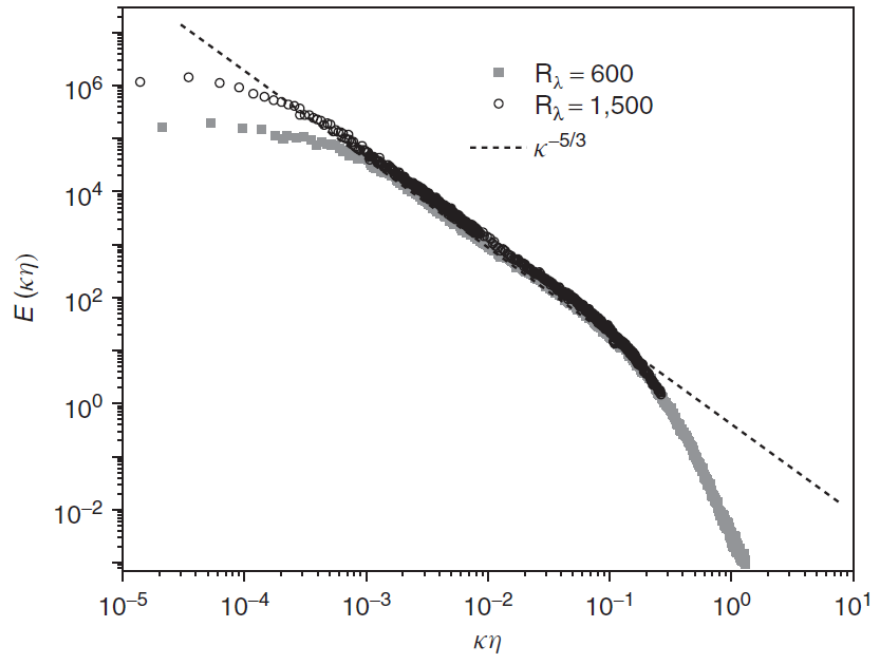
- To plot the energy spectrum, we need to sample the velocity fluctuations in a location behind the wake (the red sphere in this case, as shown in the left figure).
- Then, by using signal processing (FFT), we can convert physical space into frequency space.
- The plot of the energy spectrum is illustrated in the right figure.
- Remember, this plot is local.
- In the plot, the inertial sub-range is compared against the  $-5/3$  law (red line with a slope of  $-5/3$ ).

# Introduction to turbulence modeling

## Validity of the Kolmogorov theory (K41)



One-dimensional spectra scaled with respect to the microstructure from various turbulent flows. This demonstrates the universal character of the microstructure and illustrates the  $-5/3$  behavior of the inertial subrange [1].



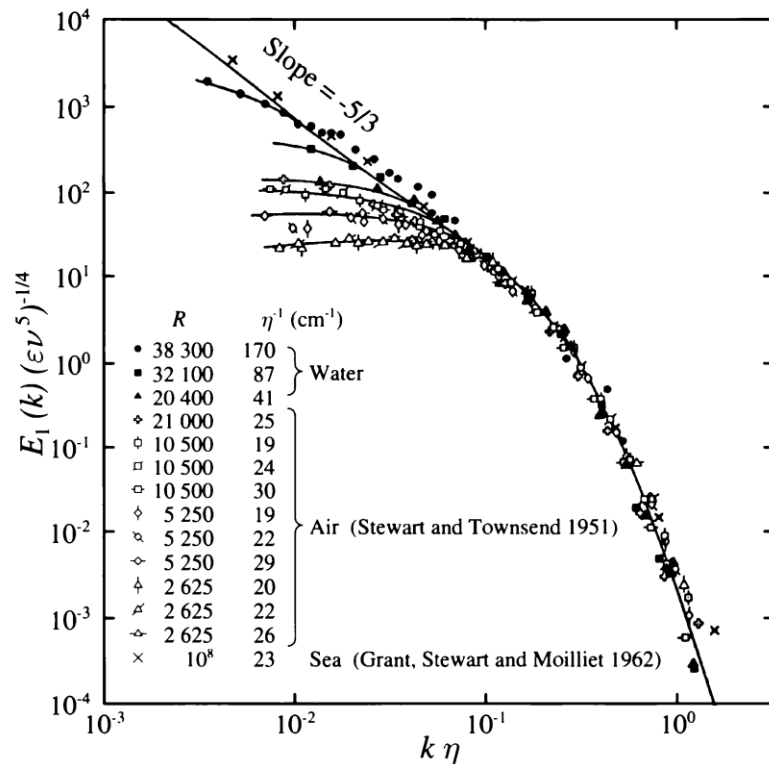
Experimental spectra measured in the boundary layer of the NASA Ames 80 x 100 foot wind tunnel [2].

[1] F. Nieuwstadt, B. Boersma, J. Westerweel. Turbulence. Introduction to Theory and Applications of Turbulent Flows. Springer, 2016.

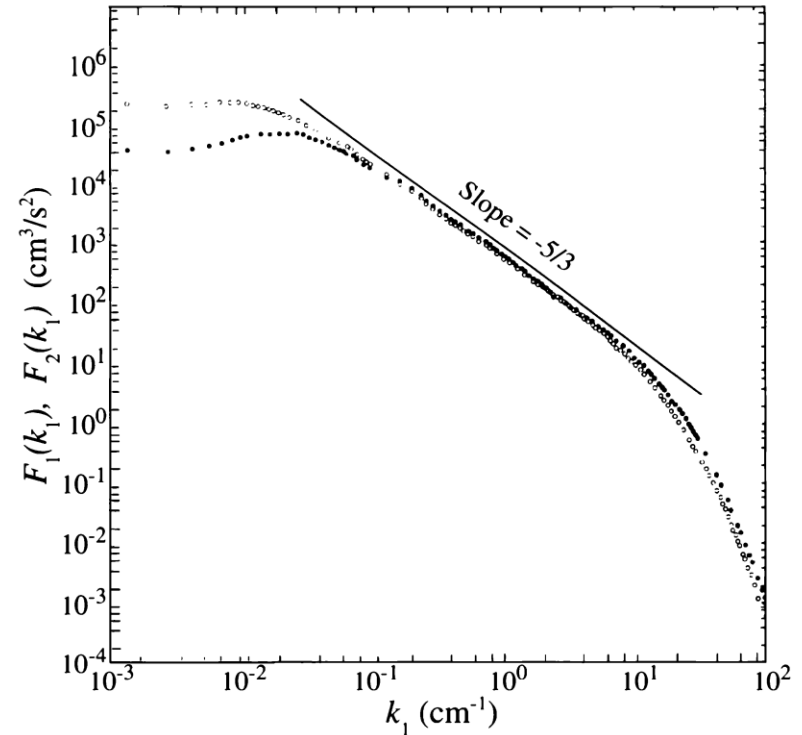
[2] P. Durbin, B.A. Pettersson-Reif. Statistical Theory and Modeling for Turbulent Flow. Wiley, 2011.

# Introduction to turbulence modeling

## Validity of the Kolmogorov theory (K41)



Normalized longitudinal velocity spectrum in the time domain according to different authors [1].



Log-log plot of the energy spectra of the streamwise components (white circles) and the lateral component (black circles) of the velocity fluctuations [2].

[1] F. Champagne. The fine-scale structure of the turbulent velocity field. J. Fluid Mech. 1978.

[2] C. Gibson, W. Schwarz. The universal equilibrium spectra of turbulent velocity and scalar fields. J. Fluid Mech. 1963.

# Introduction to turbulence modeling

## Energy spectrum and mesh resolution

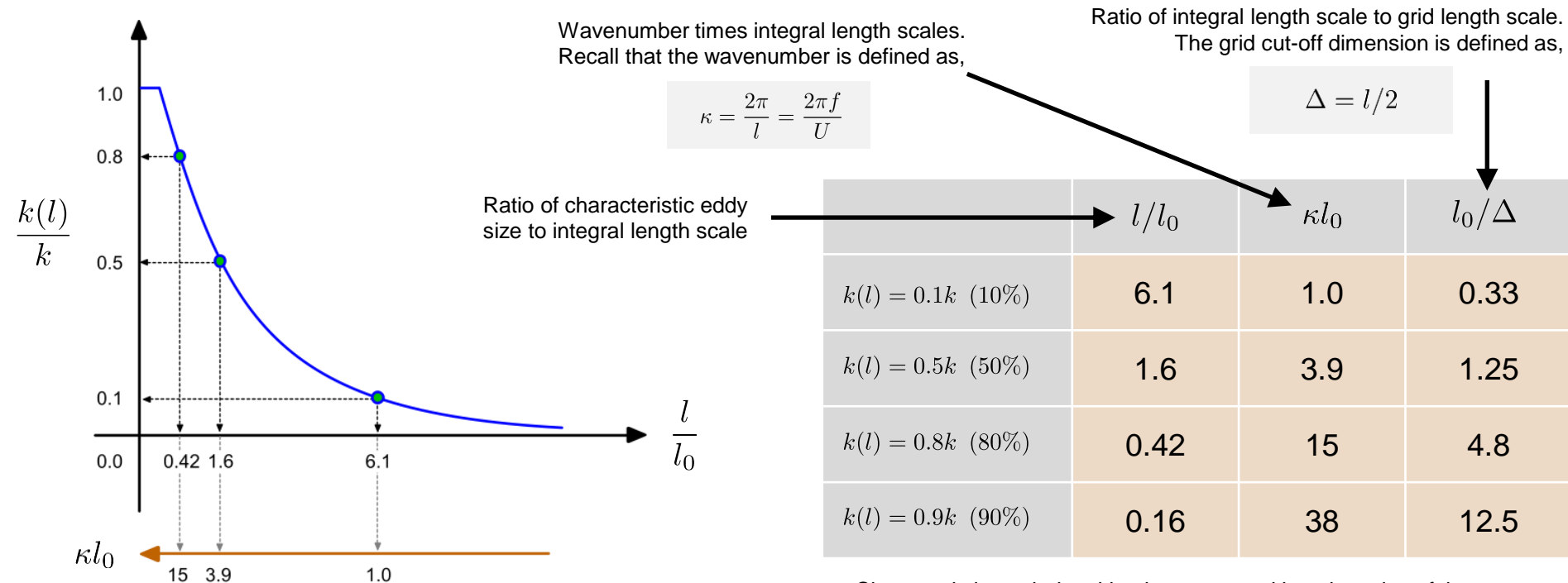
- DNS simulations are quite expensive, they require a lot of grid points/cells in order to resolve all the turbulent scales.
- Thus, in a DNS simulation the gridding requirements scales proportional to  $Re_T^{9/4}$  or approximately proportional to  $Re_T^3$  for a single time step.
- And every time 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.



# Introduction to turbulence modeling

## 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 scale-resolving simulation (SRS).
- Then, approximately 5 cells ( $l_0/\Delta$ ) will be needed across the integral length scale  $l_0$ .



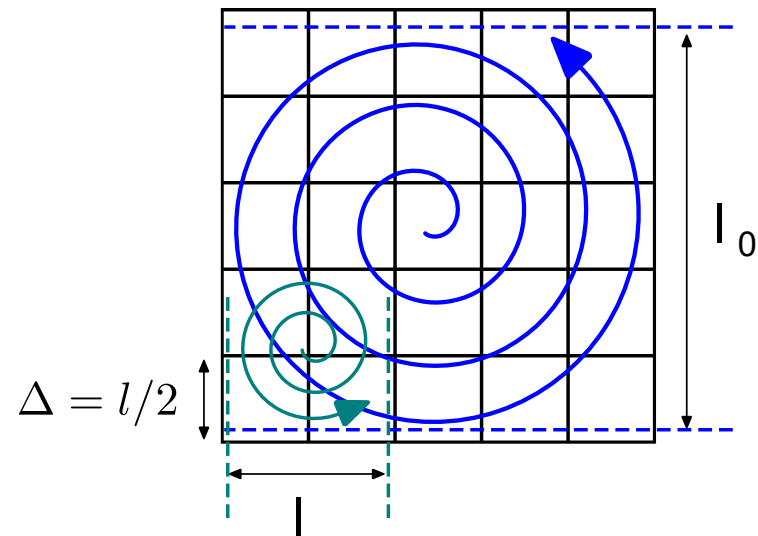
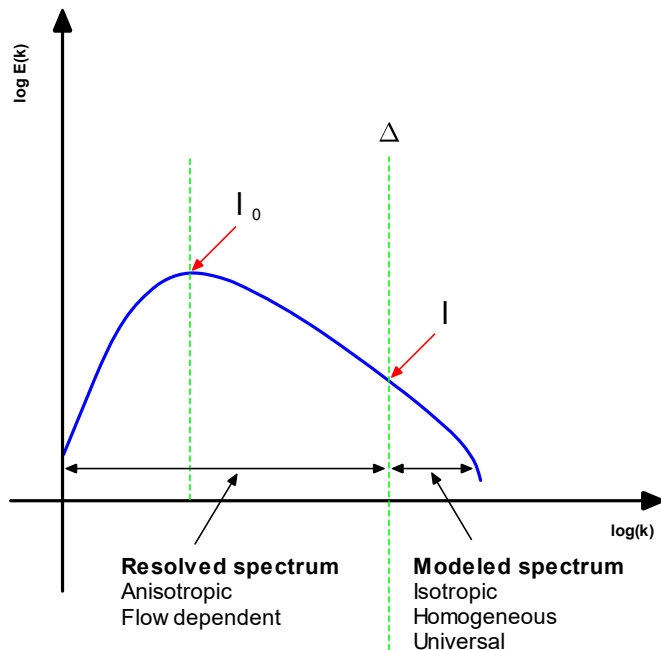
Cumulative turbulent kinetic energy against lengths scale of eddies. The figure has been adapted from Turbulent Flows by S. Pope

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

# Introduction to turbulence modeling

## Energy spectrum and mesh resolution

- The previous estimates using the cumulative turbulent kinetic energy, are very conservative regarding the minimum number of cells across the integral scales  $l_0$ .
- So, 5 to 6 cells are fine for RANS/URANS and VLES.
- However, for LES that resolves at least 80% of the turbulent spectrum, you will need to use at least 10 cells across the integral length scales  $l_0$ .
- To resolve an eddy with a length scale  $l$  (where  $l$  is the smallest scales that can be resolved with the grid or  $\Delta$ ), 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).



# Introduction to turbulence modeling

## Integral length scale and grid length scale

- The integral length scales  $l_0$  can be roughly estimated 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.

# Introduction to turbulence modeling

## Integral length scale and grid length scale

- The integral length scales  $l_0$  can be computed from a precursor RANS simulation.
- To compute the integral scales, you need to compute the turbulent kinetic energy  $k$ , dissipation rate  $\epsilon$ , and specific dissipation rate  $\omega$ .
- Therefore, you need to use a two-equation turbulence models:
  - $k - \epsilon$  family models.
  - $k - \omega$  family models.
- Depending on the model selected, you can compute  $l_0$  as follows,

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

- Again, if you check the dimensions, you will see that the base dimensions match.

Derived quantity	Symbol	Dimensional units	SI units
Energy dissipation rate per unit mass	$\epsilon$	$L^2T^{-3}$	$m^2/s^3$
Turbulent kinetic energy per unit mass	$k$	$L^2T^{-2}$	$m^2/s^2$
Specific dissipation rate	$\omega$	$T^{-1}$	$1/s$

# Introduction to turbulence modeling

## Integral length scale and grid length scale

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

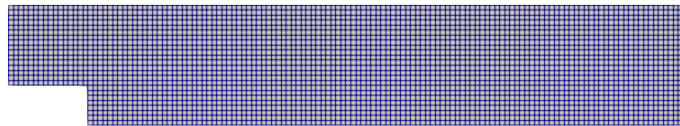
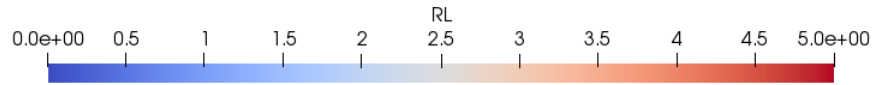
$$R_l = \frac{l_0}{\Delta} \quad \text{where } \Delta \text{ can be approximated as follows } \Delta \approx \sqrt[3]{\text{cell volume}}$$

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

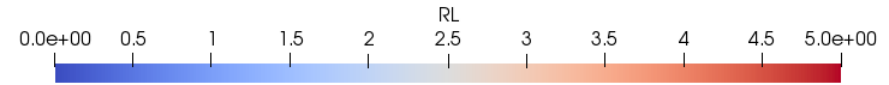
- The recommended value of  $R_l$  is about  $R_l > 5 - 10$ .
- Where 5 should be considered the lowest limit of resolution (for RANS/URANS and VLES) and 10 is the desirable lower limit (for LES/DES).
- Higher values can be used if computer power and time constraints permits.
- This is a very rough estimate,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

# Introduction to turbulence modeling

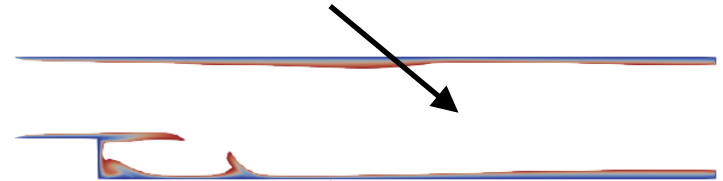
## Identification of integral length scale and grid length scale



Coarse mesh



The RL criterion is used to study the core of the flow



Close to the walls we use  $y^+$  as a criterion to determine the goodness of the mesh



Fine mesh

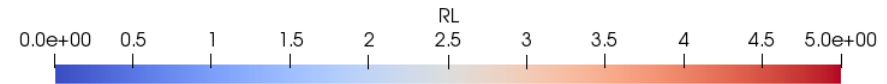
- We will show how to compute  $l_0$  and  $R_L$  during the guided tutorials.

# Introduction to turbulence modeling

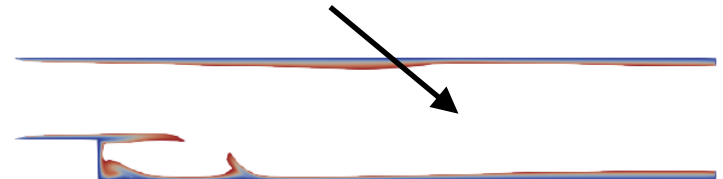
## Identification of integral length scale and grid length scale



**Coarse mesh**



The  $R_L$  criterion is used to study the core of the flow



Close to the walls we use  $y^+$  as a criterion to determine the goodness of the mesh



**Fine mesh**

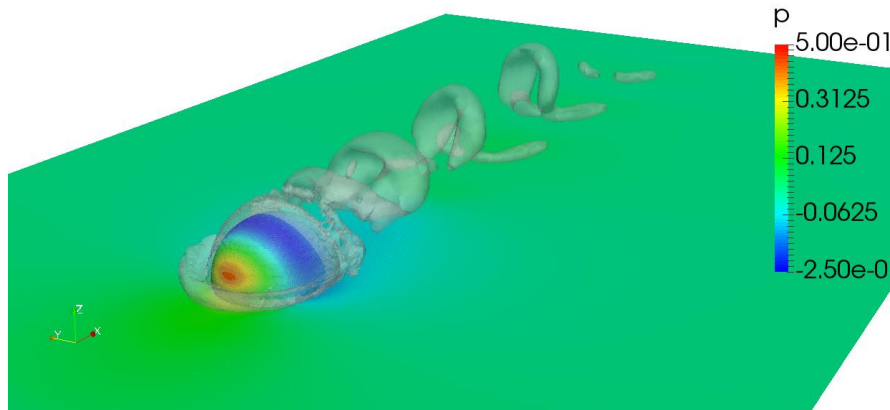
- To identify integral length scales  $l_0$  and grid length scales  $R_L$  you can plot contours of these quantities at different locations/planes in the domain.
- The lowest limit of  $R_L$  can be clipped so that the well resolved areas do not appear.
- In this case we are clipping (showing)  $0 < R_L < 5$ .
- Under-resolved areas (the areas shown), will need finer meshes or local mesh adaption.
- Near-wall regions always pose challenges.
  - In these areas is better to quantify the goodness of the mesh using quantity  $y^+$  (we will talk about it later).

# Introduction to turbulence modeling

## Turbulence modeling – Importance of mesh resolution



Time: 0



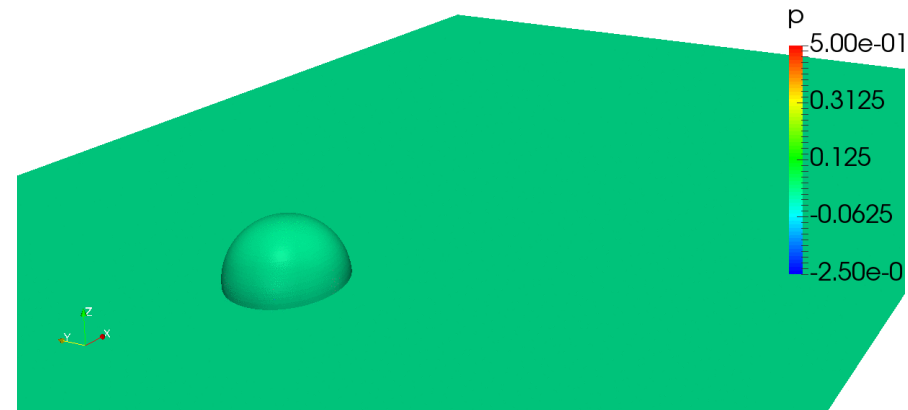
**Coarse mesh**

<http://www.wolfdynamics.com/training/turbulence/image7.gif>

- The vortices are dissipated due to numerical diffusion (low mesh resolution).
- Due to the low mesh resolution, it is quite difficult to onset the instability.



Time: 0



**Fine mesh**

<http://www.wolfdynamics.com/training/turbulence/image8.gif>

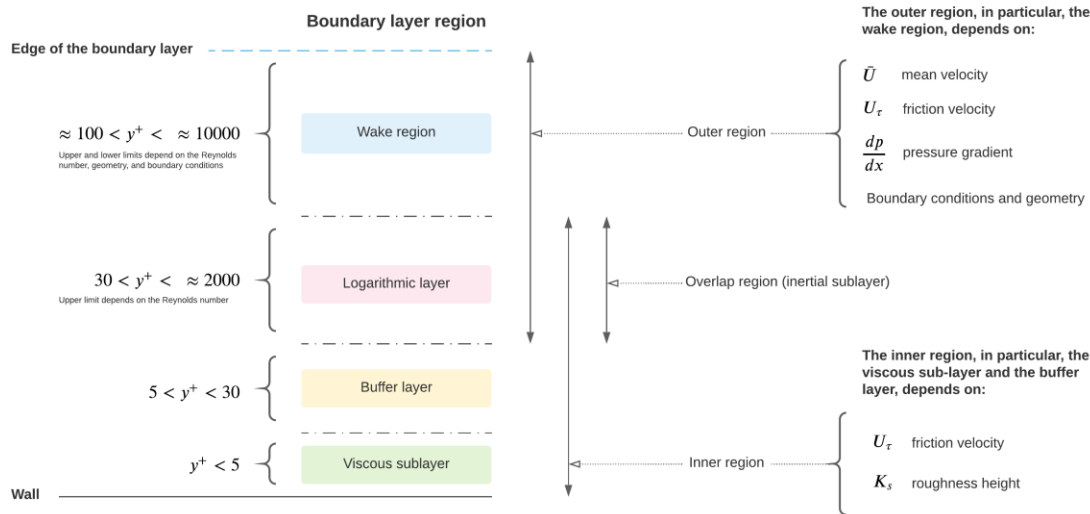
- The fine mesh captures the small spatial scales that the coarse mesh does not manage to resolve.

**Vortices visualized using Q criterion.**



# Introduction to turbulence modeling

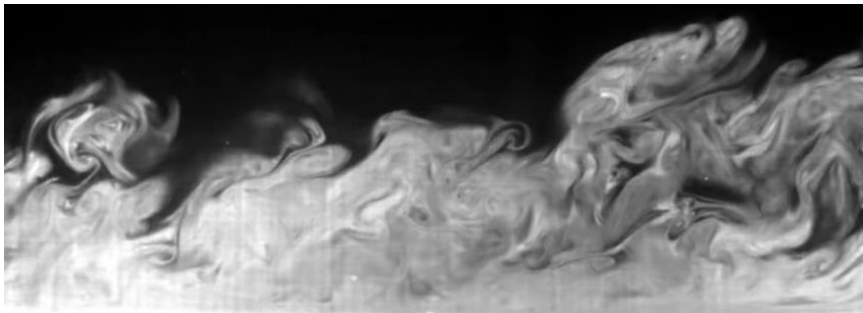
## Turbulence near the wall – The boundary layer and the Law of the wall



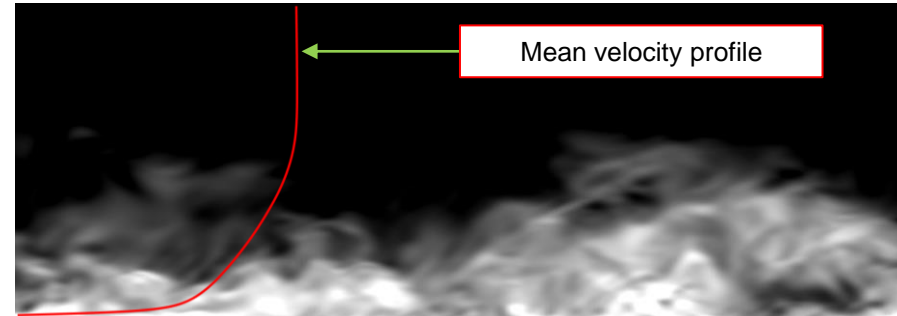
### Regions in the turbulent boundary layer.

- Adapted from references [1, 2].
- The figure is not to scale.
- The figure does not scale in reference to the images below.

### Turbulent boundary layer on a flat plate.



Experimental results [3].



DNS simulation. Instantaneous velocity field [4].

[1] F. Nieuwstadt, B. Boersma, J. Westerweel. Turbulence. Introduction to Theory and Applications of Turbulent Flows. Springer, 2016.

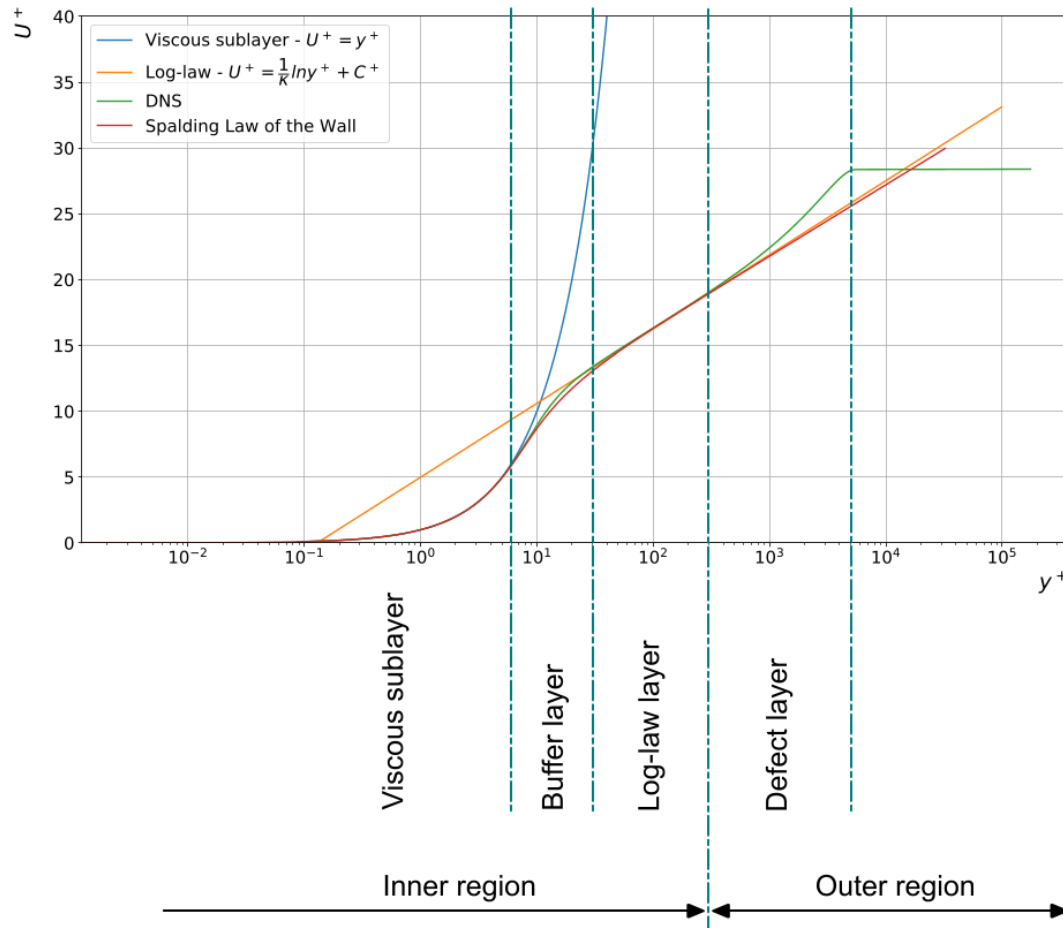
[2] S. Pope. Turbulent Flows, Cambridge University Press, 2000.

[3] Photo credit: <https://arxiv.org/abs/1210.3881>. Copyright on the images is held by the contributors. Apart from Fair Use, permission must be sought for any other purpose.

[4] J. Lee, T. Zaki. Transitional boundary layer dataset. [http://turbulence.pha.jhu.edu/Transition\\_bl.aspx](http://turbulence.pha.jhu.edu/Transition_bl.aspx)

# Introduction to turbulence modeling

## Turbulence near the wall – Law of the wall



- The law of the wall, is one of the cornerstones of fluid dynamics and turbulence modeling.
- It is based on the early works of Prandtl [1], Von Karman [2], Nikuradze [3], and Millikan [4].
- Many other authors have derived/confirmed the law of the wall using experimental or numerical measurements.
- By using dimensional analysis and taking the right assumptions, the following expression can be derived,

$$\frac{U}{u_\tau} = f\left(\frac{yu_\tau}{\nu}\right)$$

- Or by using non-dimensional groups ( $u^+$  and  $y^+$ ),

$$u^+ = f(y^+)$$

- The law of the wall basically describes the mean velocity distribution close to the wall, in the inner region of the boundary layer.
- Where viscous effects dominates.

[1] L. Prandtl. "Report on Investigation of Developed Turbulence". Technical Memorandum 1231, NACA. 1949. Translation of the 1925 paper.

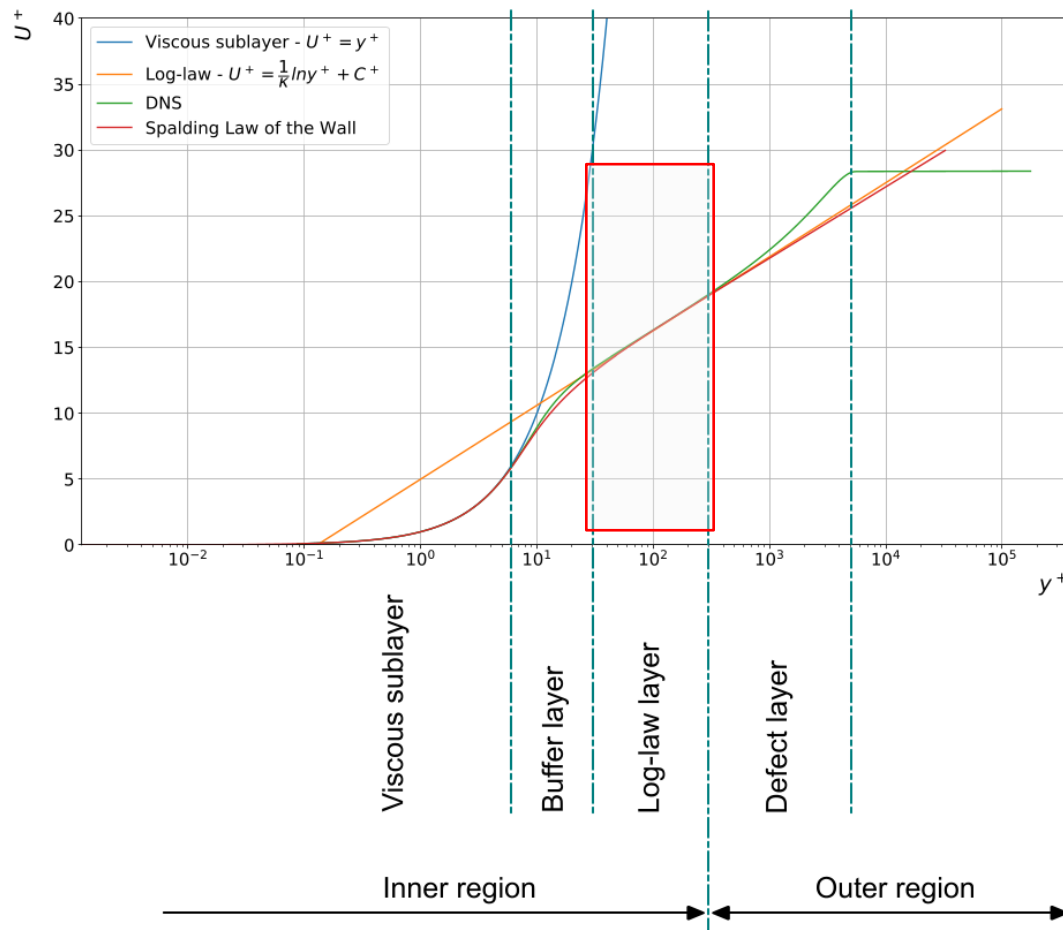
[2] T. von Karman. "Mechanical Similitude and Turbulence". Technical Memorandum 611, NACA. 1931. Translation of the 1930 paper.

[3] J. Nikuradse. "Law of Flow in Rough Pipes". Technical Memorandum 1292, NACA. 1950. Translation of the 1933 paper.

[4] C. Millikan. "A critical discussion of turbulent flows in channels and circular tubes". Proc. Fifth Int. Congress for Applied Mechanics, Harvard and MIT, 1938.

# Introduction to turbulence modeling

## Turbulence near the wall – Logarithmic law or log-law



- The logarithmic law, refers to the region of the inner-region of the boundary layer that can be described using a simple analytic function in the form of a logarithmic equation.
- This is one of the most famous empirically determined relationships in turbulent flows near solid boundaries.
- Measurements show that, for both internal and external flows, the streamwise velocity in the flow near the wall varies logarithmically with distance from the surface.
- The log-law, is stated as follows,

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

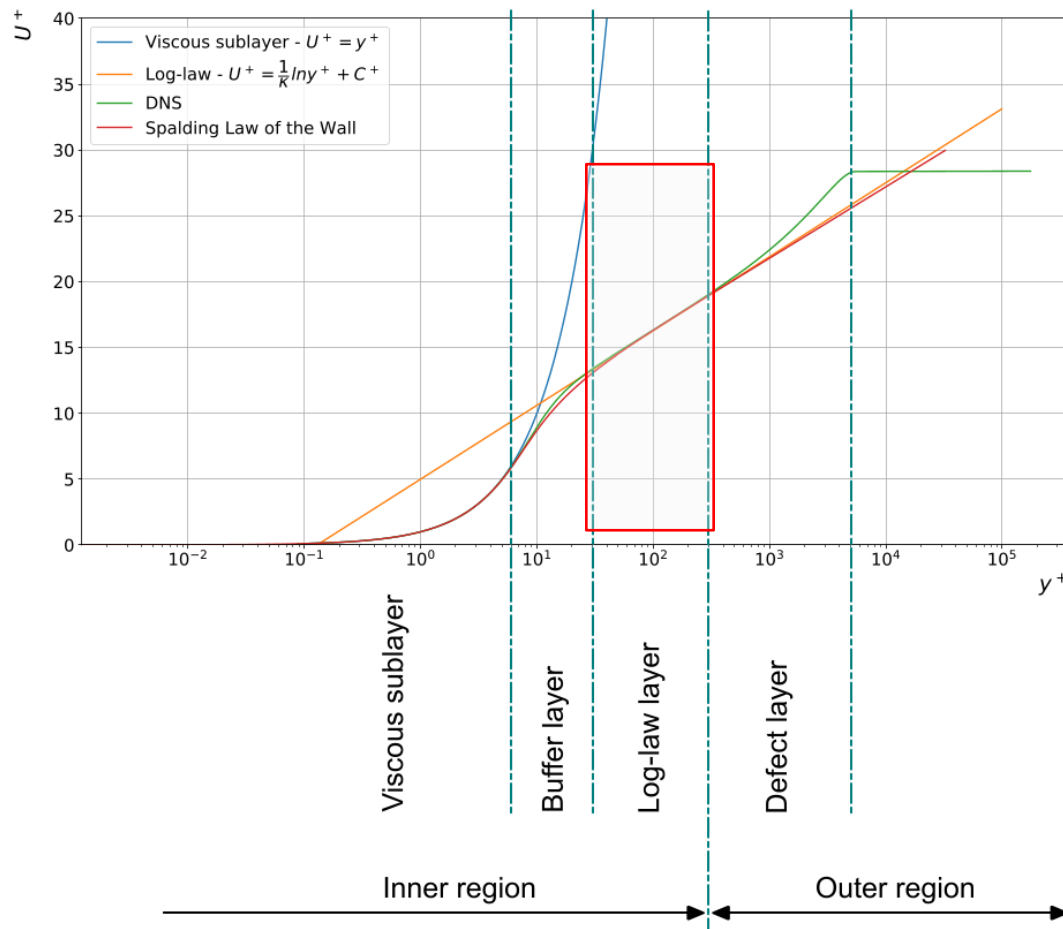
- Where the most common values for the constants appearing in the previous function are,

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

- Reported values for the constant (or coefficient)  $C^+$  can go anywhere from 4.5 to 5.5.
- Reported values of the Karman constant  $\kappa$  can go anywhere from 0.36 to 0.42.

# Introduction to turbulence modeling

## Turbulence near the wall – Logarithmic law or log-law



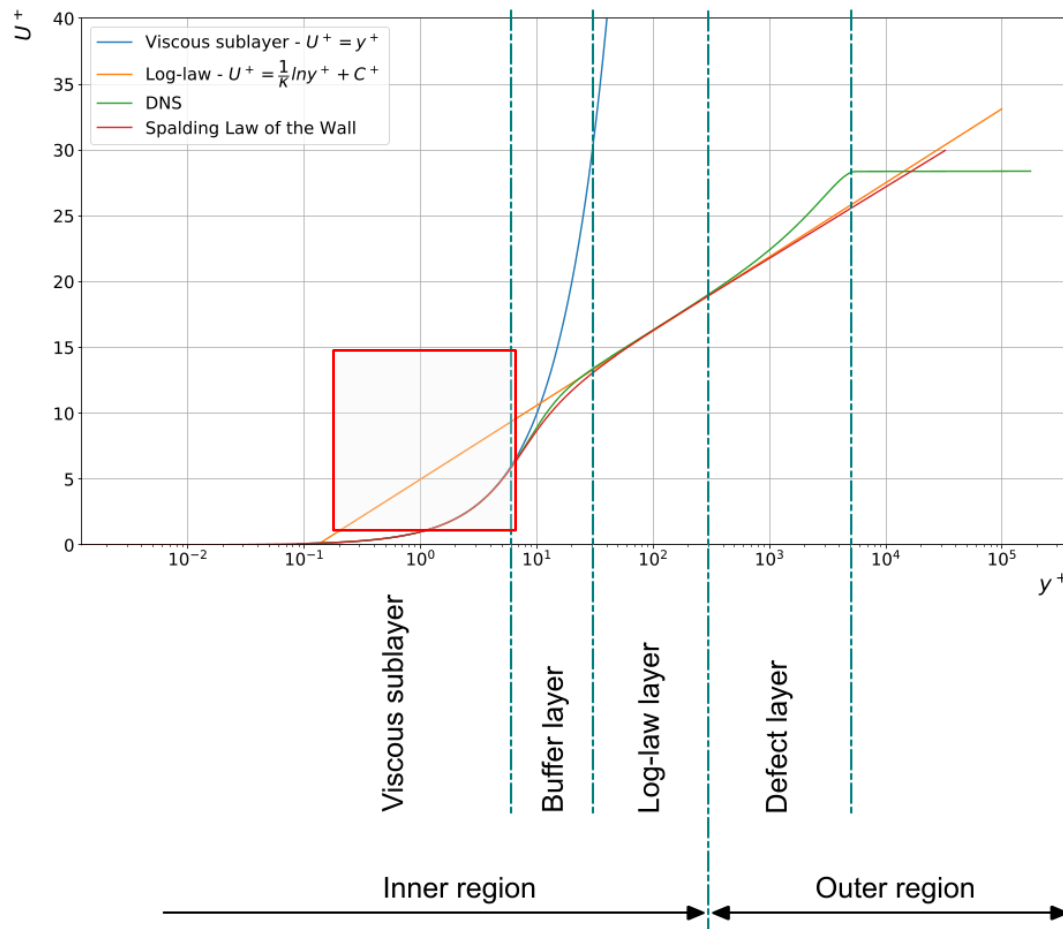
- It is interesting to mention that after the log-law was derived, it took some time to determine the constant values using experimental measurements.
- The logarithmic law or log-law is valid for values of  $y^+$  ranging from,

$$30 < y^+ < 300$$

- The previous range of  $y^+$  values is the most found in literature.
- In reality, this range depends on the Reynolds number.
- The upper limit can be as high as 10000.
- For practical purposes in CFD, let us say that the lowest limit is 30, and the upper one is 600.
- The log-law can be derived using dimensional analysis [1, 2, 3].
- In the literature and based on the assumptions taken, you will find different roads to arrive to the log-law, it is up to you to choose one.

# Introduction to turbulence modeling

## Turbulence near the wall – The viscous sublayer



- The viscous sublayer, refers to the region of the inner-region of the boundary layer, very close to the wall and where the flow is laminar.
- In this region the flow mean velocity can be described using a simple analytic function.
- The viscous sublayer law, is stated as follows,

$$u^+ = y^+$$

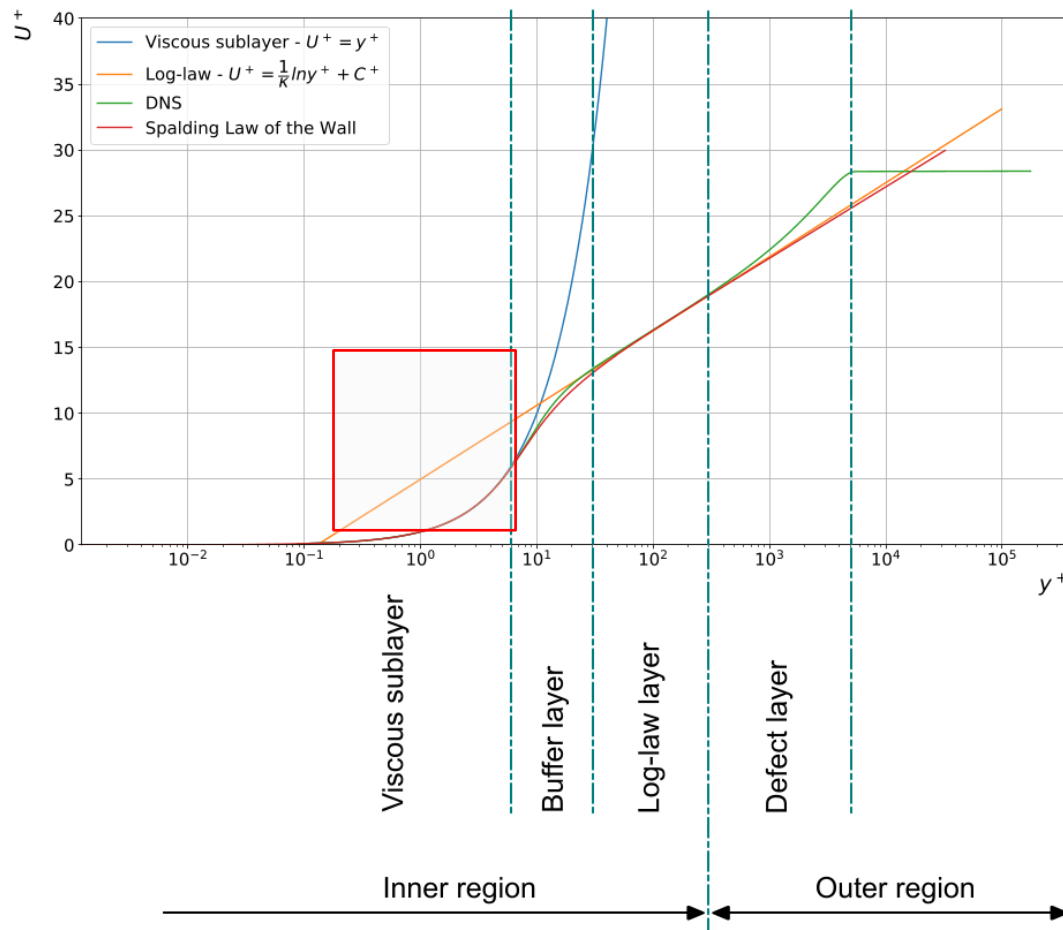
- Remember, this equation is only valid in the viscous sublayer, where the flow is laminar and viscous effects are very strong, therefore,

$$\tau_{wall} = \mu \frac{\partial u}{\partial y}$$

- According to this law, the behavior of the mean velocity is linear in this region.
- Notice that the x-axis is in logarithmic scale.
- Again, the viscous sublayer law can be derived using dimensional analysis [1, 2, 3].

# Introduction to turbulence modeling

## Turbulence near the wall – The viscous sublayer



- The behavior of the viscous sublayer has been confirmed using experimental and numerical measurements.
- The viscous sublayer law is valid for values of  $y^+$  ranging from,

$$y^+ < 5$$

- In the literature, you will find different values of the upper limit.
- But maybe this is the most agreed value of  $y^+$  for the viscous sublayer ( $y^+ < 5$ ).
- Reported values of  $y^+$  in the viscous sublayer can be as high as 10, and as low as 3.
- The range of validity of the viscous sublayer law has been questioned by many authors [1,2,3,4].
- For practical purposes in CFD, and to avoid excessive computational load, let us say that the upper limit is 6.
- Yes, one unit can make a huge difference in CFD.
- In particular regarding to the mesh size.

[1] A. Cenedese, G. Romano, R. Antonia. A comment on the linear law of the wall for fully developed turbulent channel flow. Experiments in Fluids 25, 1998.

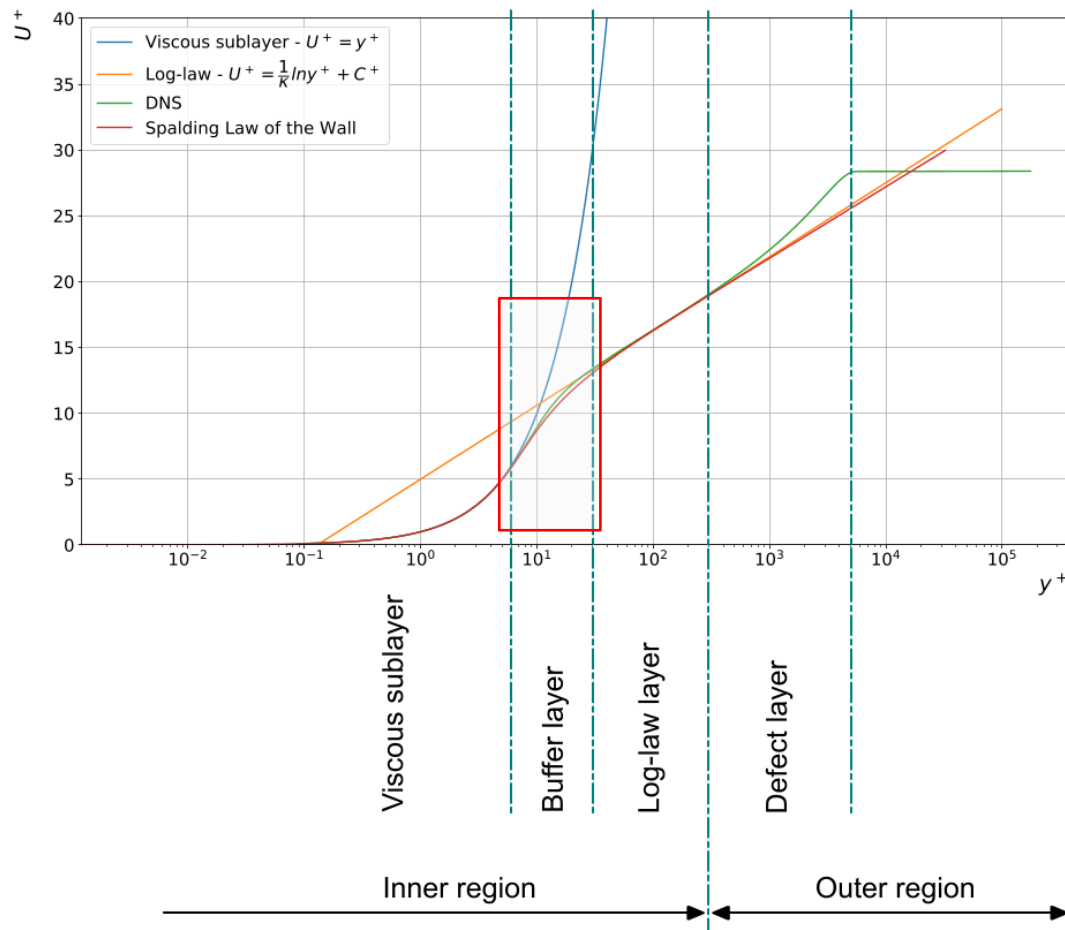
[2] A. Townsend. The Structure of Turbulent Shear Flow. Cambridge, Cambridge University Press, 1956.

[3] J. Sternberg. A theory for the viscous sublayer of a turbulent flow. Journal of Fluid Mechanics, 13(2), 1962.

[4] D. Spalding. A Single Formula for the Law of the Wall. J. Appl. Mech. Sep 1961.

# Introduction to turbulence modeling

## Turbulence near the wall – The buffer layer



- In the buffer layer, where the flow transitions from laminar to turbulent, there is nothing written in regarding.
- This region is very energetic with a behavior that is problem dependent.
- The buffer layer is enclosed in the following range of  $y^+$  values,

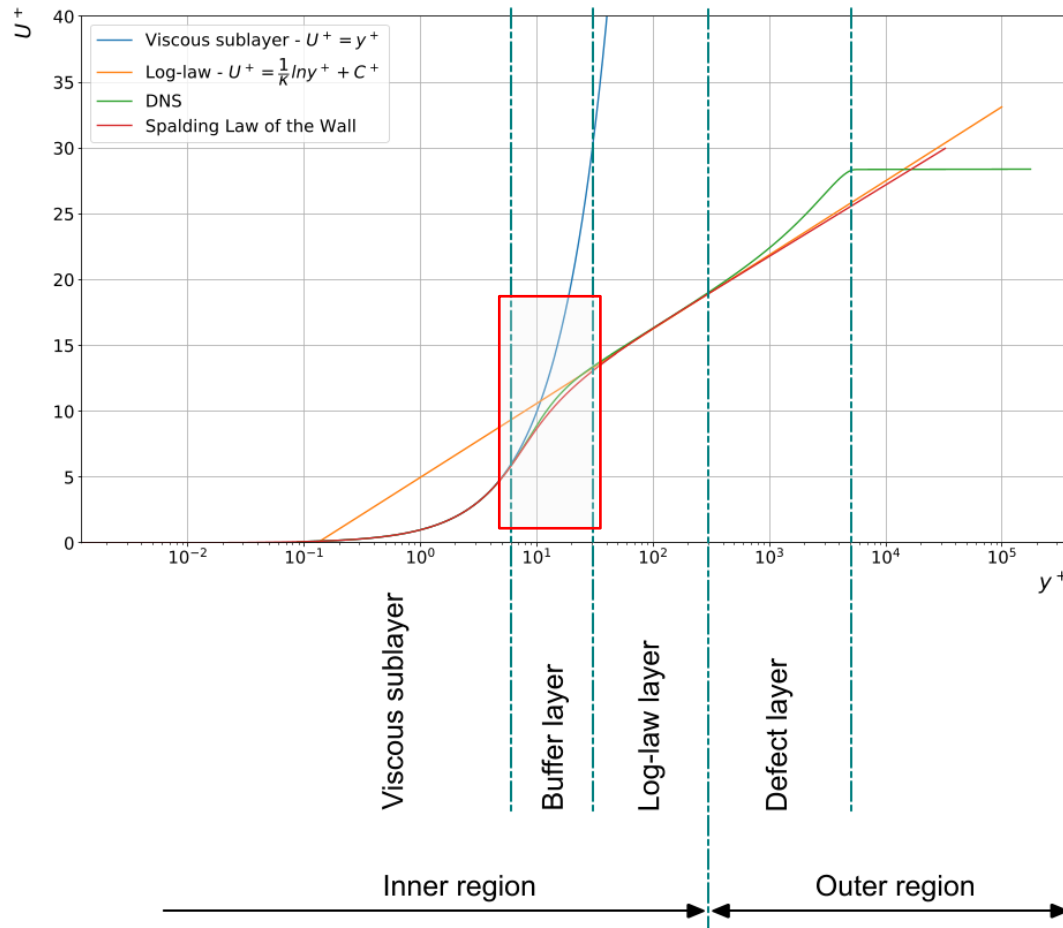
$$5 < y^+ < 30$$

- As usual, in the literature you will find different ranges of  $y^+$  values.
- But maybe this is the most common range.
- It is worth mentioning that there are a few correlations that can be used to approximate the buffer layer.
- However, they are not very accurate as uncertainty is high in the buffer layer.



# Introduction to turbulence modeling

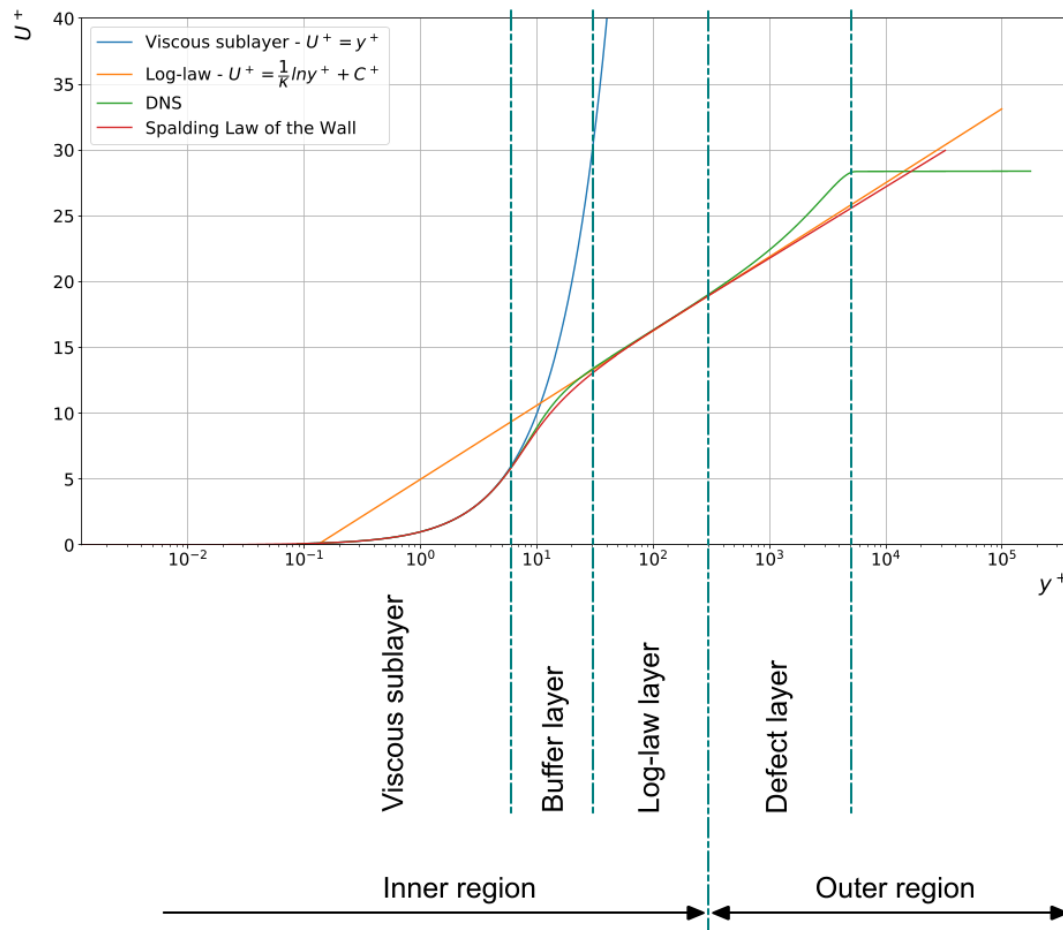
## Turbulence near the wall – The buffer layer



- In CFD, we try to avoid as much as possible this region because there is no single function that can describe accurately this region.
- This means that we try to place the first cell center or cell node (depending on the solution method used) outside this region.
- If we solve the viscous sublayer ( $y^+ < 5$ ), inevitably this region will be solved.
- What is important is not to place the first cell next to the wall in this region.
- If you are conducting simulations at relative low Reynolds number but still turbulent, the log-law layer and the buffer layer is very narrow.
- In such cases, it is recommended to always solve the viscous sublayer.

# Introduction to turbulence modeling

## Turbulence near the wall – Definition of $y^+$ and $u^+$



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

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

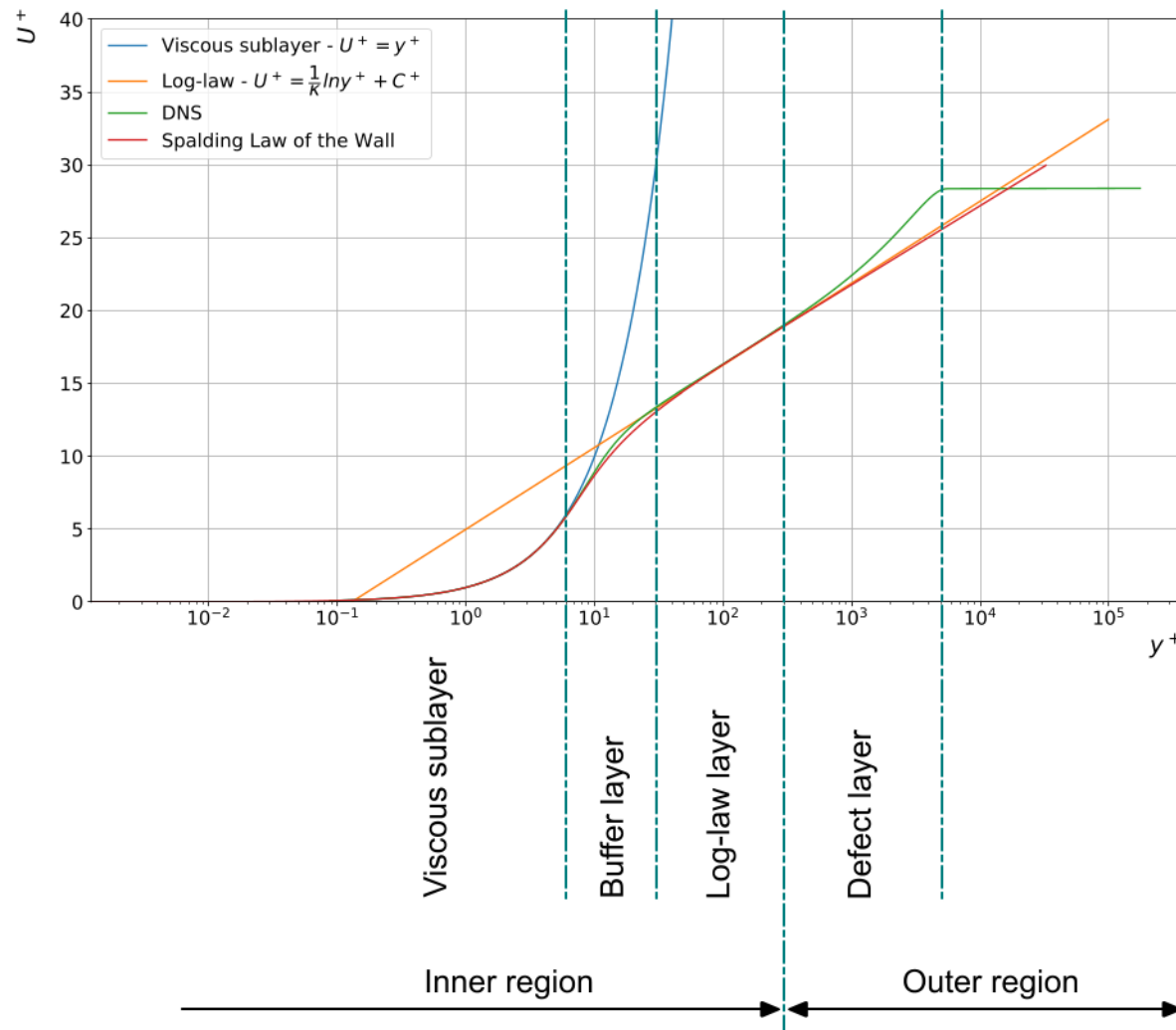
$$u^+ = \frac{U}{U_\tau}$$

Where  $y$  is the distance normal to the wall,  $U_\tau$  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.

# Introduction to turbulence modeling

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



Viscous sublayer

$$y^+ < 5$$

$$u^+ = y^+$$

Buffer layer

$$5 < y^+ < 30$$

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

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

Log-law layer

$$30 < y^+ < 300$$

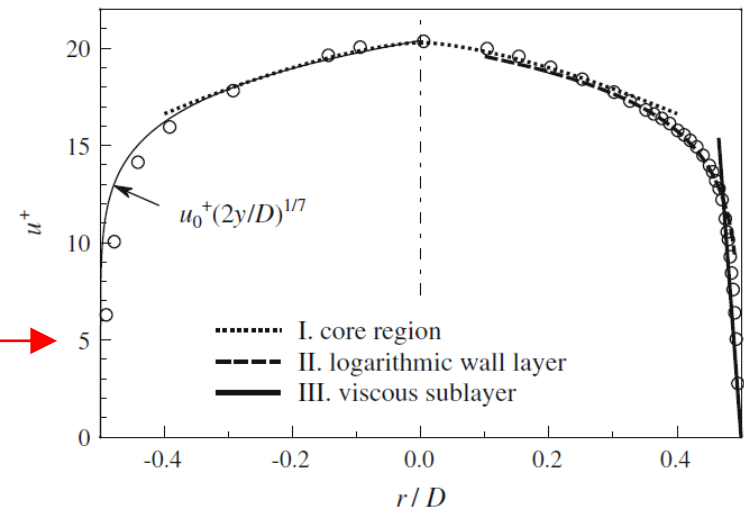
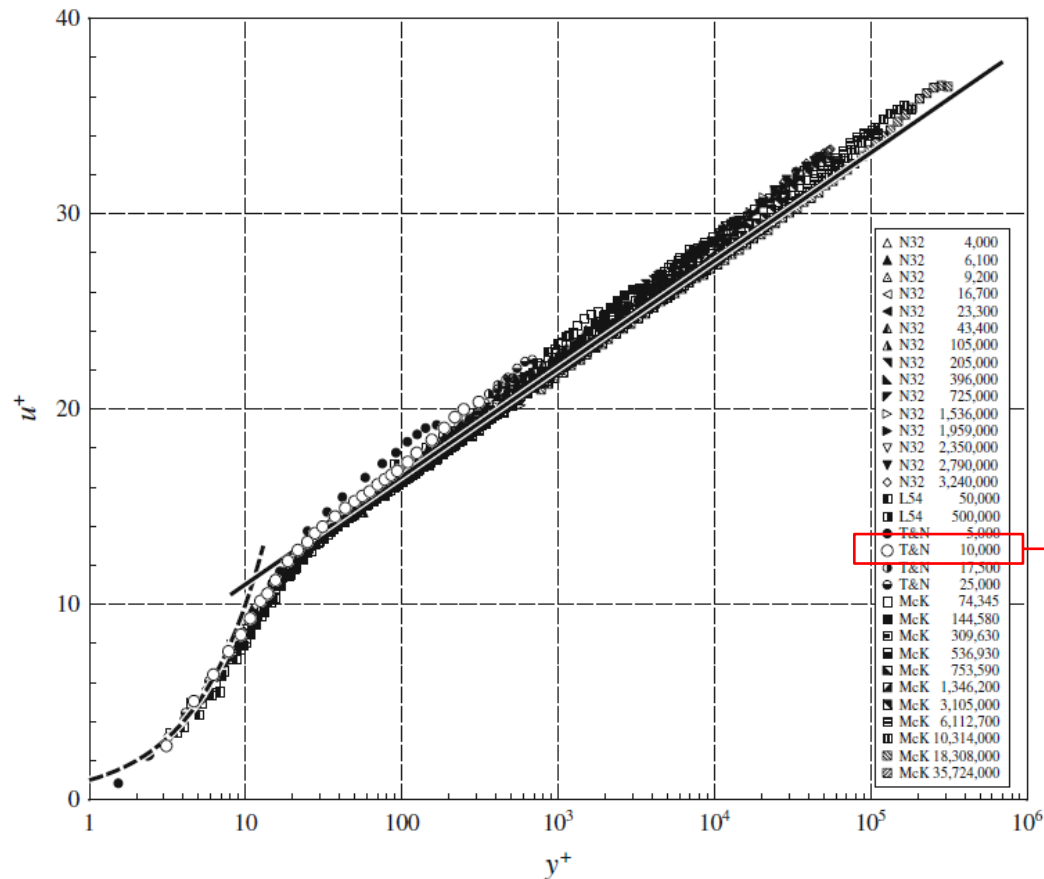
$$u^+ = \frac{1}{\kappa} \ln y^+ + C^+$$

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

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

# Introduction to turbulence modeling

## Turbulence near the wall – Experimental data



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.

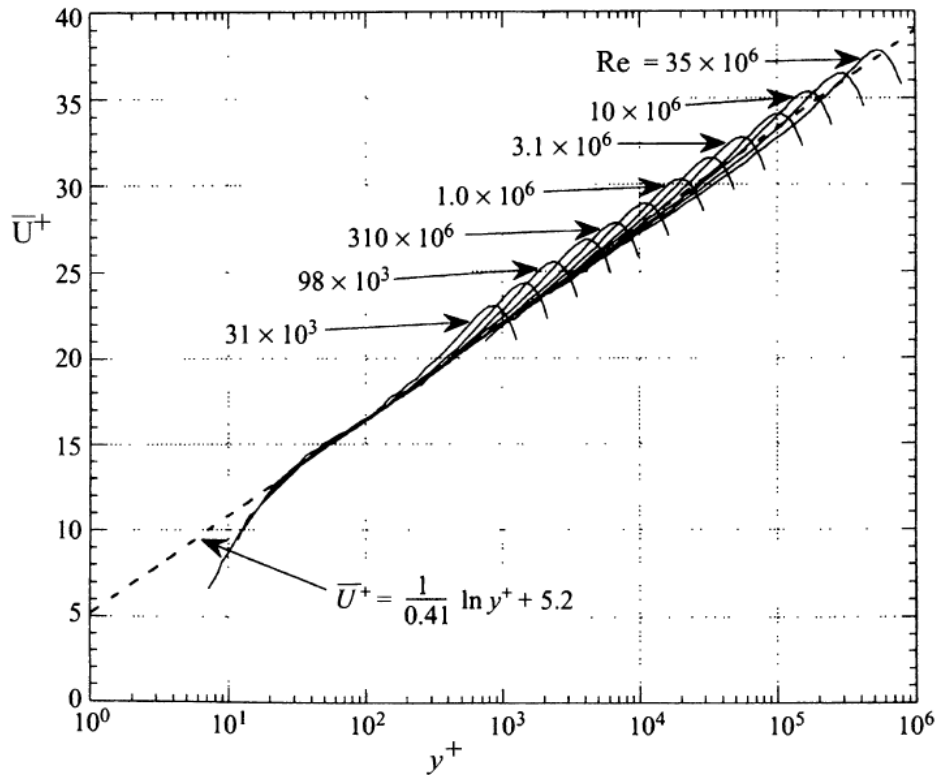
Dimensionless mean velocity profile  $u^+$  as a function of the dimensionless wall distance  $y^+$  for turbulent pipe flow with Reynolds numbers between 4000 and 3600000.

### References:

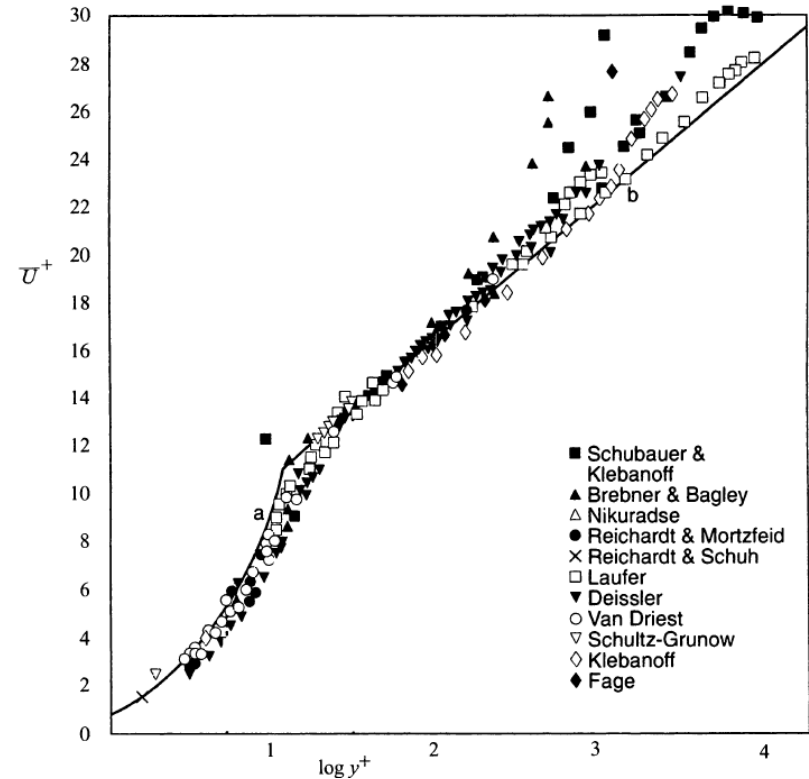
F. Nieuwstadt, B. Boersma, J. Westerweel. Turbulence. Introduction to Theory and Applications of Turbulent Flows. Springer, 2016.

# Introduction to turbulence modeling

## Turbulence near the wall – Experimental data



$U^+$  vs.  $y^+$  plot for high-Reynolds number experiments [1].



Log-law versus experiment at several Reynolds numbers [2].

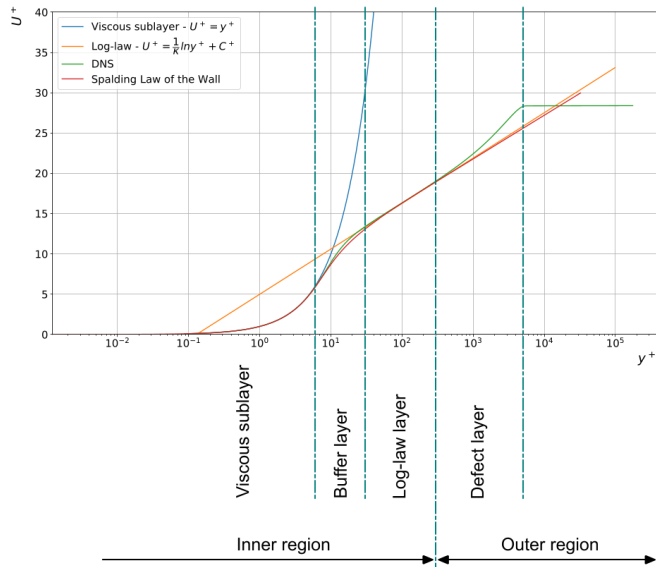
- The log-law is one of the most famous empirically determined relationships in turbulent flows near solid boundaries.

[1] M. Zagarola, A. Smits. Mean flow scaling in turbulent pipe flow. J. Fluid Mech. 373. 1998.

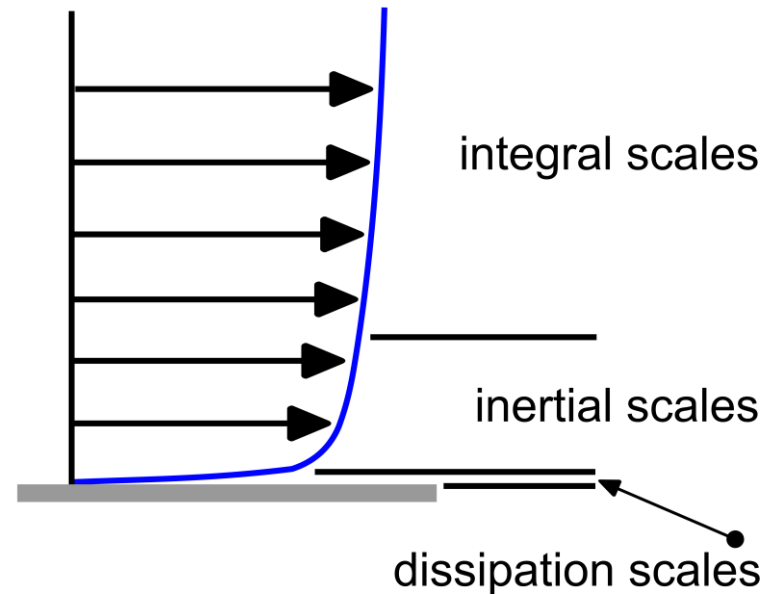
[2] A. S. Monin and A. M. Yaglom. Statistical Fluid Mechanics: The Mechanics of Turbulence, volume 1. M. I. T. Press, 1971.

# Introduction to turbulence modeling

## Non-dimensional profile against physical velocity profile



Non-dimensional profile



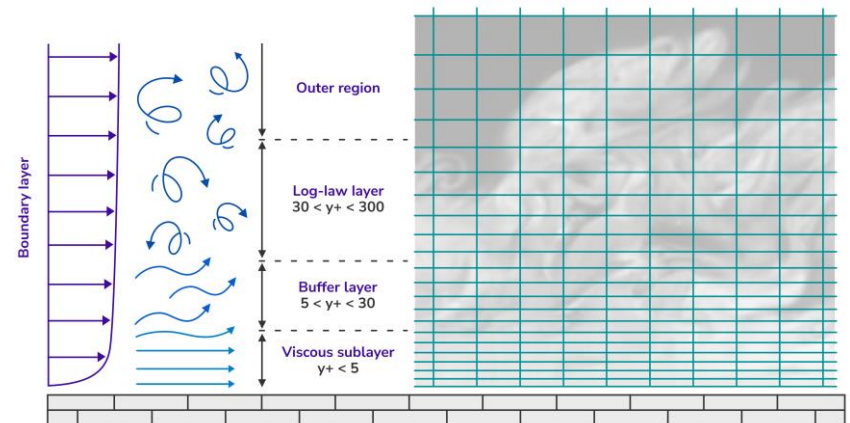
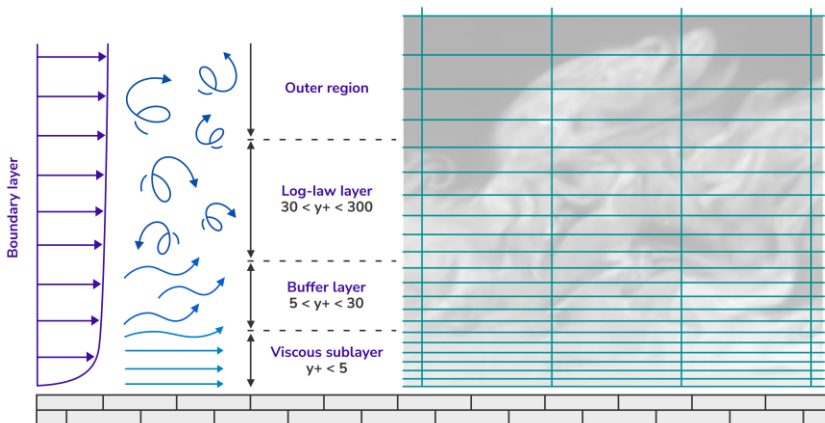
Physical velocity profile – Actual profile

- 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.

# Introduction to turbulence modeling

## 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, all the way down to the viscous sub-layer, you need very fine meshes close and normal to the wall.
- In terms of  $y^+$ , you need to cluster at least 5 to 10 layers at  $y^+ < 5$ .
  - You need to properly resolve the profiles of the transported quantities ( $U$ ,  $k$ , epsilon, Reynolds stresses and so on).
- Usually, this kind of meshes will cluster from 15 to 30 layers (or even more) close to the walls.
- This is the most accurate approach, but it is computationally expensive.



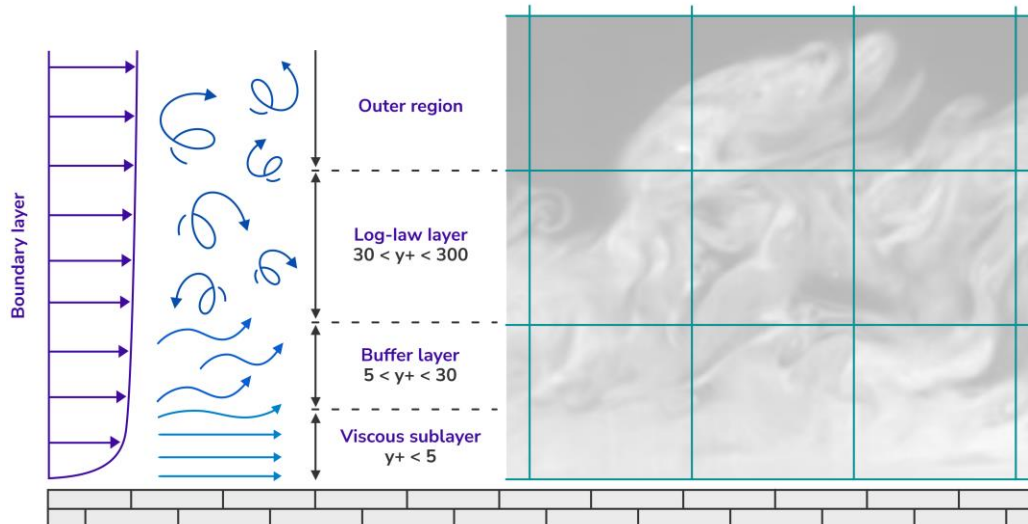
Resolving the streamwise direction is also important, for example, when dealing with transition to turbulence.



# Introduction to turbulence modeling

## Near-wall treatment and wall functions

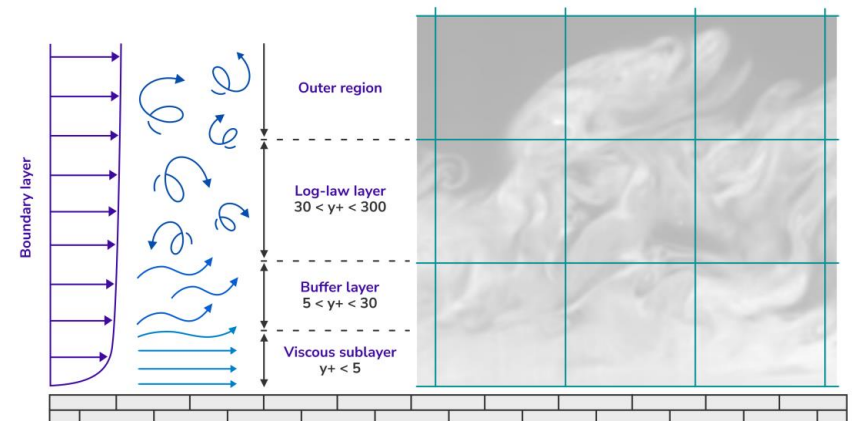
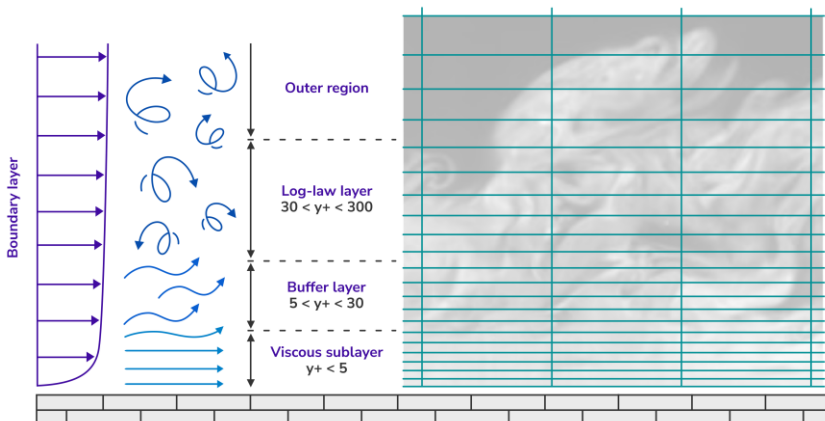
- When dealing with wall turbulence, we need to choose a near-wall treatment.
- If you are not interested in resolving the boundary layer up to the viscous sub-layer, you can use wall functions.
- In terms of  $y^+$ , wall functions will model everything below  $y^+ < 30$  or the target  $y^+$  value.
- This approach uses 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 close to the walls in order to resolve the profiles of the transported quantities (U, k, epsilon, Reynolds stresses and so on).
- As a general rule, when using wall functions, the first cell center should be located above  $y^+ > 40-50$  and below  $y \approx 0.2\delta_{99}$  (boundary layer thickness).



# Introduction to turbulence modeling

## 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  $1 < y^+ < 300-600$  (the upper limit depends on the Reynolds number).
- This approach is very flexible as it is independent of the  $y^+$  value, but is not available in all turbulence models
- Again, you should cluster enough cells close and normal to the walls to resolve the profiles of the transported quantities (at least 8-10 layers).



Insensitive wall treatment will automatically switch between the wall modeling approach or the wall resolving approach according to the  $y^+$  value.

# Introduction to turbulence modeling

## Near-wall treatment and wall functions

- Wall functions is the approach to use if you are more interested in the mixing in the outer region, rather than the accurate prediction of forces on the walls.
- If accurate prediction of forces or heat transfer rate on the walls are key to your simulation (aerodynamic drag, turbomachinery blade performance, heat transfer) you might not want to use wall functions.
- If you are conducting 2D steady simulations, do not be afraid of using a wall resolving approach.
- On the other hand, if you are conducting 3D steady simulations, everything depends on your computational resources, but if you can afford it, use a wall resolving approach.
- In all the other situations (unsteady cases) and if you do not have any restrictions in the near-wall treatment, use wall functions.
- But again, it depends on your computational resources and time available.
- 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.

# Introduction to turbulence modeling

## Near-wall treatment and 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 (buffer layer), and wall functions do not perform very well here.
  - Still, nobody knows what is going on in this region.
- If you are using the HRN approach and a cell-centered solver, the first cell center should be located in a region where  $y^+ > 30$ .
- The low-RE approach is computational expensive as it requires clustering a lot cells near the walls.
- To get good results with LRN, 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.

# Introduction to turbulence modeling

## Near-wall treatment and wall functions

- You can also use wall functions with moving walls.
- In practice, maintaining a fixed value of  $y^+$  in the cells next to the walls throughout the domain is very challenging.
  - You should monitor the average value.
- Maintaining a value of  $y^+ > 30$  when using wall functions during grid refinement studies can be difficult and problematic.
  - Be careful, some wall functions are not very accurate when  $y^+ < 30$ .
- Wall treatments that are insensitive to  $y^+$  values are preferred.
- Grid refinement studies are a common practice in CFD and a recommended best practice.
  - However, if you are planning to conduct such studies, it is strongly recommended to use  $y^+$  insensitive wall functions or a wall resolving approach.
  - Avoid using wall functions.

# Introduction to turbulence modeling

## Near-wall treatment and wall functions

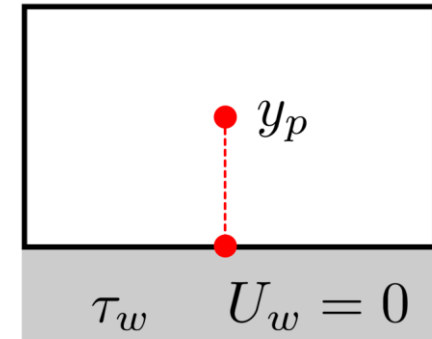
- If we are dealing with globally laminar flows or with turbulent flows with meshes fine enough to resolve the viscous sublayer, we can compute the wall shear stress as follows,

In the viscous sublayer or with laminar flows we use the molecular viscosity

$$\tau_w = \mu \frac{\partial U}{\partial y} = \mu \frac{U_p - 0}{y_p} = \mu \frac{U_p}{y_p}$$

Recall that in turbulent flows we use the effective viscosity

$$\mu_{eff} = \mu_{molecular} + \mu_{turbulent}$$



- In our notation, the subscript p indicates values at the cell center and the subscripts w indicates values at the walls
- Remember, some solvers use cell-centered quantities, and some solvers use node-centered quantities.
- From now on, we are going to assume that all quantities are computed at the cell center.
- Sometimes in this approach, damping functions are added to gain robustness.

# Introduction to turbulence modeling

## Near-wall treatment and wall functions

- We just described how to compute the wall shear stresses in laminar flows and in turbulent flows using the wall resolving approach.
- As you can see, this approach is straight forward to implement.
- Let us address how to compute the wall shear stresses using the wall modeling approach.
- If we are dealing with turbulent flows and if we are using a coarse mesh such that  $y^+ > 30$ , using the previous relations to compute the wall shear stresses is not accurate anymore.
- We are missing a lot of gradient information if we use the previous approach, namely,

$$\tau_w = \mu \frac{\partial U}{\partial y} = \mu \frac{U_p - 0}{y_p} = \mu \frac{U_p}{y_p}$$

This is not accurate if  $y^+ > 10$ .  
This value roughly corresponds to the intersection of the viscous sublayer law and the log-law

- In the wall modeling approach, we need to somehow correct this computation.
- We do this by constructing a bridge between the wall values and the log-law correlations.
- The question is?
  - How do we transfer information from the empirical correlations to the walls and to the flow?
  - How do we compute the wall shear stresses?

# Introduction to turbulence modeling

## Near-wall treatment and wall functions

- By using wall functions, we bridge the wall conditions and cell centered values with the log-law empirical correlations.
- The wall functions reduce the computational effort significantly because we do not need to resolve the viscous sublayer.
- Let us explain the standard wall functions using the method proposed by Launder and Spalding [1], which is probably the most widely used method.
- In this approach,

$$u^* = \begin{cases} y^* & \text{In the viscous sublayer} \\ \frac{1}{\kappa} \ln(Ey^*) & \text{In the log-law layer} \end{cases}$$

- Notice that we are using  $u^*$  and  $y^*$  instead of  $u^+$  and  $y^+$ .
- Also, the log-law layer correlation is slightly different from what we have seen so far.
- Let us address these two issues.



# Introduction to turbulence modeling

## Near-wall treatment and wall functions

- The idea of introducing the new quantity  $u^*$ , is to avoid the singularity that occurs when the wall shear stress is equal to zero in  $u^+$  (i.e., in a separation point).
- Recall that the shear velocity is equal to,

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

- In a separation point, the wall shear stress is equal to zero, therefore,

$$U_\tau = 0 \quad \text{if} \quad \tau_w = 0$$

- If the shear velocity is equal to zero, then we have a singularity when computing  $u^+$ , and this might pose numerical problems.

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

# Introduction to turbulence modeling

## Near-wall treatment and wall functions

- The new quantities  $u^*$  and  $y^*$  are defined as follows [1],

$$U_{\tau}^* = C_{\mu}^{1/4} k_p^{1/2} \quad y^* = \frac{C_{\mu}^{1/4} k_p^{1/2} y_p}{\nu} \quad u^* = \frac{1}{\kappa} \ln(Ey^*)$$

- Notice that these new quantities do not depend anymore on the shear velocity.
- Therefore, in the case of wall shear stress equal to zero, we do not risk anymore a singularity when computing  $u^+$ .
- It is worth noting that  $y^+$  is equal to  $y^*$  in the ideal situation of equilibrium conditions (production equal to dissipation).

# Introduction to turbulence modeling

## Near-wall treatment and wall functions

- All the relations of the standard wall functions formulation of Launder and Spalding [1], can be summarized as follows,

$$u^* = \frac{U_p C_\mu^{1/4} k_p^{1/2}}{\tau_w / \rho}$$

The only unknown quantity is the wall shear stress

$$P_k \approx \tau_w \frac{\partial \bar{u}}{\partial y} = \tau_w \frac{\tau_w}{\kappa \rho C_\mu^{1/4} k_p^{1/2} y_p}$$

$$\epsilon_p = \frac{C_\mu^{3/4} k_p^{3/2}}{\kappa y_p}$$

# Introduction to turbulence modeling

## Near-wall treatment and wall functions

- The boundary condition for TKE at the walls is,

$$\frac{\partial k}{\partial n} = 0$$

- And recall that,

$$u^* = \frac{1}{\kappa} \ln(E y^*) \quad y^* = \frac{\rho C_\mu^{1/4} k_p^{1/2} y_p}{\mu} \quad \underbrace{\kappa = 0.4187 \quad E = 9.793}_{\text{These values may vary depending on the implementation}}$$

- These relations apply only to the cells adjacent to the walls.

# Introduction to turbulence modeling

## Near-wall treatment and wall functions

- We just presented the wall functions for the momentum and turbulence variables.
- Similarly, wall functions can be derived for temperature, species, and so on.
- The approach we just presented, is also known as a log-law based approach.
  - Specifically, we presented the Launder-Spalding methodology [1].
  - Which is probably the most popular approach.
  - However, this does not mean that it is the most robust approach.
- Finally, this is not the only approach when using wall functions.
  - In the literature, you will find many approaches.

# Introduction to turbulence modeling

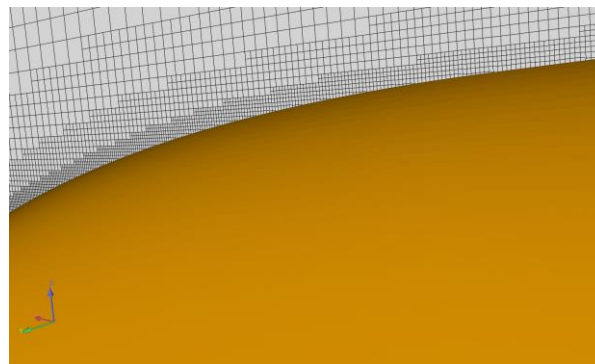
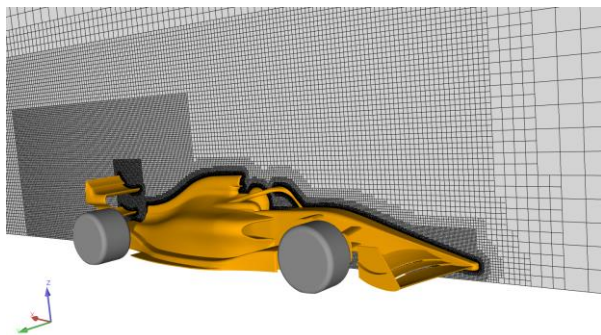
## Near-wall treatment and wall functions

- There are many wall functions implementations, just to name a few,
  - Standard wall functions.
  - Standard wall functions – Launder-Spalding methodology .
  - Generalized wall functions.
  - Scalable wall functions.
  - Non-equilibrium wall functions.
  - Two-layer approach.
  - $y^+$  insensitive wall treatment.
  - Menter-Lechner wall functions.
  - Werner and Wengle wall functions.
  - Craft wall functions.
  - Chien-Launder wall functions.
  - Eddy viscosity-based wall functions.
  - Spalding continuous wall function.
  - Roughness wall functions.
  - And the list keeps going on.

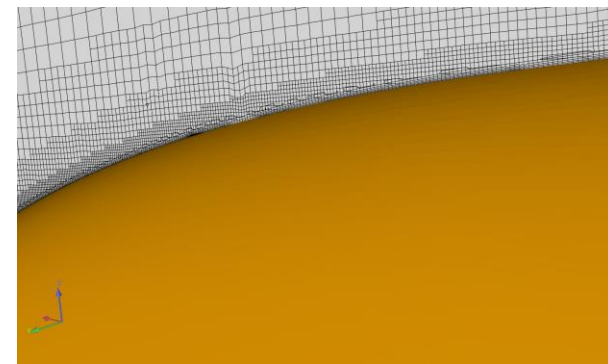
# Introduction to turbulence modeling

## Influence of near-wall treatment in cell count

- In the wall resolving mesh, we only modified the inflation layers parameters (the prismatic cells close to the walls).
- The parameters of the surface mesh and volume mesh remained the same.
- By taking a mesh resolving approach, we almost doubled the cell count.



Wall modeling mesh  
Average  $y^+$  approximately 60



Wall resolving mesh  
Average  $y^+$  approximately 1

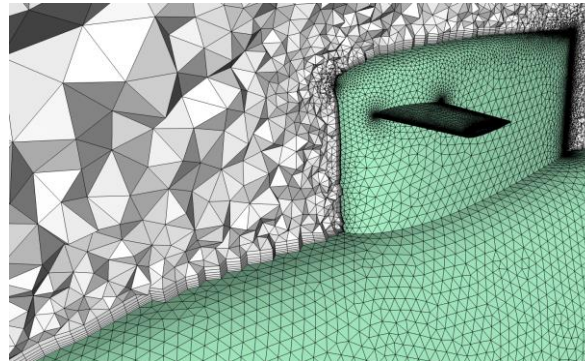
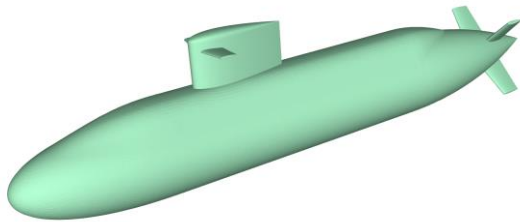
	Wall modeling mesh	Wall resolving mesh
Number of cells	57 853 037	111 137 673

**Note:** we only added seven inflation layers.

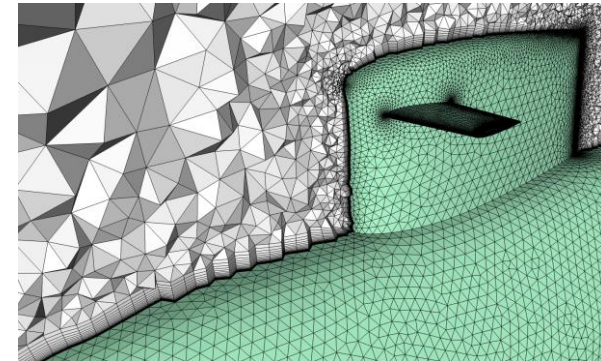
# Introduction to turbulence modeling

## Influence of near-wall treatment in cell count

- In the wall resolving mesh, we only modified the inflation layers parameters (the prismatic cells close to the walls).
- The parameters of the surface mesh and volume mesh remained the same.
- By taking a mesh resolving approach, we almost doubled the cell count.



Wall modeling mesh  
Average  $y^+$  approximately 60



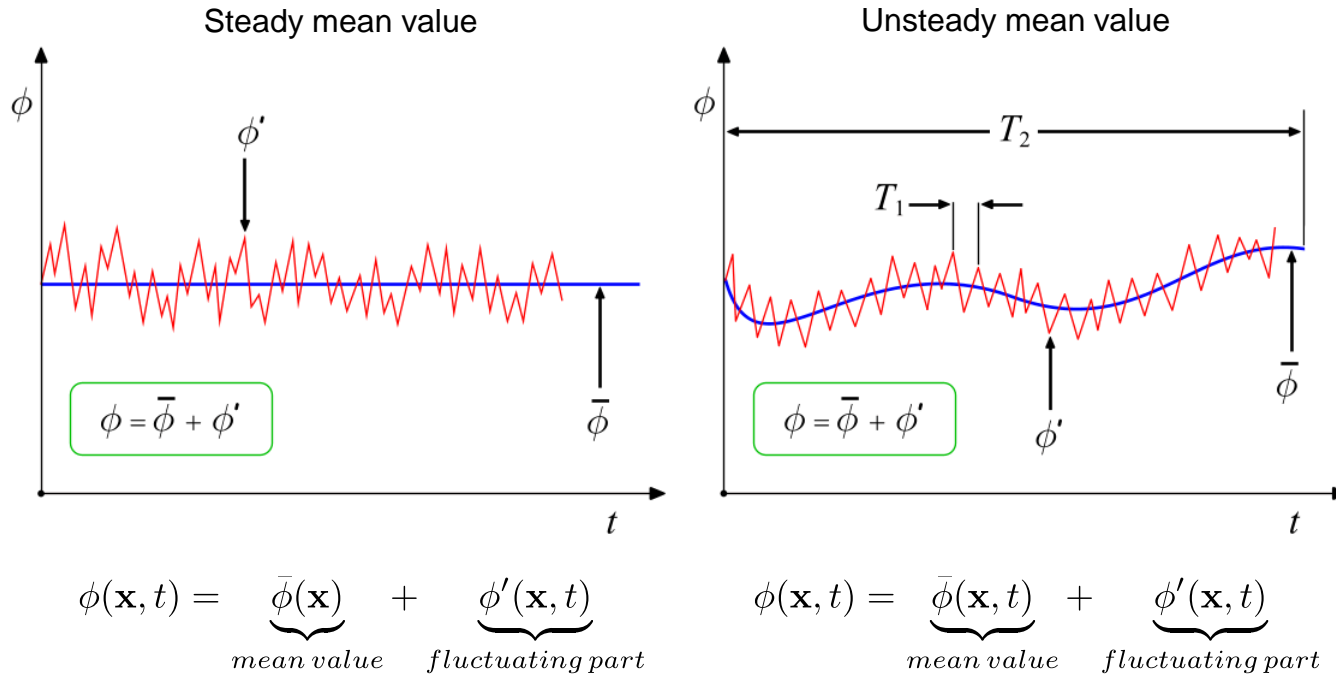
Wall resolving mesh  
Average  $y^+$  approximately 1

	Wall modeling mesh	Wall resolving mesh
Number of cells	6 613 049	11 149 266



# Introduction to turbulence modeling

## Removing small scales



- We have seen that turbulent flows are characterized by instantaneous fluctuations of velocity, pressure, and all transported quantities.
- In most engineering applications it 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.

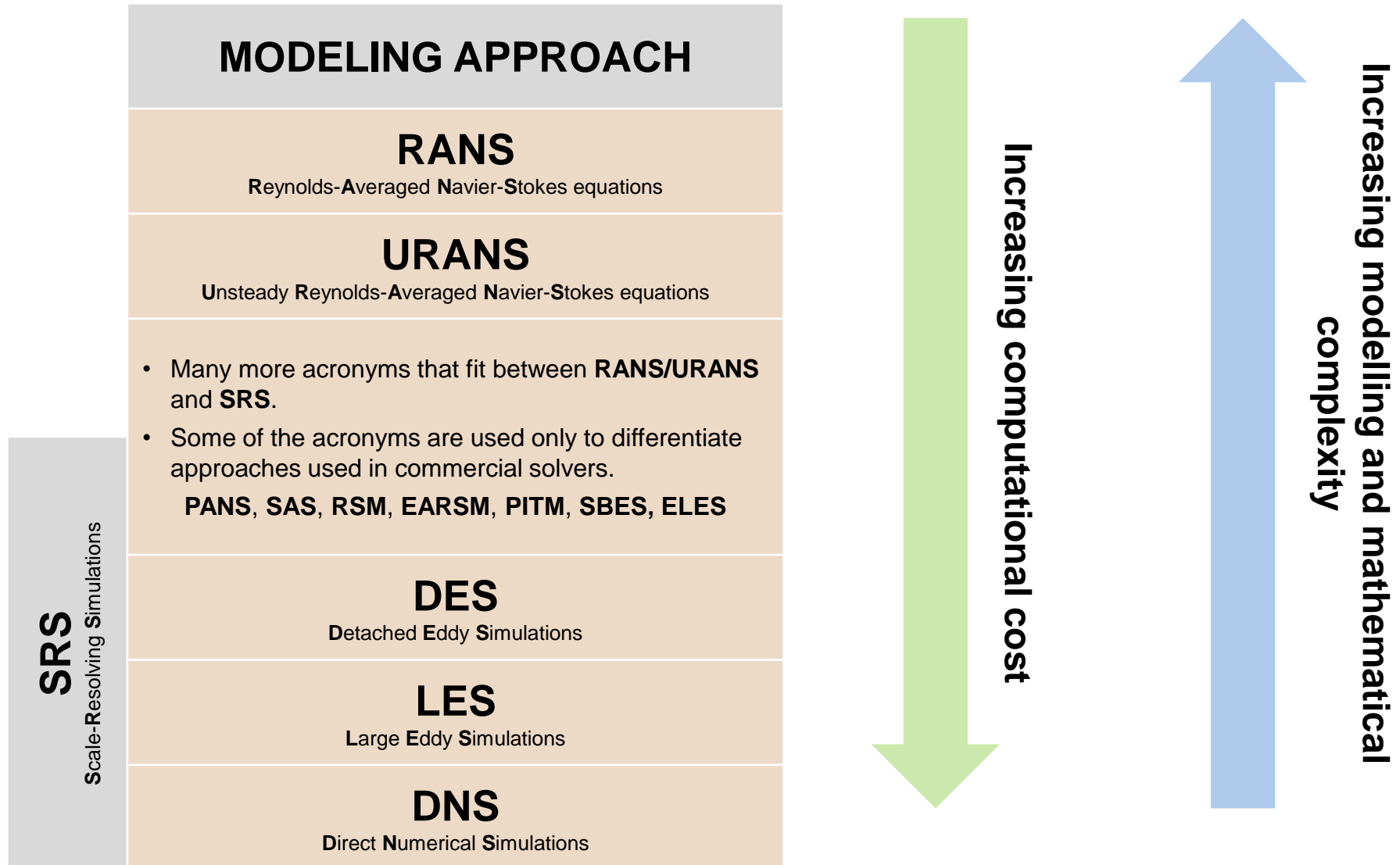
# Introduction to turbulence modeling

## 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.
    - It 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).

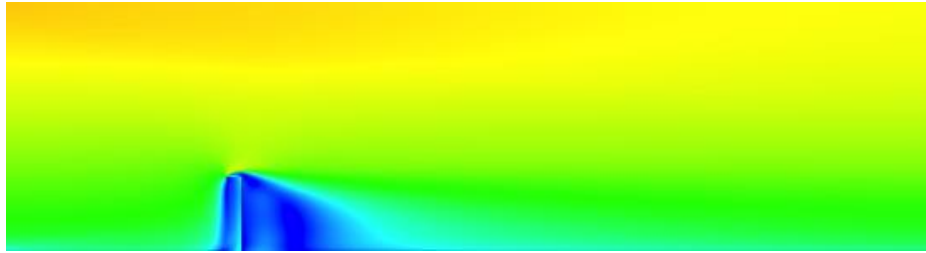
# Introduction to turbulence modeling

Overview of the main turbulence modeling approaches.

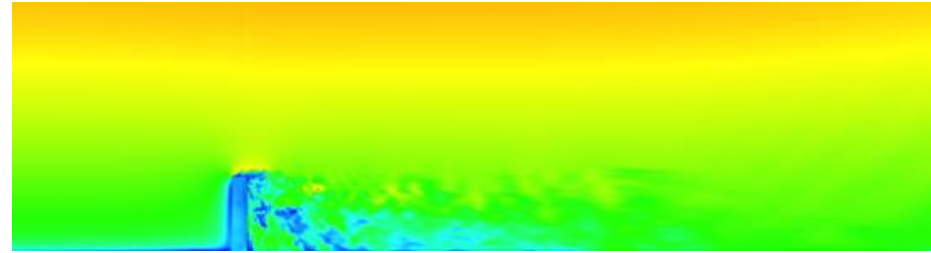


# Introduction to turbulence modeling

## Overview of turbulence modeling approaches



**RANS**



**LES – Instantaneous field**

RANS/URANS	DES/LES	DNS
<ul style="list-style-type: none"><li>• Solve the time-average NSE.</li><li>• All turbulent spatial scales are modeled.</li><li>• Many models are available. One equation models, two equation models, Reynolds stress models, transition models, and so on.</li><li>• This is the most widely approach for industrial flows.</li><li>• Unsteady RANS (URANS), use the same equations as the RANS but with the transient term retained.</li><li>• It can be used in 2D and 3D cases.</li></ul>	<ul style="list-style-type: none"><li>• Solve the filtered unsteady NSE.</li><li>• Sub-grid scales (SGS) are filtered, grid scales (GS) are resolved.</li><li>• Aim at resolving the temporal scales, hence requires small time-steps.</li><li>• For most industrial applications, it is computational expensive. However, thanks to the current advances in parallel and scientific computing it is becoming affordable.</li><li>• Many models are available.</li><li>• It is intrinsically 3D and asymmetric.</li></ul>	<ul style="list-style-type: none"><li>• Solves the unsteady NSE with no models involved (laminar NSE).</li><li>• Solves all spatial and temporal scales; hence, requires extremely fine meshes and small time-steps.</li><li>• No modeling is required.</li><li>• It is extremely computational expensive.</li><li>• Not practical for industrial flows.</li><li>• It is intrinsically 3D and asymmetric.</li></ul>

# Introduction to turbulence modeling

## Overview of turbulence modeling approaches



**LES – Smagorinsky – nut**

<http://www.wolfdynamics.com/training/turbulence/image9.gif>



**LES – U mean**



**URANS – SA – nut**

<http://www.wolfdynamics.com/training/turbulence/image10.gif>



**URANS SA – U mean**

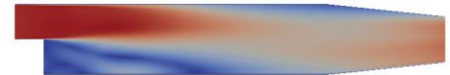


**URANS – kw – nut**

<http://www.wolfdynamics.com/training/turbulence/image11.gif>



**URANS kw – U mean**

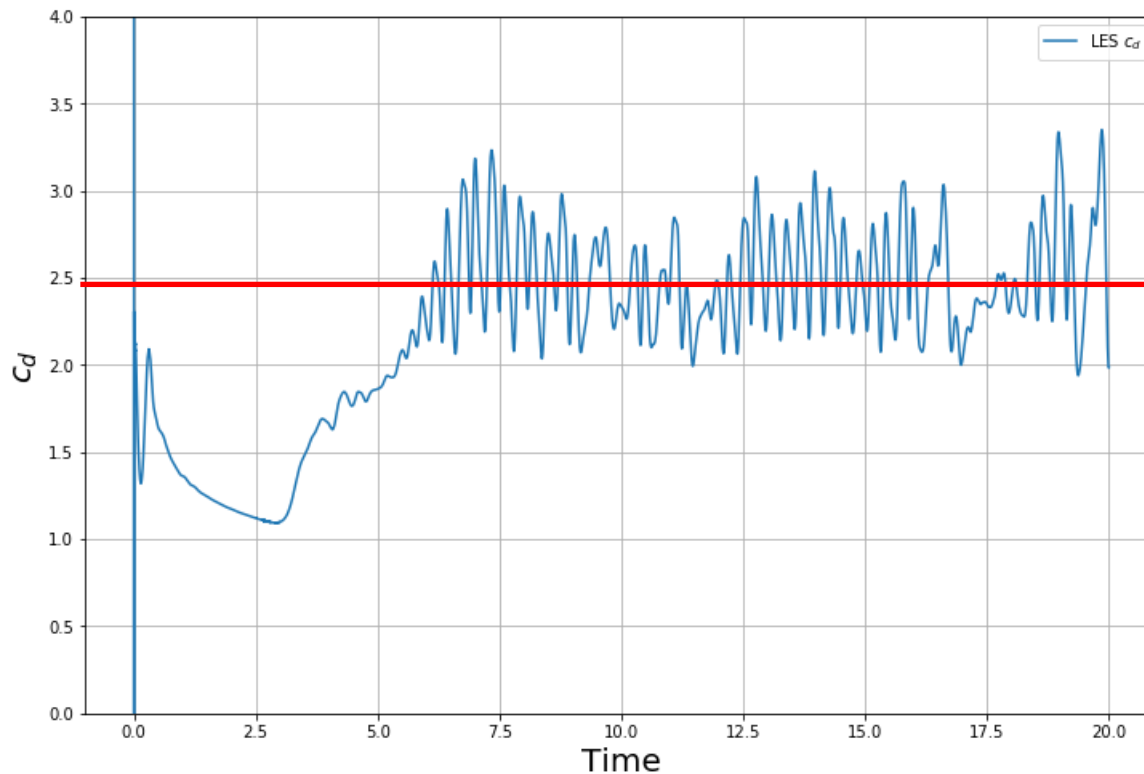


- The instantaneous field of each method might be slightly different, but when we average the solution, we obtain similar fields.
- All cases use the same mesh and have approximately the same computational overload.
- Unsteadiness makes turbulence simulations time consuming.

# Introduction to turbulence modeling

## Overview of turbulence modeling approaches

- Most of the times we are not interested in the time evolution (unsteadiness).
- 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 advisable to compute the descriptive statistics in a given iterative 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

# Introduction to turbulence modeling

## Turbulence modeling – Starting equations

$$\text{Exact NSE} \left\{ \begin{aligned} \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) &= 0 \\ \frac{\partial (\rho \mathbf{u})}{\partial t} + \nabla \cdot (\rho \mathbf{u} \mathbf{u}) &= -\nabla p + \nabla \cdot \boldsymbol{\tau} + \mathbf{S}_u \\ \frac{\partial (\rho e_t)}{\partial t} + \nabla \cdot (\rho e_t \mathbf{u}) &= -\nabla \cdot q - \nabla \cdot (p \mathbf{u}) + \boldsymbol{\tau} : \nabla \mathbf{u} + \mathbf{S}_{e_t} \end{aligned} \right.$$

+

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!

# Introduction to turbulence modeling

## Incompressible RANS equations

- Let us derive the Reynolds average equations for an incompressible flow.
- This is our starting point, the exact Navier-Stokes equations (no models),

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

- 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).



# Introduction to turbulence modeling

## Incompressible RANS equations

- To derive the incompressible RANS equations, we need to apply Reynolds decomposition to the primitive variables of the governing equations.
- The Reynolds decomposition consists in splitting the instantaneous value of a variable into a mean component and a fluctuating component, as follows,

The diagram illustrates the Reynolds decomposition of a variable  $\phi$ . It shows the equation  $\phi(\mathbf{x}, t) = \bar{\phi}(\mathbf{x}) + \phi'(\mathbf{x}, t)$ . A red arrow points from the label "Instantaneous value" to  $\phi(\mathbf{x}, t)$ . Another red arrow points from the label "Mean component" to  $\bar{\phi}(\mathbf{x})$ . A third red arrow points from the label "Fluctuating component" to  $\phi'(\mathbf{x}, t)$ .

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

- In our notation, the overbar represents the average (or mean) value, and the prime (or apostrophe) represents the fluctuating part.

# Introduction to turbulence modeling

## Incompressible RANS equations

- Before continuing, let us recall a few averaging rules that we will use when deriving the RANS equations.

$$\begin{aligned}\bar{\phi}' &= 0, & \overline{\phi\varphi} &= \overline{(\bar{\phi} + \phi')(\bar{\varphi} + \varphi')} \\ \bar{\bar{\phi}} &= \bar{\phi}, & &= \overline{\bar{\phi}\bar{\varphi} + \bar{\phi}\varphi' + \bar{\varphi}\phi' + \phi'\varphi'} \\ \bar{\bar{\phi}} &= \overline{\bar{\phi} + \phi'} = \bar{\phi}, & &= \overline{\bar{\phi}\bar{\varphi}} + \overline{\bar{\phi}\varphi'} + \overline{\bar{\varphi}\phi'} + \overline{\phi'\varphi'} \\ \overline{\phi + \varphi} &= \bar{\phi} + \bar{\varphi}, & &= \bar{\phi}\bar{\varphi} + \overline{\phi'\varphi'}, \\ \overline{\bar{\phi}\varphi} &= \bar{\bar{\phi}}\bar{\varphi} = \bar{\phi}\bar{\varphi}, & \overline{\phi'^2} &\neq 0, \\ \overline{\bar{\phi}\varphi'} &= \bar{\phi}\bar{\varphi}' = 0, & \overline{\phi'\varphi'} &\neq 0, \\ \frac{\partial \bar{\phi}}{\partial x} &= \frac{\partial \bar{\phi}}{\partial x}, & \int \phi ds &= \int \bar{\phi} ds\end{aligned}$$

# Introduction to turbulence modeling

## Incompressible RANS equations

- To derive the incompressible RANS equations, we need to apply Reynolds decomposition to the primitive variables of the governing 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 a few 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  
This term requires modeling

$$\tau^R = -\rho (\overline{\mathbf{u}'\mathbf{u}'})$$

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

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

# Introduction to turbulence modeling

## 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  $\tau^R$ .

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

- 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.

# Introduction to turbulence modeling

## 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,

$$\tau^R = -\rho \overline{(\mathbf{u}'\mathbf{u}')} = 2\mu_t \bar{\mathbf{S}}^R - \frac{2}{3}\rho k \mathbf{I} = \mu_t [\nabla \bar{\mathbf{u}} + \nabla \bar{\mathbf{u}}^T] - \frac{2}{3}\rho k \mathbf{I}$$

- In the previous equation,  $\bar{\mathbf{S}}^R = \frac{1}{2} [\nabla \bar{\mathbf{u}} + \nabla \bar{\mathbf{u}}^T]$  denotes the strain-rate tensor.
- $\mathbf{I}$  is the identity matrix.
- $\mu_t$  is the turbulent eddy viscosity.
- $k = \frac{1}{2} \overline{\mathbf{u}' \cdot \mathbf{u}'}$  or  $k = \frac{1}{2} (\overline{u'^2} + \overline{v'^2} + \overline{w'^2})$  is the turbulent kinetic energy.
- At the end of the day we want to determine the turbulent eddy viscosity. 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.

# Introduction to turbulence modeling

## 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,

$$\bar{\phi}(\mathbf{x}) = \lim_{T \rightarrow +\infty} \frac{1}{T} \int_t^{t+T} \phi(\mathbf{x}, t) dt \quad \text{where} \quad \phi(\mathbf{x}, t) = \bar{\phi}(\mathbf{x}) + \phi'(\mathbf{x}, t)$$

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

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

- 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).

# Introduction to turbulence modeling

## Incompressible RANS equations

- To derive the incompressible RANS equations, we need to apply Reynolds averaging to the governing equations.
- Reynolds averaging simple consists in:
  - Splitting the instantaneous value of the primitive variables into a mean component and a fluctuating component (Reynolds decomposition).
  - Averaging the quantities (time average, spatial average or ensemble average).
  - Applying a few averaging rules to simplify the equations.
- When we use Reynolds averaging, we are taking a statistical approach to turbulence modeling.
- When we do DNS, we take a deterministic approach to turbulence modeling.
- Usually, we are interested in the mean behavior of the flow.
- Therefore, by applying Reynolds averaging, we are only solving for the averaged variables and the fluctuations are modeled.

# Introduction to turbulence modeling

## 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 give some final touches to the incompressible RANS equations before looking at one model.



# Introduction to turbulence modeling

## Final touches to the incompressible RANS equations

- The RANS equations are very similar to the starting equations.

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

RANS/URANS equations

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

NSE with no turbulence models (DNS)

- The differences are that all quantities have been averaged (the overbar over the primitive variables).
- And the appearance of the Reynolds stress tensor  $\boldsymbol{\tau}^R$ .
- Notice that the Reynolds stress tensor is not actually a stress, it must be multiplied by density in order to have dimensions corresponding to stresses,

$$\boldsymbol{\tau}^R = -\rho \overline{(\mathbf{u}'\mathbf{u}')}$$

# Introduction to turbulence modeling

## Final touches to the incompressible RANS equations

- Recall the incompressible RANS equations,

$$\nabla \cdot (\bar{\mathbf{u}}) = 0$$
$$\frac{\partial \bar{\mathbf{u}}}{\partial t} + \nabla \cdot (\bar{\mathbf{u}}\bar{\mathbf{u}}) = -\frac{1}{\rho} (\nabla p) + \nu \nabla^2 \bar{\mathbf{u}} + \frac{1}{\rho} \nabla \cdot \boldsymbol{\tau}^R \quad \text{where} \quad \boldsymbol{\tau}^R = -\rho \overline{(\mathbf{u}'\mathbf{u}')}$$

- By using the Boussinesq approximation,

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

- We can write the governing equations in function of the turbulent eddy viscosity  $\mu_T$ .

# Introduction to turbulence modeling

## Final touches to the incompressible RANS equations

- After introducing the Boussinesq approximation and doing some algebra, we obtain the following form of the governing equations (solvable equations),

$$\nabla \cdot (\bar{\mathbf{u}}) = 0$$
$$\frac{\partial \bar{\mathbf{u}}}{\partial t} + \nabla \cdot (\bar{\mathbf{u}}\bar{\mathbf{u}}) = -\frac{1}{\rho} \left( \nabla \bar{p} + \frac{2}{3} \rho \nabla k \right) + \nabla \cdot \left[ \frac{1}{\rho} \overbrace{(\mu + \mu_t)}^{\text{Effective viscosity}} \nabla \bar{\mathbf{u}} \right]$$

Normal stresses arising from the Boussinesq approximation

Turbulent viscosity

- The problem now reduces to computing the turbulent eddy viscosity  $\mu_T$  in the momentum equation.
- Each turbulence model computes the turbulent eddy viscosity in a different way,

$$\mu_t = f(k, \epsilon, \omega, l, t, \nu, \dots)$$

# Introduction to turbulence modeling

## Final touches to the incompressible RANS equations

- To compute the turbulent eddy viscosity, we can use any of the RANS turbulence models available in literature (or in OpenFOAM for the effect of this training).
- The RANS turbulence models can be classified as follows,
  - Zero equation models.
  - Half equation models.
  - One equation models.
  - Two equation models.
  - Three, four, five, ... , equation models.
  - Transition models.
  - Reynolds stress models.
  - And so on.
- Hereafter, we will address the most widely used approaches.
- At this point, it only rest exploring some RANS/URANS turbulence models.

# Introduction to turbulence modeling

## Final touches to the incompressible RANS equations

- In most turbulence models, a relationship for the turbulent eddy viscosity is derived using dimensional arguments.
  - This can be done by using any combination of dimensional groups, that is, velocity, length, time, etc.
  - In the end, we should have viscosity units.
- This relationship can be corrected later or validated based on empirical and physical arguments, *e.g.*,
  - Asymptotic analysis.
  - Canonical solutions.
  - Analytical solutions.
  - Consistency with experimental measurements.
  - And so on.

# Introduction to turbulence modeling

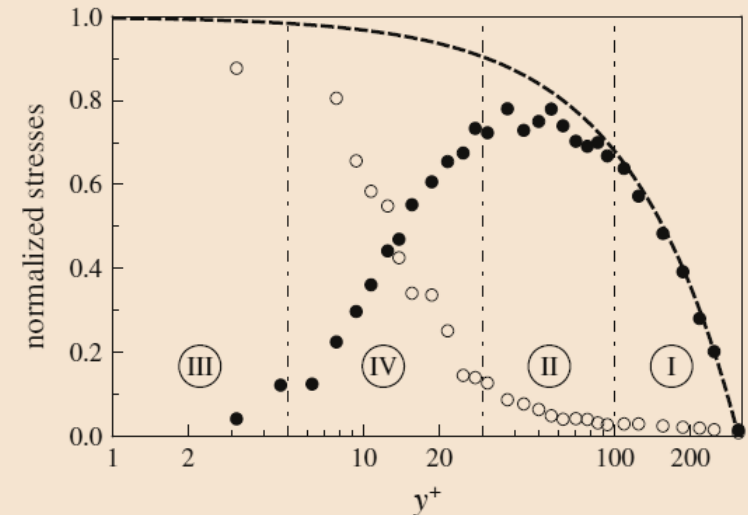
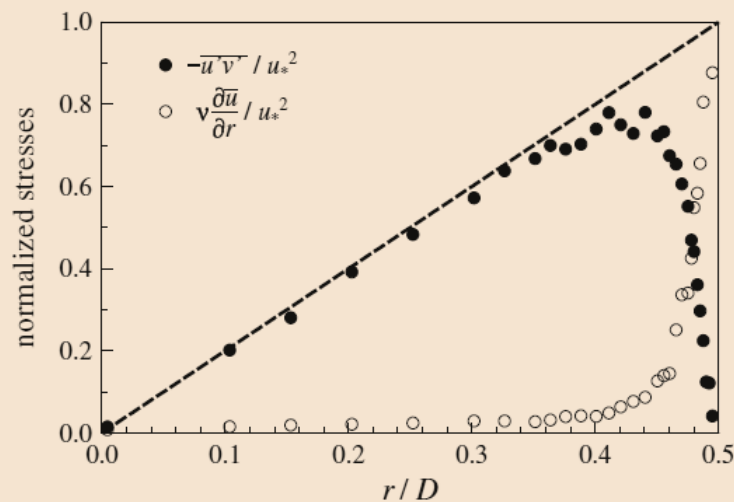
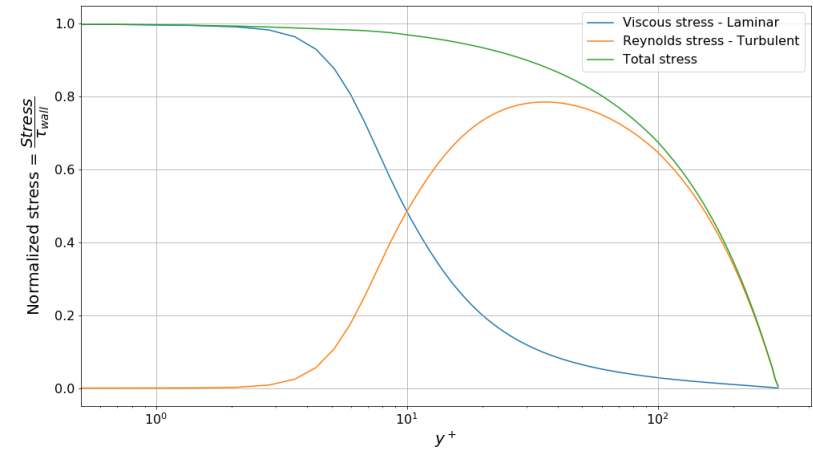
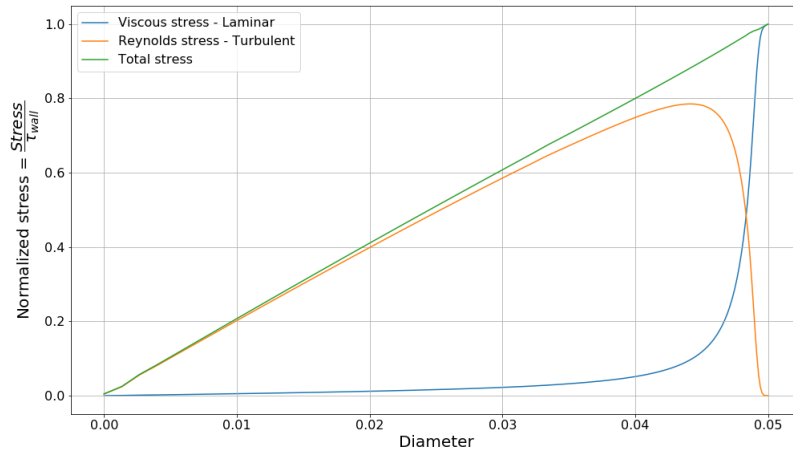
## Final touches to the incompressible RANS equations

- It is also possible the use of numerical arguments to correct, calibrate, and validate the relationship.
- To achieve this end, we rely on scale resolving simulations.
  - Most of the time DNS simulations or well resolved LES simulations.
- Regardless of the approach or model used (one equation, two equations, Reynolds stress models, etc), we see a recurring behavior.
  - Specifically, eddy viscosity and length scale are all related on the basis of dimensional arguments.
- Historically, dimensional analysis has been one of the most powerful tools available for deducing and correlating properties of turbulent flows.
- However, we should always be aware that while dimensional analysis is extremely useful, it unveils nothing about the physics underlying its implied relationships.

# Introduction to turbulence modeling

## RANS simulation – A simple validation case

- Reynolds and viscous shear stresses distribution – Comparison of numerical results (top row) and experimental results (bottom row)

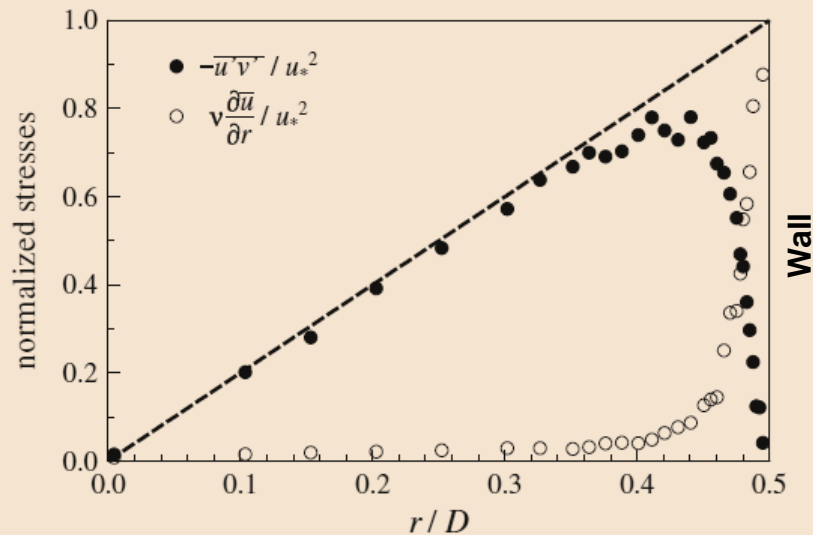


[1] F. Nieuwstadt, B. Boersma, J. Westerweel. Turbulence. Introduction to Theory and Applications of Turbulent Flows. Springer, 2016.

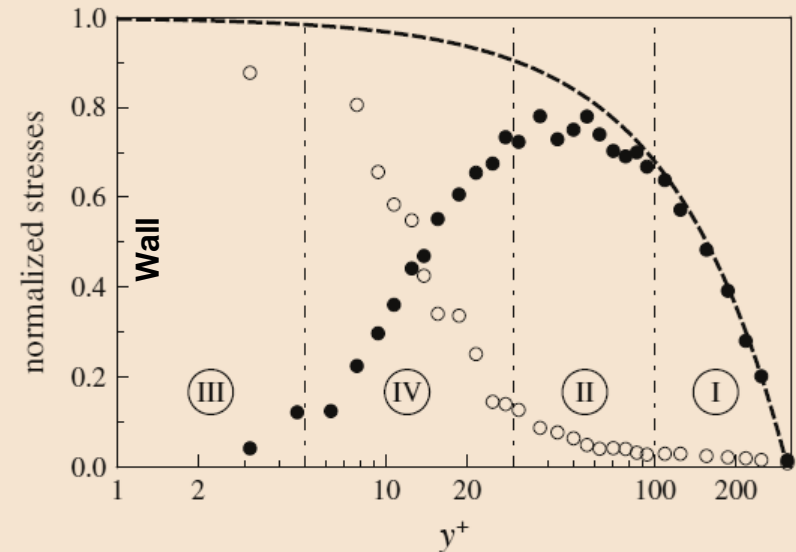
# Introduction to turbulence modeling

## RANS simulation – A simple validation case

- Reynolds and viscous shear stresses distribution – Turbulent flow in a pipe (experimental data)



The total normalized stress (using wall shear stress), as a function of the distance  $r$  from the centerline of a pipe with diameter  $D$ . The total stress consists of a contribution from the Reynolds stress (black circles) and the viscous stress (empty circles). Experimental data for a turbulent pipe flow at  $Re = 10000$  [1].



The same data as in the left figure, but now as a function of the dimensionless distance  $y^+$  from the pipe wall in a semi-log plot. In the figure, I = core region; II = logarithmic wall region; III = viscous sublayer; IV = buffer layer. Note that  $r = 0$  corresponds to  $y^+ = 312$  and  $r = 0.5D$  to  $y^+ = 0$  [1].

### Exact RANS equation

$$\frac{\partial \bar{\mathbf{u}}}{\partial t} + \nabla \cdot (\bar{\mathbf{u}}\bar{\mathbf{u}}) = -\frac{1}{\rho} (\nabla \bar{p}) + \underbrace{\nu \nabla^2 \bar{\mathbf{u}} - \frac{1}{\rho} \nabla \cdot (\rho \overline{\mathbf{u}'\mathbf{u}'})}_{\tau_{\text{Total}}}$$

\* Viscous stress

\*\* Reynolds stress

### Solvable RANS equation

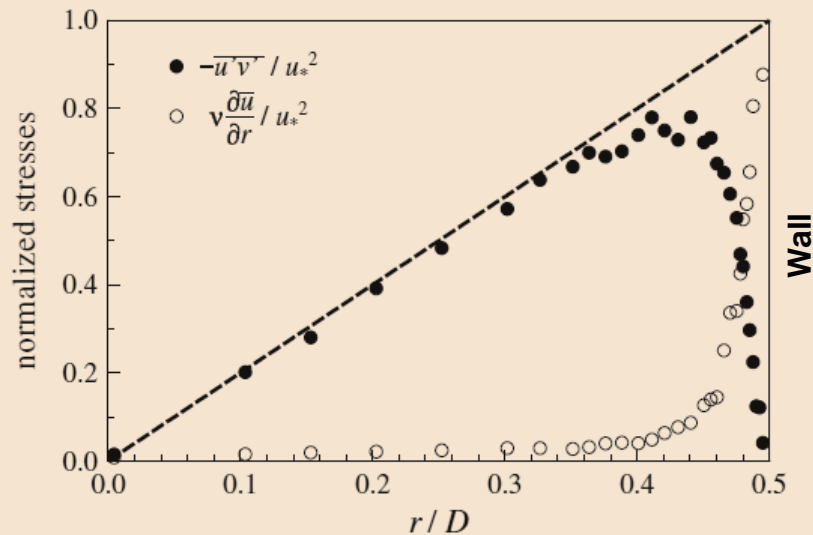
$$\frac{\partial \bar{\mathbf{u}}}{\partial t} + \nabla \cdot (\bar{\mathbf{u}}\bar{\mathbf{u}}) = -\frac{1}{\rho} \left( \nabla \bar{p} + \frac{2}{3} \rho \nabla k \right) + \underbrace{\nabla \cdot \left[ \frac{1}{\rho} (\mu + \mu_t) \nabla \bar{\mathbf{u}} \right]}_{\tau_{\text{Total}}}$$



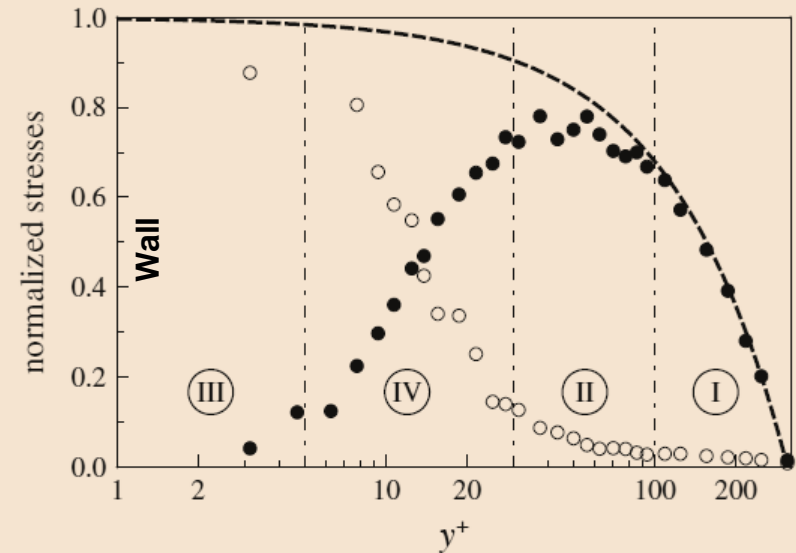
# Introduction to turbulence modeling

## RANS simulation – A simple validation case

- Reynolds and viscous shear stresses distribution – Turbulent flow in a pipe (experimental data)



The total normalized stress (using wall shear stress), as a function of the distance  $r$  from the centerline of a pipe with diameter  $D$ . The total stress consists of a contribution from the Reynolds stress (black circles) and the viscous stress (empty circles). Experimental data for a turbulent pipe flow at  $Re = 10000$  [1].



The same data as in the left figure, but now as a function of the dimensionless distance  $y^+$  from the pipe wall in a semi-log plot. In the figure, I = core region; II = logarithmic wall region; III = viscous sublayer; IV = buffer layer. Note that  $r = 0$  corresponds to  $y^+ = 312$  and  $r = 0.5D$  to  $y^+ = 0$  [1].

- Close to the walls, the viscous stress dominates, and as we get far from the wall, the Reynolds stress increases.
- In reference to the right figure. In region I and II the Reynolds stress dominates. In region III the viscous stress dominates. In region IV, both, the Reynolds stress and the viscous stress are important.
- The buffer layer is very energetic.

# Introduction to turbulence modeling

- Remember, before starting to use any turbulence model, it is strongly recommended to know its range of applicability and limitations.
- It is also important to know the recommended values for the boundary conditions and initial conditions.
- And any other information that may be useful when setting the simulation.
- Therefore, it is extremely recommended to read the original source of documentation of the model.
- This can be a paper, the source code, or the help system of the CFD solver you are using.
- From now on, we will always mention the specific version of the turbulence model that we are going to use, and we will give some useful references.
- You do not do CFD and turbulence modeling without understanding the theoretical background.
  - So, do not say that you have not been warned.

# Introduction to turbulence modeling

## The $k - \omega$ family of Turbulence models

- This is probably the most widely used family of two-equation turbulence models.
- It is based on the Boussinesq hypothesis (Eddy viscosity models or EVM).
- The initial development of this model can be attributed to Kolmogorov [1], circa 1942. This was the first two-equation model of turbulence.
- The method was further developed and improved by Saffman [2], Launder and Spalding [3], Wilcox [4,5], Menter [6] and many more.
- There are many variations of this model. Hereafter, we will address the  $k - \omega$  Wilcox 1988 model, which probably is the first formulation of the modern  $k - \omega$  family of turbulence models.
- Each variation is designed to add new capabilities and overcome the limitations of the predecessor formulations.
- This family of models is  $y^+$  insensitive.
- Variants of this family of turbulence models include the Wilcox 1998  $k - \omega$  , Wilcox 2006  $k - \omega$  , and Menter 2003  $k - \omega$  SST.

### References:

- [1] A. N. Kolmogorov. Equations of Turbulent Motion in an Incompressible Fluid. Physics. 1941.
- [2] P. Saffman. A Model for Inhomogeneous Turbulent Flow. Proceedings of the Royal Society of London. 1970.
- [3] B. E. Launder, D. B. Spalding. Mathematical Models of Turbulence. Academic Press. 1972.
- [4] D. C. Wilcox. Reassessment of the Scale-Determining Equation for Advanced Turbulence Models. AIAA Journal, 1988.
- [5] D. C. Wilcox. Turbulence Modeling for CFD. DCW Industries, 2010.
- [6] F. Menter, M. Kuntz, R. Langtry. Ten Years of Industrial Experience with the SST Turbulence Model. Turbulence, Heat and Mass Transfer. 2003.

# Introduction to turbulence modeling

## $k - \omega$ Turbulence model overview (Wilcox 1998 revision)

- 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  $\omega$  of dissipation

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

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

**Note:**

$$\tau^R = -\rho (\overline{\mathbf{u}' \mathbf{u}'})$$

- 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.
- This model uses the following relation for the kinematic eddy viscosity,

$$\nu_t = \frac{k}{\omega}$$

# Introduction to turbulence modeling

## $k - \omega$ Turbulence model overview (Wilcox 1998 revision)

- 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  $\omega$  of dissipation

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

**Note:**

$$\tau^R = -\rho (\overline{\mathbf{u}' \mathbf{u}'})$$

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

- With the following closure coefficients,

$$\alpha = 5/9 \quad \beta = 3/40 \quad \beta^* = 9/100 \quad \sigma = 1/2 \quad \sigma^* = 1/2$$

- And auxiliary relationships,

$$\epsilon = \beta^* \omega k \quad l = \frac{k^{1/2}}{\omega}$$

- And recall that the turbulent kinetic energy is equal to  $k = \frac{1}{2} (\overline{u'^2} + \overline{v'^2} + \overline{w'^2})$

# Introduction to turbulence modeling

## $k - \omega$ Turbulence model overview (Wilcox 1998 revision)

- The closure equations of the Wilcox (1988)  $k - \omega$  model have been manipulated so there are no terms including fluctuating quantities (*i.e.*, velocity and pressure), and double or triple correlations of the fluctuating quantities.
- Using compact vector notation, the closure equations can be written as,

$$\begin{aligned} \nabla_t k + \nabla \cdot (\bar{\mathbf{u}} k) &= \underbrace{\tau^R : \nabla \bar{\mathbf{u}}}_{\text{Production}} - \underbrace{\beta^* k \omega}_{\text{Dissipation}} + \underbrace{\nabla \cdot [(\nu + \sigma^* \nu_t) \nabla k]}_{\text{Diffusion}} \\ \nabla_t \omega + \nabla \cdot (\bar{\mathbf{u}} \omega) &= \underbrace{\alpha \frac{\omega}{k} \tau^R : \nabla \bar{\mathbf{u}}}_{\text{Production}} - \underbrace{\beta \omega^2}_{\text{Dissipation}} + \underbrace{\nabla \cdot [(\nu + \sigma \nu_t) \nabla \omega]}_{\text{Diffusion}} \end{aligned}$$

**Note:**

$$\tau^R = -(\overline{\mathbf{u}' \mathbf{u}'})$$

- Each equation requires the definition of appropriate boundary conditions and initial conditions.
- This model can use wall function (HRN) or can resolve the boundary layer (LRN).

# Introduction to turbulence modeling

## $k - \omega$ Turbulence model overview (Wilcox 1998 revision)

- When using wall functions, the following equations are applied in its simplest form at 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 functions for the  $k - \omega$  method.
- They relate the local wall shear stress (through the shear velocity  $u_\tau$ ) to the mean velocity, turbulence kinetic energy  $k$ , and rate of dissipation  $\omega$ .
- A small implicit system must be solved in an iterative way in order to compute the friction velocity  $u_\tau$ .
- First, we compute the friction velocity, then we compute  $k$ , and then we find  $\omega$ .
- 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 ( $u$ ,  $k$ , and  $\omega$ ) so that the turbulence model equations are not solved close to the wall.
- This is not the only formulation of wall functions; many methods have been implemented.
- Also, each turbulence model formulation has different wall function equations.

# Introduction to turbulence modeling

## $k - \omega$ Turbulence model overview (Wilcox 1998 revision)

- When resolving the boundary layer, you can use the following boundary values at the walls,

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

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>



# Introduction to turbulence modeling

## $k - \omega$ Turbulence model overview (Wilcox 1998 revision)

- The free-stream 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  $\mu_t/\mu$  is the viscosity ratio and  $I = u'/\bar{u}$  is the turbulence intensity (recommended values in the next slide).

# Introduction to turbulence modeling

## $k - \omega$ Turbulence model overview (Wilcox 1998 revision)

- 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 %
$\mu_t/\mu$	1	10	100

- Where  $\mu_t/\mu$  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.

# Introduction to turbulence modeling

## Short description of some RANS turbulence models

Model	Short description
<b>Spalart-Allmaras</b>	This is a one equation model. Suitable for external aerodynamics, turbomachinery 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.
<b>Standard k-epsilon</b>	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.
<b>Realizable k-epsilon</b>	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.
<b>Standard k-omega</b>	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).
<b>SST k-omega</b>	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.

# Introduction to turbulence modeling

General form of turbulence models

# Introduction to turbulence modeling

- The transport equations of all turbulence models can be expressed in general form as follows,

$$\underbrace{\nabla_t \phi}_{\text{Transient term}} + \underbrace{\nabla \cdot \bar{\mathbf{u}} \phi}_{\text{Convection}} = \underbrace{P^\phi}_{\text{Production}} + \underbrace{\epsilon^\phi}_{\text{Dissipation}} + \underbrace{D^\phi}_{\text{Diffusion}} + \underbrace{S^\phi}_{\text{Source terms}}$$

- Where  $\phi$  represents the transported turbulent quantity.
- In general, all equations used in turbulence modeling share the same similarities. Namely,
  - A production term or the eddy factory.
  - A dissipation or destruction term, where eddies are destroyed (the eddy graveyard).
  - A turbulence diffusion term that is responsible for the transport, diffusion, and redistribution due to turbulence.
  - Plus, any additional source term, such as buoyancy or gravity forces.

# Introduction to turbulence modeling

- It is worth mentioning that the production term appearing in the turbulence equations, it always shows a similar structure.
- That is, it is the product of the Reynolds stress tensor times the mean velocity gradient, times some proportionality constant.
- Using index notation, the production term can be written as follows,

$$\text{Constant} \times \tau_{ij} \frac{\partial \bar{u}_i}{\partial x_j}$$

- This operation represents the scalar product of two second-rank tensors.
- Using vector notation, the production term is written as follows,

$$\text{Constant} \times \boldsymbol{\tau} : \nabla \bar{\mathbf{u}}$$

- Where the semicolons (:) represents the double dot product or the scalar product of two second-rank tensors.
- And to some extension the same can be said for the dissipation and diffusion terms.

# Introduction to turbulence modeling

- For example, the production term in the Reynolds stress equation it is written as follows,

$$-\tau_{ik} \frac{\partial \bar{u}_j}{\partial x_k} - \tau_{jk} \frac{\partial \bar{u}_i}{\partial x_k}$$

- And in the turbulent kinetic energy equation, the production term is written as follows,

$$\tau_{ij} \frac{\partial \bar{u}_i}{\partial x_j}$$

- Whereas, in the turbulent dissipation equation, the production term is written as follows,

$$C_{\epsilon 1} \frac{\epsilon}{k} \tau_{ij} \frac{\partial \bar{u}_i}{\partial x_j}$$

- And in the specific turbulent dissipation equation, the production term is written as follows,

$$\alpha \frac{\omega}{k} \tau_{ij} \frac{\partial \bar{u}_i}{\partial x_j}$$

# Introduction to turbulence modeling

Review of the most widely used turbulence models



# Introduction to turbulence modeling

The  $k - \epsilon$  turbulence model

# Introduction to turbulence modeling

- This is one of the most popular family of two-equation turbulence models.
- It is based on the Boussinesq hypothesis (Eddy viscosity models or EVM).
- The initial development of this model can be attributed to Chou [1], circa 1945.
- Launder and Spalding [2] and Launder and Sharma [3] further developed and calibrated the model and created what is generally referred to as the Standard  $k - \epsilon$  model.
- This is the model that we are going to address hereafter.
- There are many variations of this model.
- Each one designed to add new capabilities and overcome the limitations of the standard  $k - \epsilon$  model.
- The most notable limitation of the standard  $k - \epsilon$  model is that it requires the use of wall functions.
- Variants of this model include the RNG  $k - \epsilon$  model [3] and the Realizable  $k - \epsilon$  model [4], just to name a few.

## References:

- [1] P. Y. Chou. On Velocity Correlations and the Solutions of the Equations of Turbulent Fluctuation. Quarterly of Applied Mathematics. 1945.
- [2] B. E. Launder, D. B. Spalding. The Numerical Computation of Turbulent Flows. Computer Methods in Applied Mechanics and Engineering. 1974.
- [3] B. E. Launder, B. I. Sharma. Application of the Energy-Dissipation Model of Turbulence to the Calculation of Flow Near a Spinning Disc. Letters in Heat and Mass Transfer. 1974.
- [4] V. Yakhot, S. A. Orszag. Renormalization Group Analysis of Turbulence I Basic Theory. Journal of Scientific Computing. 1986.
- [5] T. Shih, W. Liou, A. Shabbir, Z. Yang, J. Zhu. A New - Eddy-Viscosity Model for High Reynolds Number Turbulent Flows - Model Development and Validation. Computers Fluids. 1995.

# Introduction to turbulence modeling

- It is called  $k - \epsilon$  because it solves two additional equations for modeling the turbulent viscosity, namely, the turbulent kinetic energy  $k$  and the turbulence dissipation rate  $\epsilon$ .

$$\nabla_t k + \nabla \cdot (\bar{\mathbf{u}} k) = \tau^R : \nabla \bar{\mathbf{u}} - \epsilon + \nabla \cdot \left[ \left( \nu + \frac{\nu_t}{\sigma_k} \right) \nabla k \right]$$

$$\nabla_t \epsilon + \nabla \cdot (\bar{\mathbf{u}} \epsilon) = C_{\epsilon_1} \frac{\epsilon}{k} \tau^R : \nabla \bar{\mathbf{u}} - C_{\epsilon_2} \frac{\epsilon^2}{k} + \nabla \cdot \left[ \left( \nu + \frac{\nu_t}{\sigma_\epsilon} \right) \nabla \epsilon \right]$$

- This model uses the following relation for the kinematic eddy viscosity,

$$\nu_t = \frac{C_\mu k^2}{\epsilon}$$

- With the following closure coefficients,

$$C_{\epsilon_1} = 1.44 \quad C_{\epsilon_2} = 1.92 \quad C_\mu = 0.09 \quad \sigma_k = 1.0 \quad \sigma_\epsilon = 1.3$$

- And auxiliary relationships,

$$\omega = \frac{\epsilon}{C_\mu k} \quad l = \frac{C_\mu k^{3/2}}{\epsilon}$$

# Introduction to turbulence modeling

- The solvable closure equations of the standard  $k - \epsilon$  turbulence model have been manipulated so there are no terms including fluctuating quantities (*i.e.*, velocity and pressure), and double or triple correlations of the fluctuating quantities.
- Remember, the Reynolds stress tensor is modeled using the Boussinesq approximation.
- The turbulence dissipation rate is modeled using a second transport equation.

$$\begin{aligned} \nabla_t k + \nabla \cdot (\bar{\mathbf{u}} k) &= \overbrace{\tau^R : \nabla \bar{\mathbf{u}}}^{\text{Production}} - \overbrace{\epsilon}^{\text{Dissipation}} + \nabla \cdot \left[ \left( \nu + \frac{\nu_t}{\sigma_k} \right) \nabla k \right] \\ \nabla_t \epsilon + \nabla \cdot (\bar{\mathbf{u}} \epsilon) &= \underbrace{C_{\epsilon_1} \frac{\epsilon}{k} \tau^R : \nabla \bar{\mathbf{u}}}_{\text{Production}} - \underbrace{C_{\epsilon_2} \frac{\epsilon^2}{k}}_{\text{Dissipation}} + \nabla \cdot \left[ \left( \nu + \frac{\nu_t}{\sigma_\epsilon} \right) \nabla \epsilon \right] \end{aligned}$$

**Note:**

$$\tau^R = -(\overline{\mathbf{u}'\mathbf{u}'})$$

# Introduction to turbulence modeling

- The standard  $k - \epsilon$  turbulence model use wall functions.
- The wall boundary conditions for the field variables are all taken care of by the wall functions implementation.
- In OpenFOAM, you need to give physically realistic values of boundary conditions at the walls.
- Using the standard walls functions approach developed by Launder and Spalding [1], the numerical values of the boundary conditions at the walls are computed as follows,

$$\frac{\partial k}{\partial n} = 0 \qquad \epsilon_P = \frac{C_\mu^{3/4} k_P^{3/2}}{\kappa y_P}$$

Where the subscript P means cell center

- The free-stream values can be computed in a similar way as for the  $k - \omega$  turbulence model.
- It is strongly recommended to not initialize these quantities with the same value or with values close to zero (in particular the turbulence dissipation rate).

# Introduction to turbulence modeling

Generalization of the  $k - \epsilon$  turbulence model and low-Reynolds formulation

# Introduction to turbulence modeling

- The equations of the  $k - \epsilon$  turbulence model can be generalized as follows,

$$\nabla_t k + \nabla \cdot (\bar{\mathbf{u}} k) = \tau^R : \nabla \bar{\mathbf{u}} - \epsilon + \nabla \cdot \left[ \left( \nu + \frac{\nu_t}{\sigma_k} \right) \nabla k \right] + L_k$$

$$\nabla_t \epsilon + \nabla \cdot (\bar{\mathbf{u}} \epsilon) = C_{\epsilon_1} f_1 \frac{\epsilon}{k} \tau^R : \nabla \bar{\mathbf{u}} - C_{\epsilon_2} f_2 \frac{\epsilon^2}{k} + \nabla \cdot \left[ \left( \nu + \frac{\nu_t}{\sigma_\epsilon} \right) \nabla \epsilon \right] + L_\epsilon$$

$$\nu_t = f_\mu C_\mu \frac{k^2}{\epsilon}$$

**Note:**

$$\tau^R = -(\overline{\mathbf{u}' \mathbf{u}'})$$

- At this point, by changing the values of the coefficients  $f_1, f_2, f_\mu, L_k$ , and  $L_\epsilon$  we can obtain different formulations of the  $k - \epsilon$  turbulence model.
- For example, by setting the coefficients  $f_1, f_2, f_\mu$  to one and the coefficients  $L_k$  and  $L_\epsilon$  to zero, we recast the standard  $k - \epsilon$  turbulence model.

# Introduction to turbulence modeling

- One of the drawbacks of the standard  $k - \epsilon$  model is that it can only be used with wall functions.
  - Actually, and without any modification to the standard  $k - \epsilon$  turbulence model, the equations can be integrated in the viscous sublayer all the way down to wall, but the results will deteriorate.
- The standard  $k - \epsilon$  turbulence model is a wall modeling model.
- This also applies to the RNG  $k - \epsilon$  [1, 2] and realizable  $k - \epsilon$  [3] turbulence models, as both models are variants of the standard  $k - \epsilon$  turbulence model.
- To avoid this limitation, we can integrate the governing equations in the viscous sublayer all the way to the wall by adding a few modifications to the original formulation.
- The resulting formulations are known as low-Reynolds number  $k - \epsilon$  turbulence models.
- The terminology low-Reynolds number refers to the Reynolds number measured normal to the wall (something similar to the  $y^+$  normal to the wall) and not the system Reynolds number.



# Introduction to turbulence modeling

- The first low-Reynolds number  $k - \epsilon$  turbulence model was developed by Jones and Launder [1, 2], and subsequently it has been modified by several authors.
- The primary modifications introduced by Jones and Launder [1, 2] were to include turbulence Reynolds number dependency functions  $f_1, f_2$ , and  $f_\mu$ .
- The purpose of these functions is to correct or damp the behavior of the turbulent viscosity as we approach to the walls.
- The main idea is getting asymptotically consistent near wall behavior.
- Furthermore, additional terms  $L_k$  and  $L_\epsilon$  were added to the equations to account for the dissipation which may not be isotropic.
- Recall that the turbulence Reynolds number is related to the Reynolds number of the integral scales and can be computed as follows,

$$Re_T = \frac{k^{1/2} l_0}{\nu} = \frac{k^2}{\epsilon \nu} \approx \frac{\nu_t}{\nu}$$

[1] W. Jones, B. Launder. The prediction of laminarization with a two-equation model of turbulence. Int. J. Heat Mass Transfer, vol. 15, pp. 301–314, 1972.

[2] W. Jones, B. Launder. The calculation of low-Reynolds number phenomena with a two-equation model of turbulence. Int. J. Heat Mass Transfer, vol. 16, pp. 1119–1130, 1973.

# Introduction to turbulence modeling

- Closure functions and coefficients for the  $k - \epsilon$  turbulence models.

Model	$f_1$	$f_2$	$f_\mu$	$L_k$	$L_\epsilon$
Standard	1.0	1.0	1.0	0	0
Jones-Launder [1,2]	1.0	$1 - 0.3\exp(-Re_T^2)$	$\exp\left[\frac{-2.5}{1 + 0.02Re_T}\right]$	$-2\nu\left(\frac{\partial k^{1/2}}{\partial x_j}\right)^2$	$2\nu\nu_t\left(\frac{\partial^2 u_i}{\partial x_j \partial x_k}\right)^2$
Launder-Sharma [3]	1.0	$1 - 0.3\exp(-Re_T^2)$	$\exp\left[\frac{-3.4}{(1 + 0.02Re_T)^2}\right]$	$-2\nu\left(\frac{\partial k^{1/2}}{\partial x_j}\right)^2$	$2\nu\nu_t\left(\frac{\partial^2 u_i}{\partial x_j \partial x_k}\right)^2$
Hoffman [4]	1.0	$1 - 0.3\exp(-Re_T^2)$	$\exp\left[\frac{-1.75}{1 + 0.02Re_T}\right]$	$-\frac{\nu}{y}\frac{\partial k}{\partial y}$	0
Nagano-Hishida [5]	1.0	$1 - 0.3\exp(-Re_T^2)$	$[1 - \exp(-Re_T/26.5)]^2$	$-2\nu\left(\frac{\partial k^{1/2}}{\partial y}\right)^2$	$\nu\nu_t(1 - f_\mu)\left(\frac{\partial^2 u}{\partial y^2}\right)^2$
Chien [6]	1.0	$1 - \frac{0.4}{1.8}\exp(-Re_T^2/36)$	$1 - \exp(-0.0115d^+)$ $d^+ = \rho u_\tau / \mu$	$-2\frac{\mu k}{\rho d^2}$	$-2\frac{\mu \epsilon}{\rho d^2}\exp(-d^+/2)$

## References:

[1] W. Jones, B. Launder. The prediction of laminarization with a two-equation model of turbulence. Int. J. Heat Mass Transfer, vol. 15, pp. 301–314, 1972.

[2] W. Jones, B. Launder. The calculation of low-Reynolds number phenomena with a two-equation model of turbulence. Int. J. Heat Mass Transfer, vol. 16, pp. 1119–1130, 1973.

[3] B. Launder, B. Sharma. Application of the energy dissipation model of turbulence to the calculation of flow near a spinning disc. Letters in Heat and Mass Transfer, Vol. 1(2), pp. 131-138. 1974.

[4] G. Hoffman. Improved form of the low Reynolds number k-epsilon turbulence model. Physics of Fluids, vol. 18(3), pp. 309-312, 1975.

[5] Y. Nagado, M. Hishida. Improved form of the k-epsilon model for wall turbulent shear flows. Journal of Fluids Engineering, vol. 109, pp. 156-160, 1987.

[6] K. Chien. Predictions of Channel and Boundary-Layer Flows with a Low-Reynolds-Number Turbulence Model. AIAA Journal, vol. 20(1), pp. 33-38, 1982.

# Introduction to turbulence modeling

- Closure functions and coefficients for the  $k - \epsilon$  turbulence models.

Model	$C_\mu$	$C_{\epsilon 1}$	$C_{\epsilon 2}$	$\sigma_k$	$\sigma_\epsilon$
Standard	0.09	1.44	1.92	1.0	1.3
Jones-Launder [1,2]	0.09	1.44	1.92	1.0	1.3
Launder-Sharma [3]	0.09	1.44	1.92	1.0	1.3
Hoffman [4]	0.09	1.81	2.0	2.0	3.0
Nagano-Hishida [5]	0.09	1.45	1.9	1.0	1.3
Chien [6]	0.09	1.35	1.8	1.0	1.3

**References:**

[1] W. Jones, B. Launder. The prediction of laminarization with a two-equation model of turbulence. Int. J. Heat Mass Transfer, vol. 15, pp. 301–314, 1972.

[2] W. Jones, B. Launder. The calculation of low-Reynolds number phenomena with a two-equation model of turbulence. Int. J. Heat Mass Transfer, vol. 16, pp. 1119–1130, 1973.

[3] B. Launder, B. Sharma. Application of the energy dissipation model of turbulence to the calculation of flow near a spinning disc. Letters in Heat and Mass Transfer, Vol. 1(2), pp. 131-138. 1974.

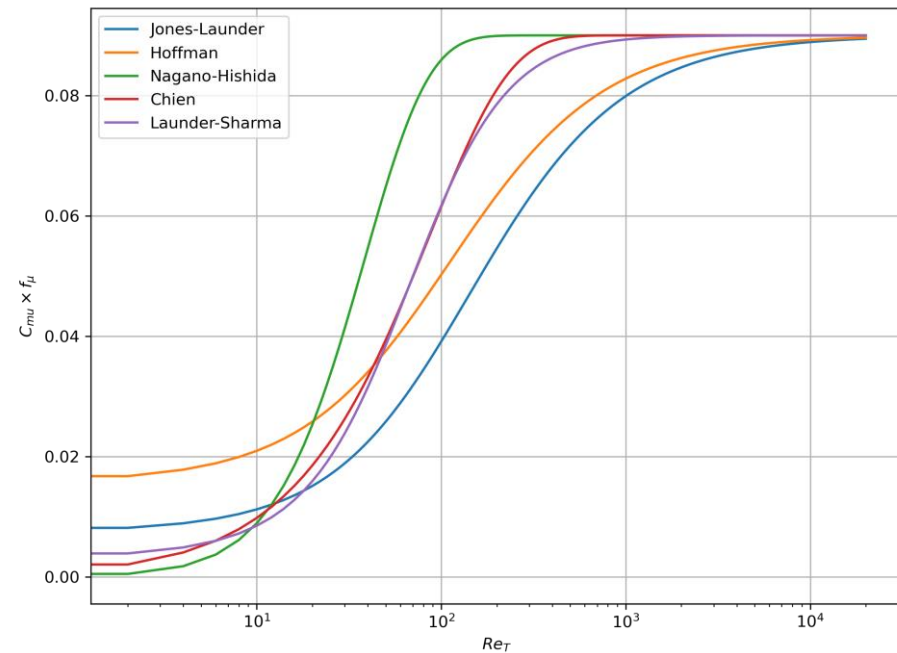
[4] G. Hoffman. Improved form of the low Reynolds number k-epsilon turbulence model. Physics of Fluids, vol. 18(3), pp. 309-312,1975.

[5] Y. Nagado, M. Hishida. Improved form of the k-epsilon model for wall turbulent shear flows. Journal of Fluids Engineering, vol. 109, pp. 156-160, 1987.

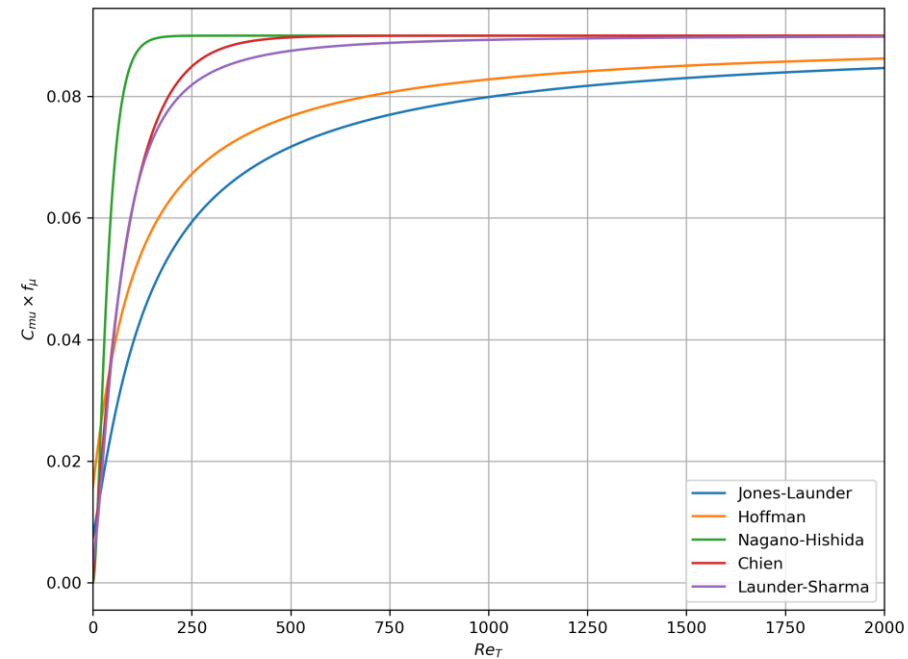
[6] K. Chien. Predictions of Channel and Boundary-Layer Flows with a Low-Reynolds-Number Turbulence Model. AIAA Journal, vol. 20(1), pp. 33-38, 1982.

# Introduction to turbulence modeling

- Closure coefficient  $C_\mu$  as a function of  $Re_T$ .
- These plots illustrate the damping effect towards the walls of the function  $f_\mu$ .



Semi-logarithmic scale



Linear scale

# Introduction to turbulence modeling

The Spalart-Allmaras turbulence model

# Introduction to turbulence modeling

- The Spalart-Allmaras model [1,2] is a one-equation model that solves a model transport equation for the modified turbulent kinematic viscosity.
- It is based on the Boussinesq hypothesis (Eddy viscosity models or EVM).
- By far this is the most popular and successful one-equation model.
- It also has been adopted as the foundation for DES models [3].
- The Spalart-Allmaras model was designed specifically for aerospace applications involving wall-bounded flows.
- In its original form, the Spalart-Allmaras model is a wall resolving method, requiring the use of fine meshes in order to resolve the viscous sublayer.
- Over the years this method has been improved. Each variation is designed to add new capabilities and overcome the limitations of the predecessor formulations.
- The most notable drawback is its limitation to deal with massive flow separation.
- Variants of this model include the addition of rotation/curvature corrections, trip terms, production limiters, strain adaptive formulations, wall roughness corrections, compressibility corrections, extension to  $y^+$  insensitive treatment, and so on.
- Hereafter, we will address the model formulation described in reference [1] (which is probably the original formulation).

[1] P. Spalart, S. Allmaras, A One-Equation Turbulence Model for Aerodynamic Flows, Recherche Aerospatiale, No. 1, pp. 5-21, 1994.

[2] P. Spalart, S. Allmaras. A One-Equation Turbulence Model for Aerodynamic Flows. AIAA Conference Paper AIAA-92-0439, 1992.

[3] M. Shur, P. R. Spalart, M. Strelets, A. Travin. Detached-Eddy Simulation of an Airfoil at High Angle of Attack. 1999.

# Introduction to turbulence modeling

- The Spalart-Allmaras model is based on the Boussinesq hypothesis (EVM).

$$\tau_{ij}^R = -\rho \overline{u'_i u'_j} = 2\mu_t S_{ij} - \frac{2}{3}\rho k \delta_{ij}$$

- Where the circled term is generally ignored because information about the turbulent kinetic energy  $k$  is not readily available.
- Details regarding a nonlinear implementation that also includes an approximation for the term,

$$-\frac{2}{3}\rho k \delta_{ij}$$

can be found in references [1, 2, 3].

- Most of the one equation turbulence models (unless they are based on a transport equation for the turbulent variable  $k$ ), do not provide information about the turbulent kinetic energy.

[1] M. Mani, D. Babcock, C. Winkler, P. Spalart, Predictions of a Supersonic Turbulent Flow in a Square Duct, AIAA Paper 2013-0860, January 2013.

[2] C. Rumsey, H. Lee, T. Pulliam, Reynolds-Averaged Navier-Stokes Computations of the NASA Juncture Flow Model Using FUN3D and OVERFLOW, AIAA Paper 2020-1304, 2020.

[3] C. Rumsey, J. Carlson, T. Pulliam, P. Spalart, Improvements to the Quadratic Constitutive Relation Based on NASA Juncture Flow Data, AIAA Journal, Vol. 58, No. 10, pp. 4374-4384, 2020.

# Introduction to turbulence modeling

- In the Spalart-Allmaras model (SA), a closed equation for the turbulent eddy viscosity is artificially created that fits well a range of experimental and empirical data.
- To accomplish this, the SA  $\nu_T$  equation is built up term by term in a series of calibrations involving flows of increasing complexity.

$$\underbrace{\nabla_t \nu_T}_{\text{Transient term}} + \underbrace{\nabla \cdot \bar{\mathbf{u}} \nu_T}_{\text{Convection}} = \underbrace{P^{\nu_T}}_{\text{Production}} + \underbrace{\epsilon^{\nu_T}}_{\text{Dissipation}} + \underbrace{D^{\nu_T}}_{\text{Diffusion}} + \underbrace{S^{\nu_T}}_{\text{Source terms}}$$

- The resulting model has gone through a number of developmental iterations beyond its original form and has been widely tested for different external aerodynamics applications.
- Probably, this is the turbulence model that has undergone more modifications.
- It is beyond the scope of this discussion to delve the different calibration steps of each term and the choice of the closure coefficients.
- The interested reader should take a look at the following link:
  - <https://turbmodels.larc.nasa.gov/spalart.html>



# Introduction to turbulence modeling

- Before introducing the SA model, it is worth mentioning that this model does not actually solve a transport equation for the turbulent eddy viscosity  $\nu_T$ .
- It solves a modified version of the turbulent eddy viscosity.
- Namely, modified eddy viscosity  $\tilde{\nu}$ .
- Therefore, we deal with the following general transport equation,

$$\underbrace{\nabla_t \tilde{\nu}}_{\text{Transient term}} + \underbrace{\nabla \cdot \bar{\mathbf{u}} \tilde{\nu}}_{\text{Convection}} = \underbrace{P^{\tilde{\nu}}}_{\text{Production}} + \underbrace{\epsilon^{\tilde{\nu}}}_{\text{Dissipation}} + \underbrace{D^{\tilde{\nu}}}_{\text{Diffusion}} + \underbrace{S^{\tilde{\nu}}}_{\text{Source terms}}$$

- Also, as it is an artificial method, the structure of the production and dissipation terms is slightly different from that of the two and more equations models.

# Introduction to turbulence modeling

- The closure equations of the standard SA model are as follows,

$$\begin{aligned} \frac{\partial \tilde{\nu}}{\partial t} + \bar{u}_j \frac{\partial \tilde{\nu}}{\partial x_j} = & c_{b1} (1 - f_{t2}) \tilde{S} \tilde{\nu} - \left[ c_{w1} f_w - \frac{c_{b1}}{\kappa^2} f_{t2} \right] \left( \frac{\tilde{\nu}}{d} \right)^2 \\ & + \frac{1}{\sigma} \frac{\partial}{\partial x_j} \left[ (\nu + \tilde{\nu}) \frac{\partial \tilde{\nu}}{\partial x_j} \right] + \frac{c_{b2}}{\sigma} \frac{\partial \tilde{\nu}}{\partial x_i} \frac{\partial \tilde{\nu}}{\partial x_i} \end{aligned}$$

- Where  $\tilde{\nu}$  is the modified eddy viscosity.
- This model uses the following relation for the kinematic eddy viscosity,

$$\nu_T = \tilde{\nu} f_{v1}$$

- Where  $f_{v1}$  can be interpreted as a wall damping function [1].

[1] G. Mellor, H. Herring. Two methods of calculating turbulent boundary layer behavior based on numerical solutions of the equations of motion. 1968.

# Introduction to turbulence modeling

- With the following closure relationships,

$$f_{v1} = \frac{\chi^3}{\chi^3 + c_{v1}^3}$$

$$\chi = \frac{\tilde{\nu}}{\nu}$$

$$f_{t2} = c_{t3} e^{-c_{t4} \chi^2}$$

$$\tilde{S} = \Omega + \frac{\tilde{\nu}}{\kappa^2 d^2} f_{v2}$$

$$f_{v2} = 1 - \frac{\chi}{1 + \chi f_{v1}}$$

d is the minimum distance to the nearest wall

$$\Omega = \sqrt{2W_{ij}W_{ij}}$$

Magnitude of the vorticity tensor

$$W_{ij} = \frac{1}{2} \left( \frac{\partial \bar{u}_i}{\partial x_j} - \frac{\partial \bar{u}_j}{\partial x_i} \right)$$

Anti-symmetric part of the velocity gradient (vorticity tensor)

$$f_w = g \left[ \frac{1 + c_{w3}^6}{g^6 + c_{w3}^6} \right]^{1/6}$$

$$g = r + c_{w2} (r^6 - r)$$

$$r = \min \left[ \frac{\tilde{\nu}}{\tilde{S} \kappa^2 d^2}, 10 \right]$$

# Introduction to turbulence modeling

- And with the following closure coefficients,

$$c_{b1} = 0.1355 \quad c_{b2} = 0.622$$

$$c_{v1} = 7.1 \quad \sigma = 2/3$$

$$c_{w1} = \frac{c_{b1}}{\kappa^2} + \frac{(1 + c_{b2})}{\sigma} \quad c_{w2} = 0.3$$

$$c_{w3} = 2 \quad \kappa = 0.41$$

$$c_{t3} = 1.2 \quad c_{t4} = 0.5$$

# Introduction to turbulence modeling

- Remember, the standard SA model is wall resolving.
- The wall boundary conditions for the turbulent variables can be computed as follows,

$$\tilde{\nu} = 0 \quad \nu_t = 0$$

- The freestream conditions can be computed using the following relationships,

$$\tilde{\nu}_{\text{farfield}} = 3\nu_{\infty} \quad \text{to} \quad 5\nu_{\infty} \quad \nu_{t_{\text{farfield}}} = 0.210438\nu_{\infty} \quad \text{to} \quad 1.29423\nu_{\infty}$$

- If information related to the turbulent kinetic energy and turbulent dissipation is available, the freestream conditions can be computed as follows,

$$\tilde{\nu} = \nu_t = \frac{C_{\mu} k^2}{\epsilon}$$

$$\tilde{\nu} = \nu_t = \frac{k}{\omega}$$

# Introduction to turbulence modeling

The  $k - \omega$  SST turbulence model

# Introduction to turbulence modeling

- Let us address the Menter 2003  $k - \omega$  SST, also known as  $k - \omega$  SST [1].
- Of all Eddy viscosity models (EVM), this is probably the most successful one.
- This model is  $y^+$  insensitive.
- This model features several closure coefficients, blending functions, and auxiliary relations.
- Many of the coefficients used in this model are computed by a blend function between the respective constants of the  $k - \epsilon$  and  $k - \omega$  models.
- The  $k - \omega$  SST model (and its variants), is considered the most efficient and general RANS/URANS turbulence model.
- Therefore, it is strongly recommended to use this model.
- During the guided tutorials, you will see that among the EVM this is the only model that consistently gives good/acceptable results for all cases that we are going to address.

# Introduction to turbulence modeling

- The solvable equations of the  $k - \omega$  SST turbulence model are the following ones,

$$\nabla_t k + \nabla \cdot (\bar{\mathbf{u}} k) = \tau^R : \nabla \bar{\mathbf{u}} - \beta^* k \omega + \nabla \cdot \left[ \left( \nu + \frac{\nu_t}{\sigma_k} \right) \nabla k \right]$$

$$\frac{\partial \omega}{\partial t} + \nabla \cdot (\mathbf{u} \omega) = \frac{\alpha}{\nu_t} \tau^R : \nabla \bar{\mathbf{u}} - \beta \omega^2 + \nabla \cdot \left[ \left( \nu + \frac{\nu_t}{\sigma_\omega} \right) \nabla \omega \right] + 2(1 - F_1) \frac{\sigma_\omega}{\omega} \nabla k \cdot \nabla \omega$$

$$\mu_t = \rho \frac{a_1 k}{\max(a_1 \omega, F_2 \Omega)}$$

**Note:**

$$\tau^R = -(\overline{\mathbf{u}' \mathbf{u}'})$$

$$\Omega = \sqrt{2 W_{ij} W_{ij}}$$

Magnitude of the vorticity tensor

$$W_{ij} = \frac{1}{2} \left( \frac{\partial \bar{u}_i}{\partial x_j} - \frac{\partial \bar{u}_j}{\partial x_i} \right)$$

Anti-symmetric part of the velocity gradient (vorticity tensor)



# Introduction to turbulence modeling

- Letting  $\phi$  denote any one of the parameters  $\alpha, \sigma_k, \sigma_\omega, \beta, \beta^*, \mu_t$ , then each of these parameters varies between a near-wall state  $\phi_1$  and a far-wall state  $\phi_2$  according to,

$$\phi = F_1 \phi_1 + (1 - F_1) \phi_2$$

- Where  $F_1$  and  $\arg_1$  are defined as follows,

$$F_1 = \tanh(\arg_1^4) \quad \arg_1 = \min \left[ \max \left( \frac{\sqrt{k}}{\beta^* \omega d}, \frac{500\nu}{d^2 \omega} \right), \frac{4\rho \sigma_\omega k}{C D_{k\omega} d^2} \right]$$

- Notice that  $0 \leq F_1 \leq 1$ , and  $\arg_1 \geq 0$ .
- In the equations,  $d$  is the distance from the wall. So, as  $d$  increases, the two expressions in the maximum of  $\arg_1$  become smaller as well the term that the maximum is being compared to.
- Thus,  $\arg_1$  diminishes with  $d$ , causing  $F_1$  to approach zero and  $\phi$  to approach the far-field value  $\phi_2$  (the coefficients of the  $k - \epsilon$  turbulence model).
- The opposite behavior occurs as the wall is approached with,

$$\arg_1 \rightarrow \infty, \quad F_1 \rightarrow 1, \quad \phi \rightarrow \phi_1$$

- That is, we obtain the coefficients of the  $k - \omega$  turbulence model.

# Introduction to turbulence modeling

- The additional expressions and coefficients used in the formulation are,


$$CD_{k\omega} = \max \left( 2\rho\sigma_{\omega 2} \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}, 10^{-20} \right)$$

$$F_2 = \tanh(\arg_2^2) \qquad \arg_2 = \max \left( \frac{2\sqrt{k}}{\beta^* \omega d}, \frac{500\nu}{d^2 \omega} \right)$$

$$\alpha_1 = \frac{\beta_1}{\beta^*} - \frac{\sigma_{\omega 1} \kappa^2}{\sqrt{\beta^*}} \qquad \alpha_2 = \frac{\beta_2}{\beta^*} - \frac{\sigma_{\omega 2} \kappa^2}{\sqrt{\beta^*}}$$

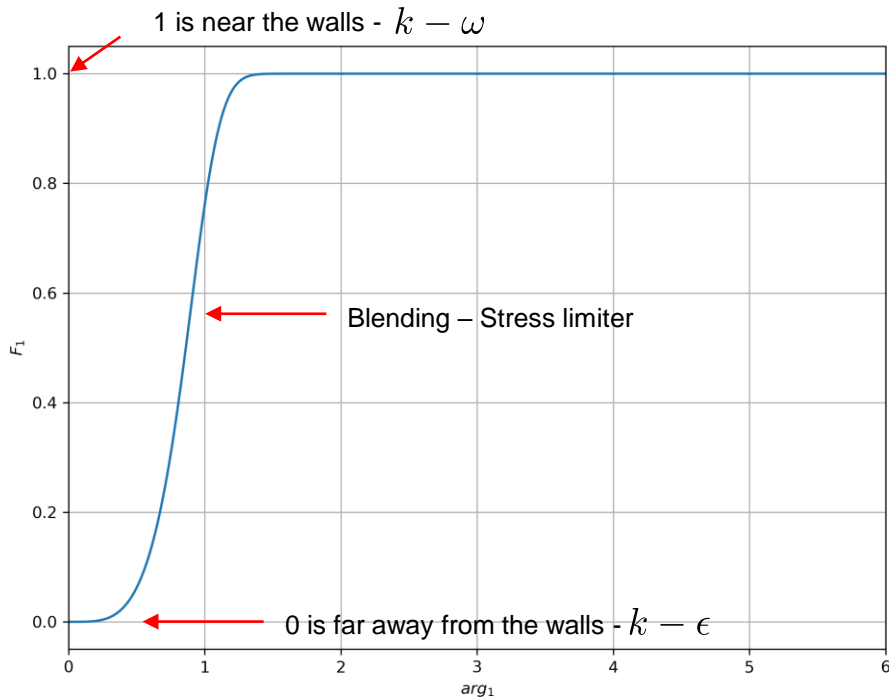
$$\sigma_{k1} = 0.85, \quad \sigma_{k2} = 1.0, \quad \sigma_{\omega 1} = 0.5, \quad \sigma_{\omega 2} = 0.856,$$

$$\beta_1 = 0.075, \quad \beta_2 = 0.0828, \quad \beta^* = 0.09, \quad \kappa = 0.41, \quad a_1 = 0.31$$

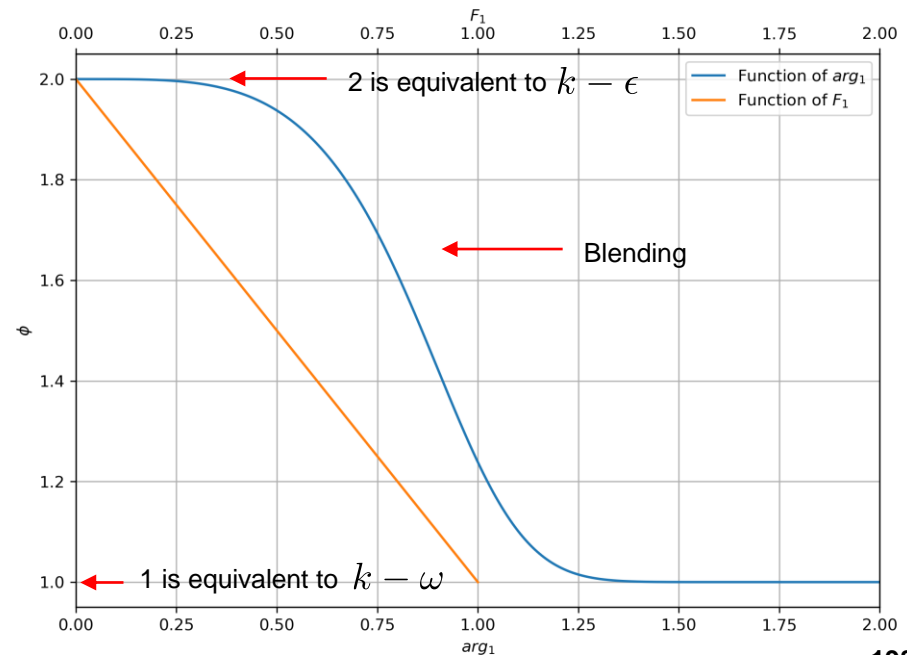

$$\approx \sqrt{C_\mu} \quad \text{or} \quad \approx \sqrt{\beta^*}$$

# Introduction to turbulence modeling

- Plot of the function  $F_1$  as a function of  $arg_1$ .
- When  $F_1 = 0$  the formulation is in the far-field conditions. That is, the formulation uses the  $k - \epsilon$  model.
- And when  $F_1 = 1$  the formulation is in the near wall region. That is, the formulation uses the  $k - \omega$  model.
- The function  $F_1$  represents a blending between the two turbulence models.



- Plot of any of the parameters  $\phi$  of the turbulence model as a function of  $F_1$ .
- When  $F_1 = 0$  we use the parameters of the  $k - \epsilon$  model.
- This function indicates how the parameters vary between a near-wall state and a far-wall state.
- The function  $\phi$  blends the parameters of the two turbulence models.



# Introduction to turbulence modeling

- It is worth noting that the SST model conforms to the  $k - \epsilon$  model away from walls.
- And close to the walls it uses a  $y^+$  insensitive formulation.
- The SST model and the Wilcox  $k - \omega$  model share many common parameters.
- By using the following relationship,

$$\epsilon = \beta^* \omega k$$

- And by taking its substantial derivative, and after substituting for the  $k$  and  $\omega$  derivatives using the solvable equations of the SST models, gives a differential equation for  $\epsilon$  of essentially the same form as the solvable equation of the dissipation rate  $\epsilon$  of the  $k - \epsilon$  turbulence model.
- Substituting the far-field form of the constants into the expression yields an equation for the turbulent dissipation rate  $\epsilon$  that conforms to that in the  $k - \epsilon$  closure with only small differences.

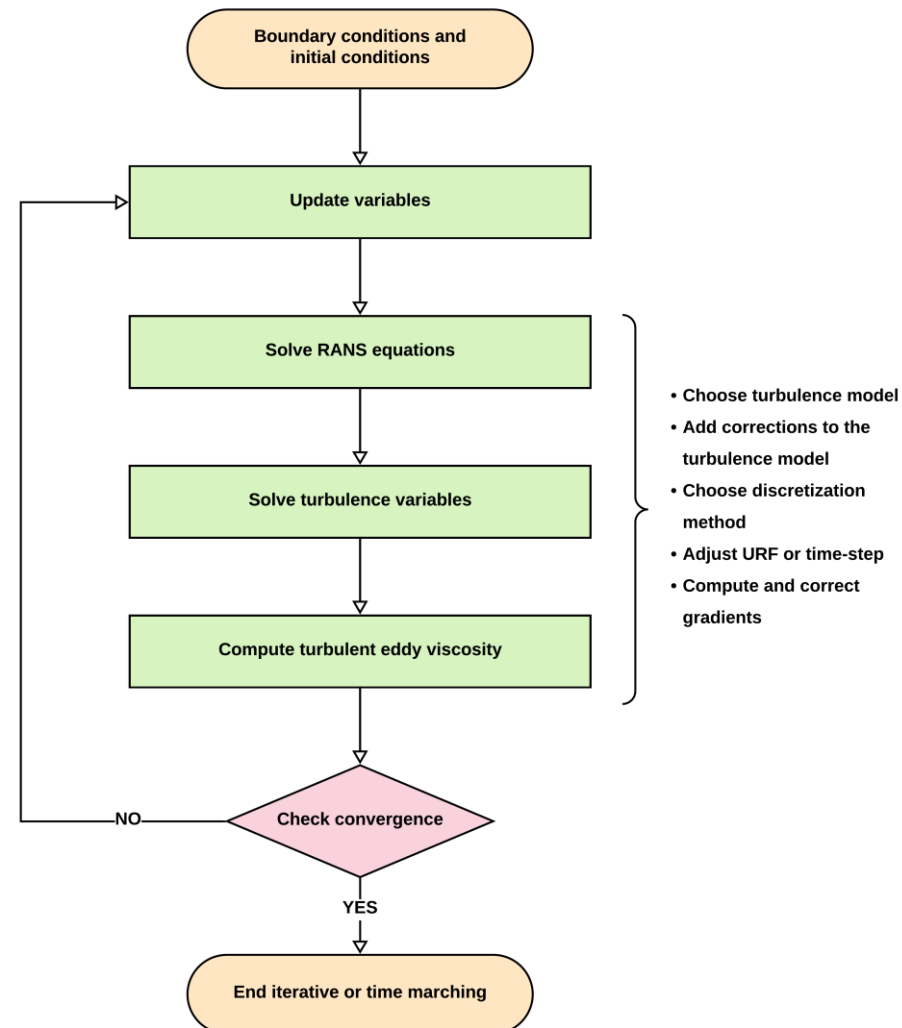
# Introduction to turbulence modeling

End of review of turbulence models

# Introduction to turbulence modeling

## A naïve CFD loop for turbulence modeling

- The first step consist in defining the boundary and initial conditions of all variables. Including the variables related to the turbulence model.
- The loop will solve first the RANS/URANS equations.
- Then, it will move to the next step, where it solves the equations of the turbulence model using the mean values of the RANS/URANS equations.
- It will then compute the turbulent eddy viscosity.
- At this point, it will check the convergence.
- If it is necessary, it will update the variables and it will do another sweep.
- To solve this problem, you can use pressure-based or density-based methods.
- Generally speaking, it is better to use pressure-based methods.
- If you are conducting RANS simulations (steady simulations) we recommend you use coupled methods.
- If coupled methods are not available, you can use any segregated method (SIMPLE or SIMPLEC).
- Instead, if you are conducting URANS simulations (unsteady simulations), we recommend you use the PISO method or the fractional step method.



# Introduction to turbulence modeling

## Final summary

- By using the Reynolds decomposition and time-averaging the exact Navier-Stokes equations, we obtain the RANS/URANS equations.
- The Reynolds stress tensor  $\tau^R$  appearing in the RANS/URANS equations needs to be modeled.
  - The most widely used approach is the Boussinesq approximation.
- From the Boussinesq approximation a new variable emerges, namely, the turbulent viscosity  $\mu_t$ .
- To compute the turbulent viscosity  $\mu_t$ , we need to use additional closure equations.
- All equations used must be expressed in terms of mean flow quantities.
  - That is, we need to remove the instantaneous fluctuations from the equations by using proper engineering approximations.
- This is our closure problem.
- The derivation of the equation for the turbulent viscosity is based on dimensional analysis.
- Which does not tell much about the underlying physics of the relationships used, so we need to be very critical when using turbulence models.

# Introduction to turbulence modeling

## Final remarks on RANS turbulence models

- In terms of meshing requirements, RANS simulations are not computationally 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 placing the first cell center (or first cell grid point) in the buffer layer  $10 < y^+ < 30$ .
- Use RANS simulations as starting point for LES/DES (which we will address in the following slides).



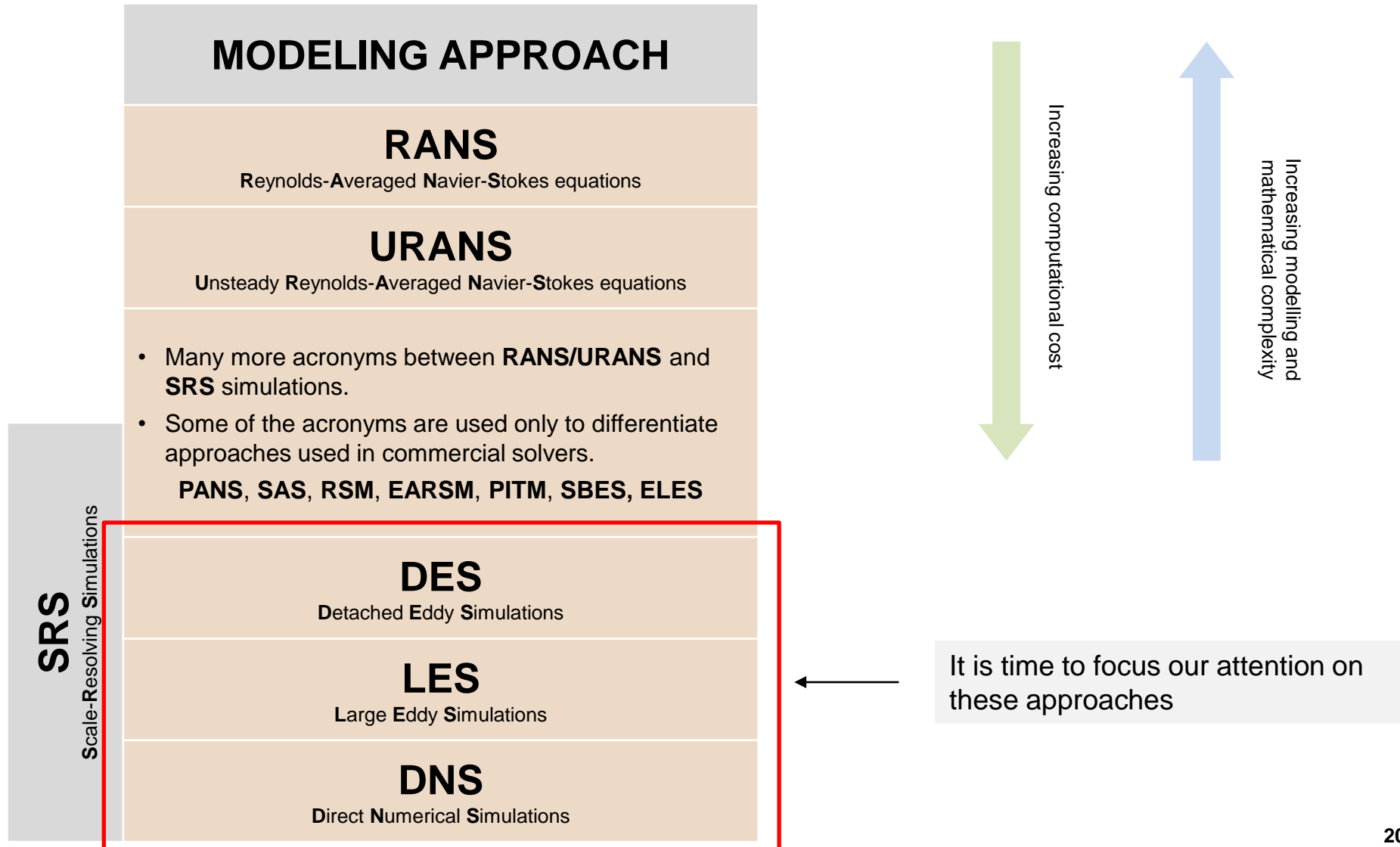
## **Guided tutorial 1 Laminar-Turbulent pipe**

## **Guided tutorial 2 Laminar-Turbulent flat plate**

## **Guided tutorial 3 Transitional flat plate**

# Introduction to turbulence modeling

- Overview of the main turbulence modeling approaches.

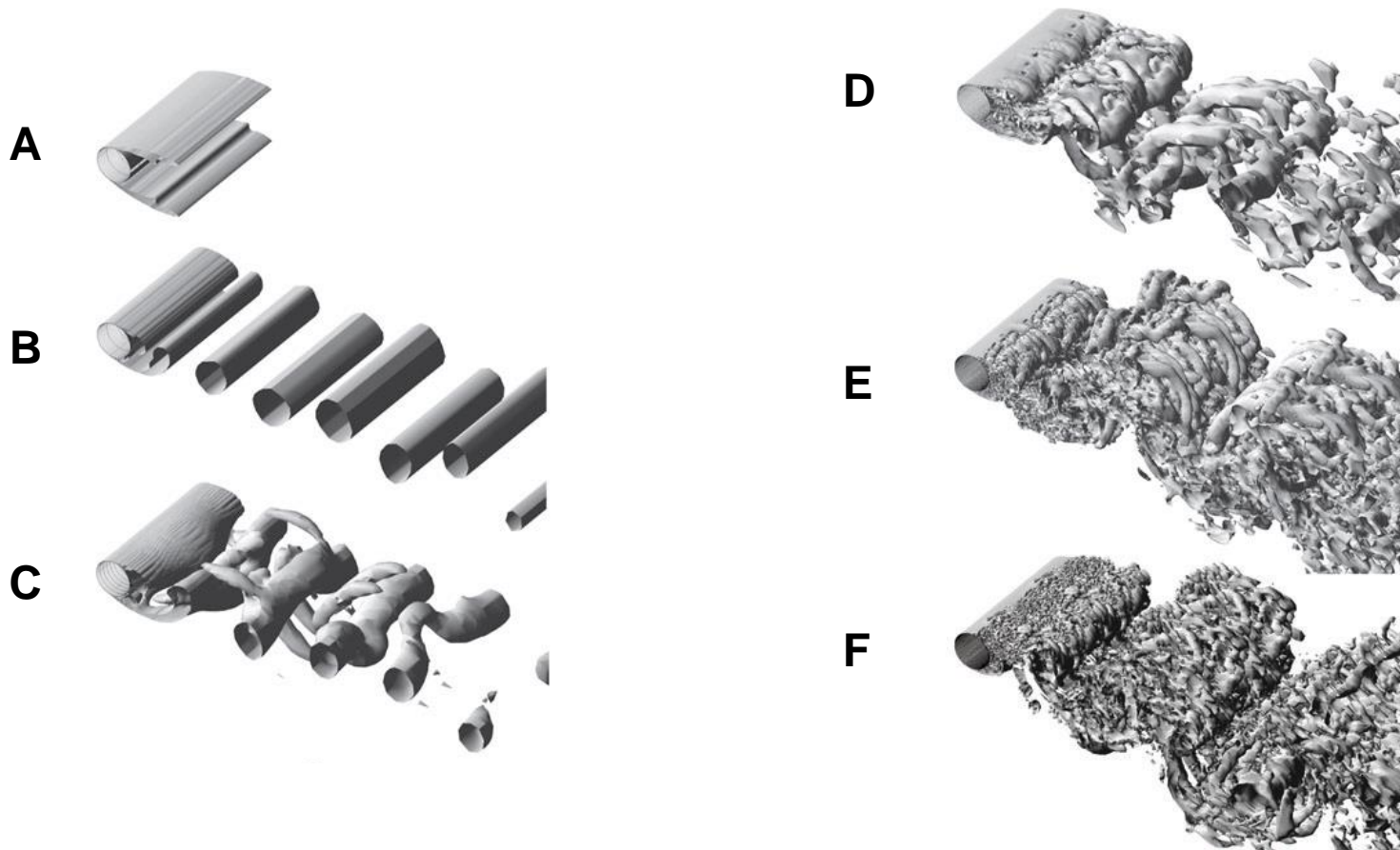


# Introduction to turbulence modeling

- Scale-resolving simulations or SRS, aim at resolving all or most of the scales in space and time.
- SRS simulations are intrinsically three-dimensional and unsteady.
  - Therefore, the meshing requirements are much larger than those for RANS/URANS simulations.
  - Also, the time-step requirements are more stringent.
- SRS simulations requires good meshes, accurate and robust numerical schemes, and CFL number requirements less than 1 for good accuracy.
  - We must minimize the numerical diffusion due to mesh resolution and numerical discretization.
- Sometimes, the two-dimensional approximation can be taken in SRS, but do it with a lot of care.
- The most well-known SRS approaches are LES (large eddy simulations) and DES (detached eddy simulations).
- DNS simulations also belong to the group of SRS methods.
- DNS simulations do not use any models.
  - They directly resolve all scales in space and time.

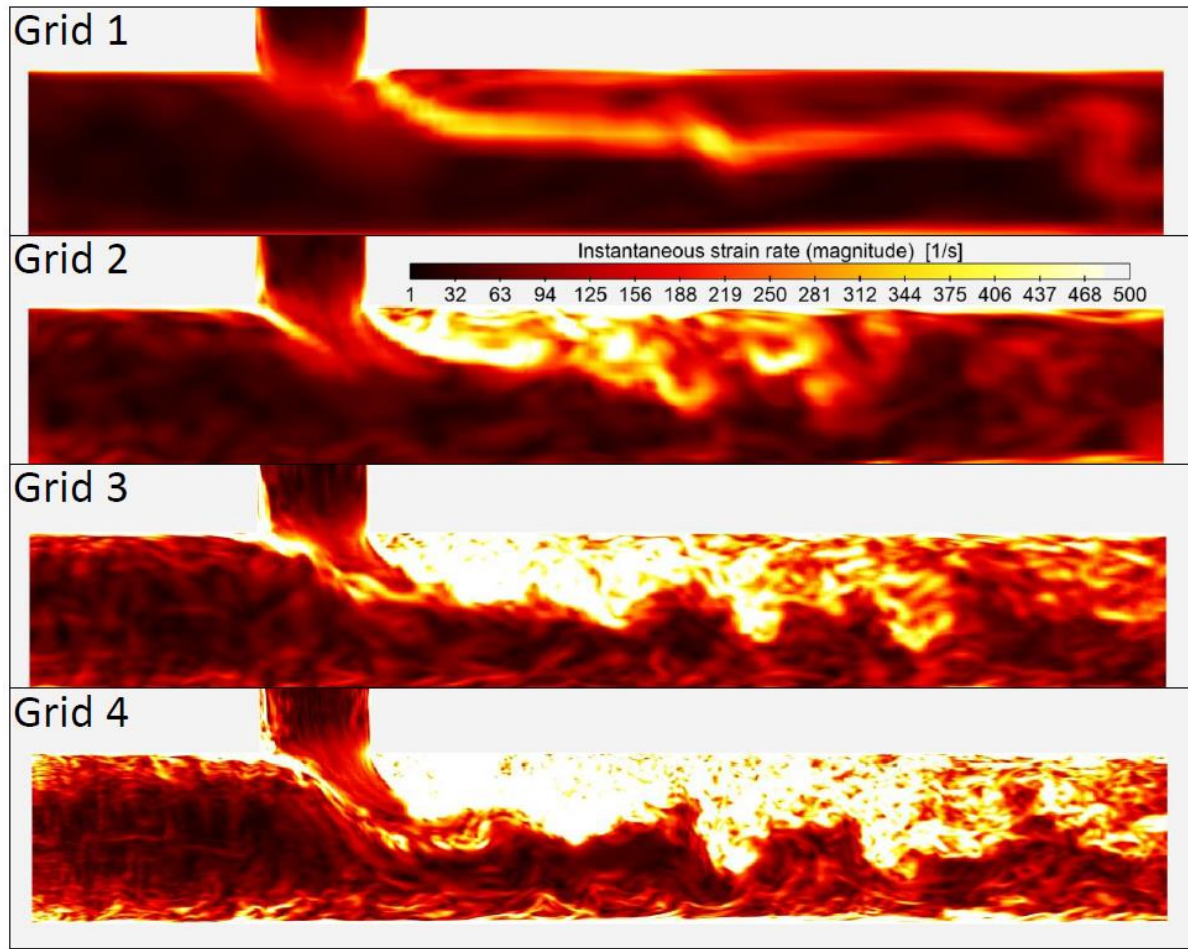
# Introduction to turbulence modeling

- Simulation results for flow over a cylinder (vorticity isosurfaces).
- Turbulence is modeled as (a) shear-stress-transport (SST) steady RANS. (b) 2D SST URANS. (c) 3D SST URANS. Spalart-Allmaras DES on a (d) coarse grid and a (e) fine grid/ (f) SST DES on a fine grid [1].



# Introduction to turbulence modeling

- Influence of mesh density on the turbulent flow structures in SRS LES simulations [1].



[1] J. Lavedrine. Impact of grid refinement on SRS LES calculation. Version 1.1. Ansys France, 08/2009.

# Introduction to turbulence modeling

## Filtered Navier-Stokes equations

- As for RANS, in LES simulations the smallest scales need to be modeled.
- Therefore, we need to somehow remove or filter the smallest scales of the instantaneous field variables.
- In LES, we use a similar approach to the Reynolds decomposition, but we call it LES decomposition (or filtering decomposition),

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

- Where the tilde represents the filtered quantity.
- In the literature, most of the time the tilde is used to denote the filtered quantity.
- Hereafter, and in the context of LES/DES simulations, we will interchange the tilde and overbar symbols.
- Also, the following averaging rules are different from RANS,

$$\tilde{\tilde{\phi}} \neq \tilde{\phi} \qquad \tilde{\phi}' \neq 0 \qquad \tilde{\phi\psi} \neq \tilde{\phi}\tilde{\psi}$$

# Introduction to turbulence modeling

## Filtered Navier-Stokes equations

- In LES, to remove the small scales we use a filter in space.
- You can see this operation as a rolling average.
- This filtered quantity represents the resolved scales (grid scales).
- Whereas, in RANS/URANS we use a time averaging to remove the fluctuations.
- This is the main difference between the techniques used to remove the fluctuations of small scales.
- By the way, these two concepts (filtering and time average) appear to be similar, but they are not. In fact, the rules used are different.
- At the end of the day, the instantaneous field can be decomposed as follows (LES decomposition),

The diagram illustrates the LES decomposition of an instantaneous field  $\phi(\mathbf{x}, t)$ . It shows the equation  $\phi(\mathbf{x}, t) = \tilde{\phi}(\mathbf{x}, t) + \phi'(\mathbf{x}, t)$ . A red arrow points from the term  $\tilde{\phi}(\mathbf{x}, t)$  to a box labeled "Filtered quantity or resolved field" and "Grid scales GS". Another red arrow points from the term  $\phi'(\mathbf{x}, t)$  to a box labeled "Small scales, residual quantity, or unresolved fields" and "Sub-grid scales SGS".

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

Small scales, residual quantity, or unresolved fields  
**Sub-grid scales SGS**

Filtered quantity or resolved field  
**Grid scales GS**

# Introduction to turbulence modeling

## Filtered Navier-Stokes equations

- The derivation of the filtered incompressible Navier-Stokes equations (FNS) is somehow similar to the derivation of the incompressible RANS equations.
- The main difference is that instead of using the Reynolds decomposition, we use the LES decomposition (first introduced by Deardoff [1]),

$$\begin{aligned}\mathbf{u}(\mathbf{x}, t) &= \tilde{\mathbf{u}}(\mathbf{x}) + \mathbf{u}'(\mathbf{x}, t), \\ p(\mathbf{x}, t) &= \tilde{p}(\mathbf{x}) + p'(\mathbf{x}, t),\end{aligned}$$

- Remember, the tilde represents the filtered quantity.
- The prime symbol represents the unresolved quantity (or residual).
- The LES decomposition uses local spatial filter (or in the simplest case, spatial average).
- This is the main difference between the LES and Reynolds decomposition (which uses time averaging).



# Introduction to turbulence modeling

## Filtered Navier-Stokes equations

- To derive the filtered incompressible Navier-Stokes equations (FNS), we apply the filtering operator directly to the primitive variables.
- After some algebra, we arrive to the FNS equations,

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

- As you can see, the FNS are very similar to the starting equations and the RANS equations.
- The main different is that all variables are filtered in space and we need to somehow model the sub-grid scale stress tensor  $\tau^{SGS}$ .
- To derive the compressible LES equations, we proceed in a similar way, but we use mass-weighted averaging.

# Introduction to turbulence modeling

## Filtered Navier-Stokes equations

- The sub-grid scale stress tensor represents the effect of small scales and can be written as,

$$\tau^{SGS} = (\widetilde{\tilde{\mathbf{u}}\tilde{\mathbf{u}}} - \tilde{\mathbf{u}}\tilde{\mathbf{u}}) + (\widetilde{\tilde{\mathbf{u}}\mathbf{u}'} + \widetilde{\mathbf{u}'\tilde{\mathbf{u}}}) + \widetilde{\mathbf{u}'\mathbf{u}'} = \mathbf{L} + \mathbf{C} + \mathbf{R}$$

- This is known as the triple decomposition, and it accounts for the interaction of the resolved scales and the small scales.
- In this definition,  $\mathbf{L}$  is called the Leonard stresses,  $\mathbf{C}$  is called the cross-stress term, and  $\mathbf{R}$  is called the sub-grid scale Reynolds stress (equivalent to the Reynolds stress tensor).
- The Leonard stresses ( $\mathbf{L}$ ) involves only the resolved quantities, and therefore it can be computed.
- The cross-term stresses ( $\mathbf{C}$ ) and SGS Reynolds stresses ( $\mathbf{R}$ ), they both involve unresolved scales and must be modeled.
- The cross-term stress represents the interaction of resolved and unresolved scales, whereas the SGS Reynolds stress represents the interaction of unresolved scales.
- At this point, the problem is how to model the cross-term stress and the sub-grid scale Reynolds stress.

# Introduction to turbulence modeling

## Filtered Navier-Stokes equations

- It is important to mention that the decomposition of the sub-grid scale stress tensor is not unique.

$$\tau^{SGS} = \underbrace{(\overline{\bar{u}\bar{u}} - \bar{u}\bar{u})}_{\mathbf{L}} + \underbrace{(\overline{\bar{u}u'} + \overline{u'\bar{u}})}_{\mathbf{C}} + \underbrace{\overline{u'u'}}_{\mathbf{R}} = \mathbf{L} + \mathbf{C} + \mathbf{R}$$

- In this case we used the triple decomposition or Leonard decomposition.
- Also, this decomposition is not Galilean invariant (i.e., the solution is not the same in every inertial frame).
- Germano [1], proposed a modification to make this decomposition Galilean invariant,

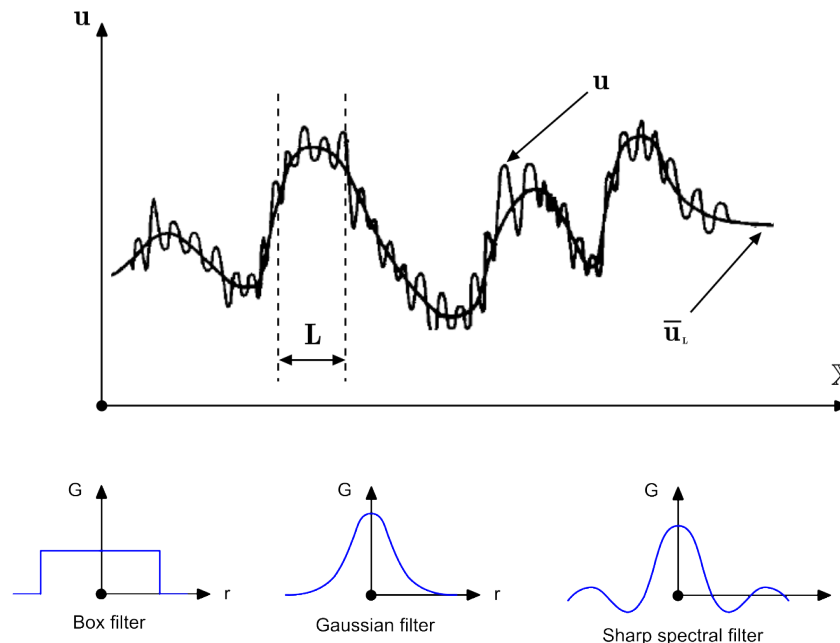
$$\tau^{SGS} = L_{ij}^m + C_{ij}^m + R_{ij}^m$$

- Where the superscript m indicates the modified stresses.

# Introduction to turbulence modeling

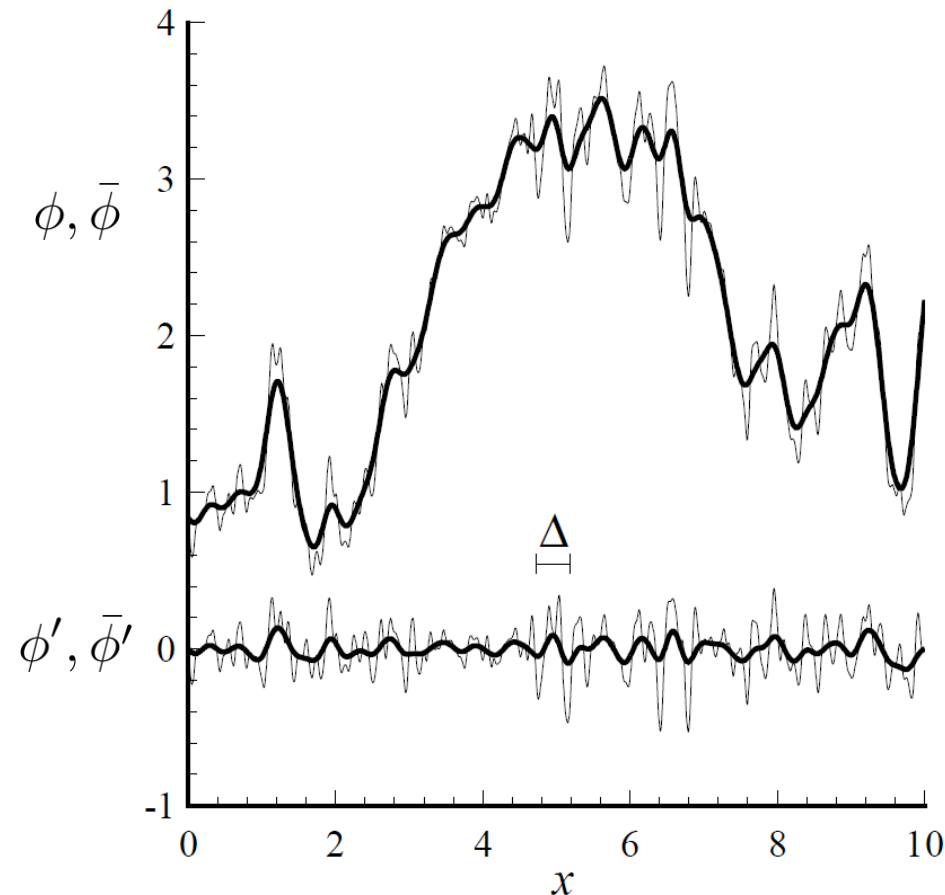
## Filtered Navier-Stokes equations

- In LES models, the governing equations need to be filtered (local spatial filter).
- You can see these filters as a rolling average used to filter fluctuations.
- In the figure below, you can see the effect of filtering a signal and three common filter kernels.
- The local spatial filtering operation simply consists in removing the very small fluctuations.
- Different filters have different properties.



# Introduction to turbulence modeling

## Filtered Navier-Stokes equations



- In the figure, the upper thin line represents the sampled velocity and the bold line the filtered velocity using a Gaussian filter.

$$\bar{\phi}(\mathbf{x}, t) = \int G(r, \mathbf{x}) \mathbf{u}(\mathbf{x} - r, t) dr$$

Filter function

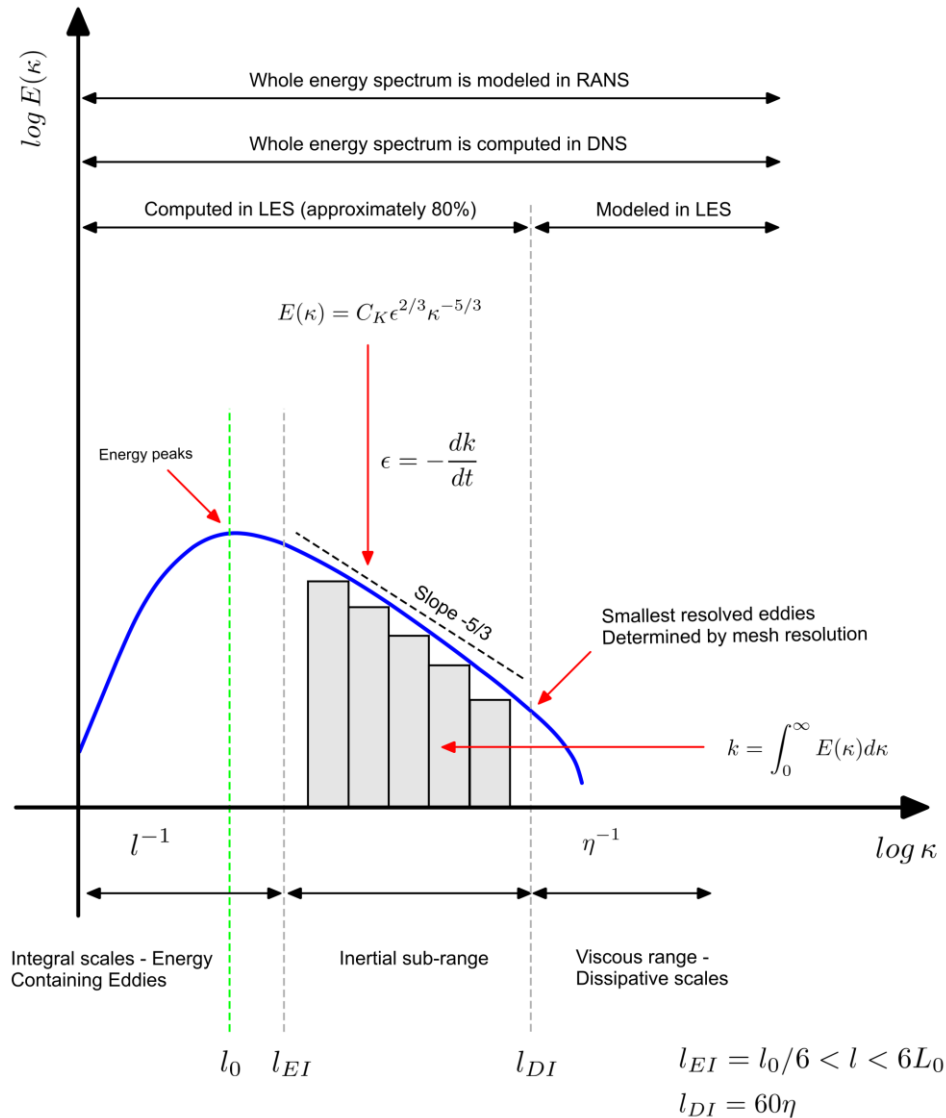
- In the figure, the lower thin line represents the field residual (fluctuations) and the bold line the filtered field residual.

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

- Remember, the filtering operation is done in space and in every single time-step.

# Introduction to turbulence modeling

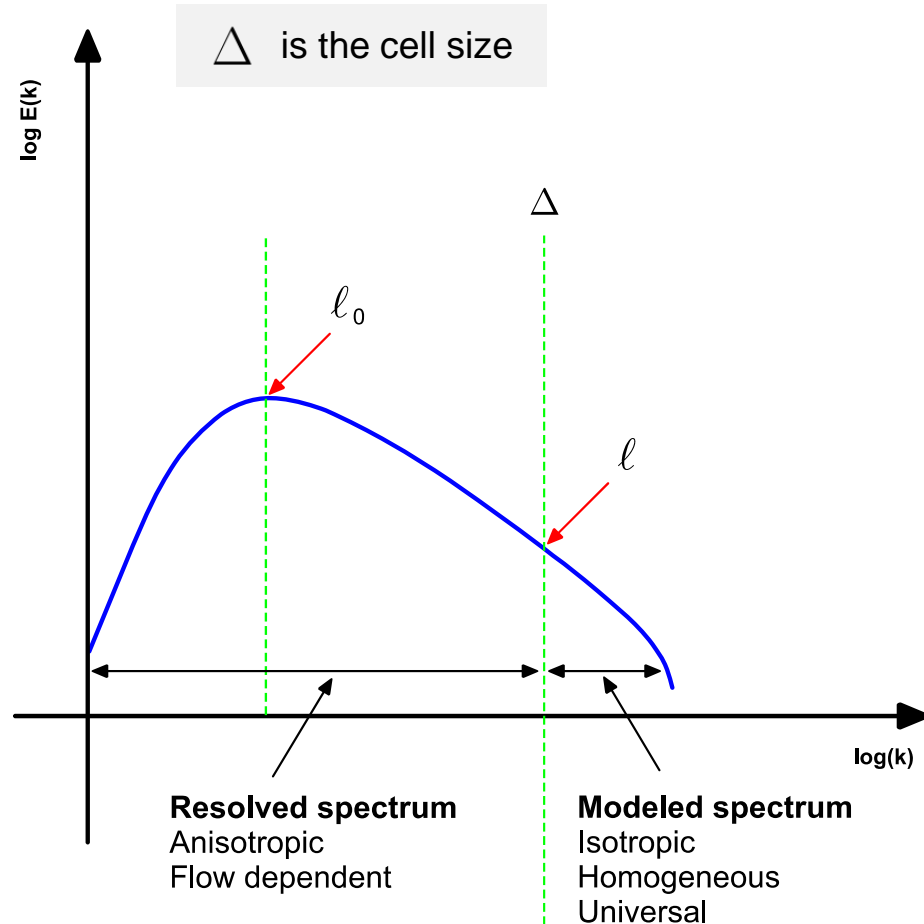
## Filtered Navier-Stokes equations – Turbulence energy spectrum



- In LES simulations, we aim at resolving 80% of the turbulence energy spectrum.
- Recall that 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.
- Remember, the turbulent kinetic energy peaks at integral length scale  $l_0$ .
- In SRS simulations, this scale must be sufficiently resolved.

# Introduction to turbulence modeling

## Filtered Navier-Stokes equations – Turbulence energy spectrum



- The finer the mesh the less energy that is being modeled.
- The width of the filter function is determined by the local mesh size.
- Many of the filtering functions are designed for uniform hexahedral meshes.
- Therefore, in LES simulations it is highly recommended to use hexahedral meshes with a low growth rate factor (preferably less than 1.1).
- Your goal is to have a mesh fine enough so that it will resolve 80% of the turbulence energy spectrum.

# Introduction to turbulence modeling

## LES sub-grid scale models

- As for RANS/URANS turbulence models, in LES there are many different models to approximate the apparent stress tensor that appears in the governing equations as a consequence of the filtering operation.
- We will call this tensor the sub-grid scale stress tensor or  $\tau_{sgs}$ .
- Just to name a few models to approximate the sub-grid scale stress tensor:
  - Smagorinsky, dynamic Smagorinsky-Lilly, Deardorff, WALE, Germano dynamic model, Algebraic WMLES S-Omega Model Formulation, one equation kinetic energy transport (standard and dynamic), Bardina model, and so on.
- As for RANS, the sub-grid scale models are based on the Boussinesq hypothesis and the gradient diffusion hypothesis.
- Most of the models are algebraic (or zero equations).
- Meaning that they do not use additional equations to model the stress tensor.



# Introduction to turbulence modeling

## LES sub-grid scale models – The Smagorinsky model

- Let us introduce the Smagorinsky model [1].
- In this model, the sub-grid scale stress tensor  $\tau^{SGS}$  is modeled as follows,

$$\tau^{SGS} = -2\nu_{SGS}\bar{\mathbf{S}}$$

- Where  $\bar{\mathbf{S}}$  is the strain-rate tensor, and  $\nu_{SGS}$  is the sub-grid scale eddy viscosity.
- As for RANS models, our task is to somehow compute the turbulent viscosity, or sub-grid scale eddy viscosity  $\nu_{SGS}$ .

# Introduction to turbulence modeling

## LES sub-grid scale models – The Smagorinsky model

- In the Smagorinsky model, the sub-grid scale stress tensor  $\tau^{SGS}$  is modeled as follows,

$$\tau^{SGS} = -2\nu_{SGS}\bar{\mathbf{S}}$$

- Where  $\bar{\mathbf{S}}$  is the resolved strain-rate tensor, and is defined as follows,

$$\bar{\mathbf{S}} = \frac{1}{2} (\nabla \bar{\mathbf{u}} + \nabla \bar{\mathbf{u}}^T)$$

- And  $\nu_{SGS}$  is the sub-grid scale eddy viscosity, and is computed using the following relationship,

$$\nu_{SGS} = (C_S \Delta)^2 |\bar{\mathbf{S}}|$$

# Introduction to turbulence modeling

## LES sub-grid scale models – The Smagorinsky model

- In the sub-grid scale eddy viscosity relation,
  - $\Delta$  is the filter width (proportional the cell spacing).
  - $C_s$  is the Smagorinsky constant (or coefficient).
  - and  $|\bar{\mathbf{S}}|$  is defined as (magnitude of the strain rate tensor),

$$|\bar{\mathbf{S}}| = (2\bar{\mathbf{S}} : \bar{\mathbf{S}})^{1/2}$$

- The filter width  $\Delta$  is usually computed as follows,

$$(\Delta_x \Delta_y \Delta_z)^{1/3} = \text{Cell volume}^{1/3}$$

- Obviously, the filter width approximation is accurate for uniform hexahedral cell.
- Depending on the cell shape different formulations are available.

# Introduction to turbulence modeling

## LES sub-grid scale models – The Smagorinsky model

- Notice that the sub-grid scale eddy viscosity relation resembles a mixing length formulation (refer to the Prandtl mixing length formulation),

Smagorinsky formulation

$$\nu_{SGS} = l_m^2 |\bar{\mathbf{S}}|$$

$$l_m = C_s \Delta$$

Prandtl mixing length formulation

$$\nu_t = l_m^2 \left| \frac{\partial \bar{u}}{\partial y} \right|$$

$$l_m = \kappa y$$

- In these relations,  $l_m$  represents a characteristics mixing length.
- Based on dimensional grounds, the Smagorinsky model can be written as follows,

$$\nu_{SGS} = (C_s \Delta)^2 |\bar{\mathbf{S}}|$$

# Introduction to turbulence modeling

## LES sub-grid scale models – The Smagorinsky model

- The Smagorinsky model is the oldest LES model, but because of its simplicity, it is widely used.
- It is not a particularly good choice for wall-bounded flows, but for shear free flows it can be quite adequate.
- In the Smagorinsky model, the value of the constant (or coefficient)  $C_s$  is of the order  $\mathcal{O}(10^{-1})$ .
- The values found in the literature can range anywhere from 0.1 to 0.35.
- Many of the drawbacks of the Smagorinsky model are due to the fact that the value of this coefficient can depend on the flow conditions.
- To overcome the drawbacks of the Smagorinsky model, more refined models have been developed.
  - Such as dynamic models that automatically compute the value of the coefficient in space and time.

# Introduction to turbulence modeling

## LES sub-grid scale models – The Smagorinsky model

- In the Smagorinsky model, the original value of the constant (or coefficient)  $C_s$  is equal to 0.18.
- The value of the coefficient  $C_s$  was first suggested by Lilly [1].
- Different authors have found similar values [2,3].
- The Smagorinsky coefficient  $C_s$  can be approximated in terms of the Kolmogorov constant  $C_K$  as follows,

$$C_s \approx \frac{1}{\pi} \left( \frac{3}{2} C_K \right)^{-3/4}$$

- Where if  $C_K$  is equal to 1.4, then  $C_s$  is equal to 0.18.
- It is worth stressing that the value of this coefficient is not universal.
- For wall bounded flows a value of 0.1 is recommended, for isotropic turbulence a value of 0.18 can be used (the same value found by Lilly using isotropic turbulence data), and for shear free flows a value of 0.24 is usually a good choice.

[1] D. Lilly. The representation of small-scale turbulence in numerical simulation experiments. Proc. IBM Scientific Computing Symposium on Environmental Sciences, IBM, White Plains, NT, 1967.

[2] V. Canuto, Y. Cheng. Determination of the Smagorinsky-Lilly constant  $C_s$ . Phys. Fluids, 9, 1368-1378, 1997.

[3] F. Nicoud, F. Ducros. Subgrid-scale stress modelling based on the square of the velocity gradient tensor. Flow, Turb. Combustion, 62. 1999.

# Introduction to turbulence modeling

## LES sub-grid scale models – The Smagorinsky model

- Let us study the rationale behind the Smagorinsky model.
- On dimensional grounds, the sub-grid scale turbulent viscosity has the following units,

$$\nu_{SGS} = \frac{m^2}{s}$$

- Using the length scale and the velocity, the sub-grid scale turbulent viscosity can be expressed as follows,

$$\nu_{SGS} \propto lU$$

- However, this relation of proportionality is not Galilean invariant.
- Moreover, we need to provide a mean to compute the length scales.
- Therefore, it should be a better way.

# Introduction to turbulence modeling

## LES sub-grid scale models – The Smagorinsky model

- By using the Boussinesq hypothesis, we can compute the sub-grid scale turbulent viscosity using the strain rate, as follows,

$$\nu_{SGS} = l^2 |\overline{\mathbf{S}}| \rightarrow \frac{m^2}{s}$$

- This relation is dimensionally consistent and Galilean invariant.
- At this point, we need estimate the integral length scale.
- In LES simulations, the length scales are determined by the cell size. Therefore,

$$l = C_s \Delta$$

- Where  $\Delta$  is the filter width, that can be approximated as follows,

$$\Delta = \text{Cell volume}^{1/3}$$



# Introduction to turbulence modeling

## LES sub-grid scale models – The Smagorinsky model

- At this point, the Smagorinsky model can be written as follows,

$$\nu_{SGS} = (C_S \Delta)^2 |\bar{\mathbf{S}}|$$

- Notice that as the mesh gets finer, the model produces less sub-grid scale turbulent viscosity (it approaches to a DNS).
- The Smagorinsky coefficient  $C_s$  can be seen as a scaling factor of the turbulent eddies.
- That is, it is used to scale down the eddies size so there are no larger than the filter width.
- Obviously, to respect this behavior the value of the coefficient  $C_s$  has to be less than 1 and larger than 0.
- As you can see, this model resembles the Prandtl mixing length formulation.
- This formulation is also based on the idea that the cells are hexahedral (same applies for all LES turbulence models).
- Nevertheless, LES models can be used with general unstructured polyhedral cells.
- But have in mind that irregular cells can add significant numerical diffusion.

# Introduction to turbulence modeling

## LES sub-grid scale models – The Smagorinsky model

- This model does not provide any means to compute sub-grid scale turbulent kinetic energy or  $k_{SGS}$  (the modelled turbulent kinetic energy).
- Again, using dimensional analysis we can derive a relation for  $k_{SGS}$ , as follows,

$$k_{SGS} = \left( \frac{\nu_{SGS}}{C_S \Delta} \right)^2$$

Recall that the dimensions of the turbulent kinetic energy are  $\text{m}^2/\text{s}^2$

- Another alternative to compute  $k_{SGS}$  has been proposed in references [1,2,3], as follows,

$$k_{SGS} = C_I \Delta^2 |\bar{\mathbf{S}}|^2$$

- Where  $C_I$  is approximately equal to 0.10 (this approach is widely use in atmospheric flows).

[1] G. Erlebacher, M. Hussaini, C. Speziale, T. Zang. Toward the large-eddy simulation of compressible turbulent flows. J. Fluid Mech. 238, 155–185, 1992.  
[2] P. Moin, K. Squires, K., W. Cabot, S. Lee. A dynamic subgrid-scale model for compressible turbulence and scalar transport. Phys. Fluids A 3(11), 2746–2757, 1991.  
[3] A. Yoshizawa. Statistical theory for compressible turbulent shear flows, with the application to subgrid modeling. Phys. Fluids 29(7), 2152–2164, 1986.

# Introduction to turbulence modeling

## LES sub-grid scale models – Dynamic Smagorinsky-Lilly model

- One of the largest deficiencies of the Smagorinsky model is that the coefficient  $C_s$  needs to be calibrated.
- Germano et al. [1] and Lilly [2], conceived a procedure in which the Smagorinsky model constant is dynamically computed in space and time based on the information provided by the resolved scales of motion.
- The concept of the dynamic procedure is to apply a second filter (called the test filter) to the equations of motion.
- The new filter width is usually equal to twice the grid filter width.
  - Both filters produce a resolved flow field.
- The difference between the two resolved fields is the contribution of the small scales whose size is in between the grid filter and the test filter.
- The information related to these scales is used to compute the model constant,

$$C_s (\mathbf{x}, t)$$

[1] M. Germano, U. Piomelli, P. Moin, W. H. Cabot. Dynamic Subgrid-Scale Eddy Viscosity Model. In Summer Workshop. Center for Turbulence Research, Stanford, CA. 1996.

[2] D. K. Lilly. A Proposed Modification of the Germano Subgrid-Scale Closure Model. Physics of Fluids. 4. 633–635. 1992.

# Introduction to turbulence modeling

## LES sub-grid scale models – Dynamic Smagorinsky-Lilly model

- In the dynamic Smagorinsky model, the sub-grid scale eddy viscosity is computed as follows,

$$\nu_{SGS} = C_s (\mathbf{x}, t) \Delta^2 |\bar{\mathbf{S}}|$$

- The use of the second filter leads to the so-called sub-test scale stresses,

$$\tau_{ij}^{ST} = \widehat{\overline{u_i u_j}} - \hat{u}_i \hat{u}_j$$

- The sub-test scale stresses are related to the SGS stresses via the Germano identity [1] as follows,

$$\hat{L}_{ij} = \tau_{ij}^{ST} - \hat{\tau}_{ij}^{SGS} = \widehat{\overline{u_i u_j}} - \hat{u}_i \hat{u}_j$$

- Where  $\hat{L}_{ij}$  denotes the Leonard stresses associated with the test filter.
- It represents the contribution to the Reynolds stresses by the scales whose length is contained between the filter width  $\Delta$  and the test filter width  $\hat{\Delta} = 2\Delta$ .

# Introduction to turbulence modeling

## LES sub-grid scale models – Dynamic Smagorinsky-Lilly model

- If we express the sub-test scale stresses and SGS stresses using the eddy viscosity approach, we obtain,

$$\hat{L}_{ij} - \frac{\delta_{ij}}{3} L_{kk} = -2C_s M_{ij}$$

- Where,

$$M_{ij} = \hat{\Delta}^2 \left| \hat{\bar{S}} \right| \hat{\bar{S}}_{ij} - \left[ \widehat{\Delta^2 \left| \bar{S} \right|} \bar{S}_{ij} \right]$$

- At this point,  $C_s$  can be computed using the least-squares minimization proposed by Lilly [1],

$$C_s(\mathbf{x}, t) = -\frac{1}{2} \frac{L_{ij} M_{ij}}{M_{ij}^2}$$

# Introduction to turbulence modeling

## LES sub-grid scale models – Dynamic Smagorinsky-Lilly model

- As for the standard Smagorinsky model, this model does not provide any means to compute sub-grid scale turbulent kinetic energy  $k_{SGS}$ .
- Based on dimensional grounds,  $k_{SGS}$  can be computed as follows,

$$k_{SGS} = \left( \frac{\nu_{SGS}}{C_{DS}\Delta} \right)^2$$

- Where  $C_{DS}$  is the dynamic coefficient computed by the model.
- Usually,  $C_{DS}$  is allowed to vary between 0 and 0.25.
- The dynamic Smagorinsky-Lilly model performs much better than standard Smagorinsky model.
- However, from a computational point of view, it is much more expensive.
- Generally speaking, this model is recommended over the standard Smagorinsky model.

# Introduction to turbulence modeling

## LES sub-grid scale models – The WALE model

- An alternative to the Smagorinsky model is the Wall-Adaptive Local Eddy (WALE) viscosity model [1].
- This model is designed to overcome many of the deficiencies of the Smagorinsky model without adding significant new complexities.
- In this model, the sub-grid scale eddy viscosity is computed as follows,

$$\nu_{SGS} = (C_W \Delta)^2 \frac{(S_{ij}^d S_{ij}^d)^{3/2}}{(\bar{S}_{ij} \bar{S}_{ij})^{5/2} + (S_{ij}^d S_{ij}^d)^{5/4}}$$

- Where  $S_{ij}^d$  (filtered velocity gradient tensor) is defined as follows,

$$S_{ij}^d = \frac{1}{2} \left( \frac{\partial \bar{u}_i}{\partial x_k} \frac{\partial \bar{u}_k}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_k} \frac{\partial \bar{u}_k}{\partial x_i} \right) - \frac{1}{3} \delta_{ij} \frac{\partial \bar{u}_k}{\partial x_l} \frac{\partial \bar{u}_l}{\partial x_k}$$

- And  $C_W$  is the model coefficient, which values can go anywhere between 0.3 to 0.6.

# Introduction to turbulence modeling

## LES sub-grid scale models – The WALE model

- As for the previous models, this model does not provide any means to compute sub-grid scale turbulent kinetic energy  $k_{SGS}$ .
- Again, based on dimensional grounds,  $k_{SGS}$  can be computed as follows,

$$k_{SGS} = \left( \frac{\nu_{SGS}}{C_W \Delta} \right)^2$$

- Where  $C_W$  is the model coefficient and is usually between  $0.55 < C_W < 0.6$  [1].
- This model performs much better than the standard Smagorinsky model.
- Also, its performance is comparable to that of the dynamic Smagorinsky-Lilly but with a lower computational cost.
- This model is recommended over the standard Smagorinsky model.



# Introduction to turbulence modeling

## LES – Final remarks

- A very good alternative to the Smagorinsky model is the WALE model (Wall-Adaptive Local Eddy Viscosity model).
- This model overcomes many of the drawbacks of the Smagorinsky model and retains its simplicity.
  - The WALE model predicts accurately the flow near the walls.
  - It also predicts transition, but you need very fine meshes.
  - The model constants (or coefficients) can depend on the flow conditions.
- Dynamic models where the Smagorinsky coefficient is dynamically computed in function of space and time offer superior performance, but at a slightly higher computational cost.
  - Dynamic models are strongly recommended.

# Introduction to turbulence modeling

## LES – Final remarks

- Some more advanced LES models introduce transport effects by solving an equation for the turbulent kinetic energy and using double filtering to find out more information about the sub-grid scales.
  - Clearly, these models are more expensive than the algebraic models.
  - However, they give information about the modeled TKE.
  - These models are also a good choice.
  - One equation TKE LES models (standard and dynamic) are often available in many CFD solvers.

# Introduction to turbulence modeling

## LES wall functions

- It is highly recommended to use wall functions with LES simulations.
- Otherwise, the meshing requirements will be close to those of DNS.
- Conservatively speaking, the upper limit of  $y^+$  for LES and DES simulations should be less than  $y^+ < 200$ .
- And you should cluster enough points in the boundary layer to accurately resolve the velocity profile normal to the wall.
- Usually, at least 15 inflation layers are required to meet good resolution requirements.
- It is also imperative to use  $y^+$  insensitive wall functions.
- That is, formulations that cover viscous sublayer, buffer region, and log-law region.

# Introduction to turbulence modeling

## LES wall functions

- Remember, you also need to resolve the boundary layer in the spanwise and streamwise directions, and this is what mainly imposes stringent meshing requirements in LES simulations.
- Recommended wall distance units values are (these are indicative values based on different references),

$$\Delta x^+ < 50, \Delta z^+ < 50 \quad \text{for} \quad y^+ < 5$$

Wall resolving

$$\Delta x^+ < 4\Delta y^+, \Delta z^+ < 4\Delta y^+ \quad \text{for} \quad y^+ \leq 200$$

Wall modeling

**Note:** it is recommended to use a surface mesh aspect ratio of 10 or less.

- As you can see, these requirements translate in much finer surface meshes, low surface aspect ratios, and clustered meshes towards the walls.
- You also need to use low growth rate factors, in the order of 1.1 or less.

# Introduction to turbulence modeling

## LES wall functions

- The wall functions treatment is similar to that of RANS/URANS models.
- Remember, it is imperative to use  $y^+$  insensitive wall functions.
- This can be achieved by using a blending function between the viscous sublayer and the log-law layer [1].
- Kader [1] proposed the following blending function to obtain a  $y^+$  insensitive formulation,

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

$$\Gamma = -\frac{a (y^+)^4}{1 + by^+} \quad a = 0.01 \quad b = 5$$

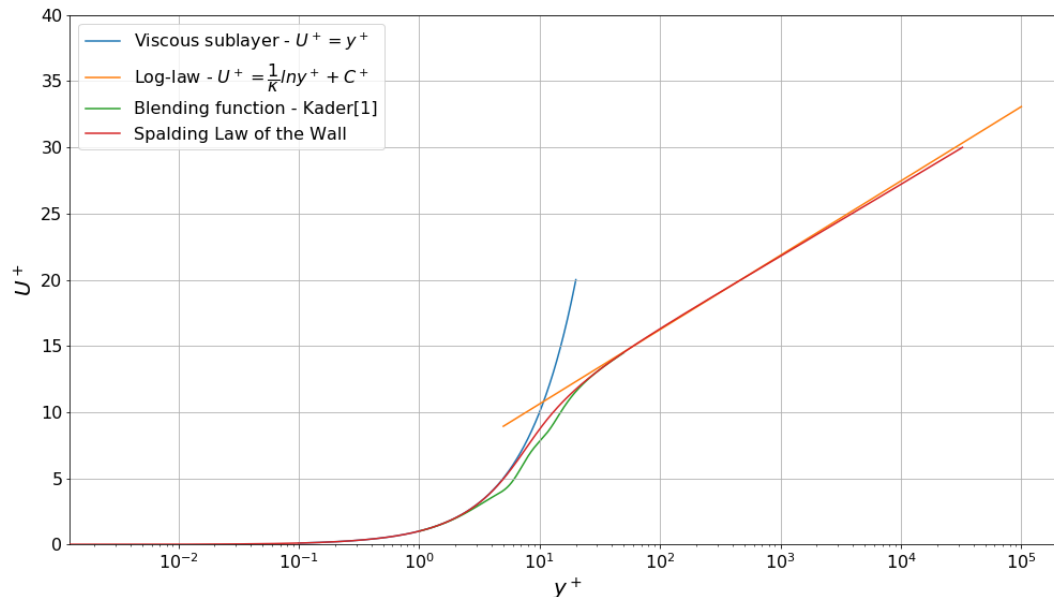
$$u_{lam}^+ = y^+ \quad u_{turb}^+ = \frac{1}{\kappa} \ln y^+ + C^+$$

- This formula guarantees the correct asymptotic behavior for large and small values of  $y^+$  and a reasonable representation of velocity profiles in the cases where  $y^+$  falls inside the buffer region.

# Introduction to turbulence modeling

## LES wall functions

- Plot of Kader's [1] blending function.
- In the plot, the Spalding function [2] is also represented.
- The Spalding function is another alternative to obtain a  $y^+$  insensitive treatment.
- The Spalding function is essentially a fit of the laminar, buffer and logarithmic regions of the boundary layer.



- Kader's [1] blending function,

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

$$u^+ = \begin{cases} y^+ & y^+ < 11.225 \\ \frac{1}{\kappa} \ln(y^+) + C^+ & y^+ > 11.225 \end{cases}$$

- Or using  $u^*$  wall units

$$u^* = \begin{cases} y^* & y^* < 11.225 \\ \frac{1}{\kappa} \ln(E y^*) & y^* > 11.225 \end{cases}$$

- And recall that in equilibrium conditions,

$$u^+ = u^* \quad y^* = \frac{C_\mu^{1/4} k_p^{1/2} y_p}{\nu} \quad y^+ = \frac{\rho \times U_\tau \times y}{\mu}$$

- However, in the context of LES simulations the use of  $u^+$  wall units is preferred because  $k^{sgs}$  is not explicitly computed.
- The Spalding's law [2] is written as follows,

$$y^+ = u^+ + \frac{1}{E} \left[ e^{\kappa u^+} - 1 - \frac{\kappa u^+}{1!} - \frac{(\kappa u^+)^2}{2!} - \frac{(\kappa u^+)^3}{3!} - \frac{(\kappa u^+)^4}{4!} \right]$$

# Introduction to turbulence modeling

## LES wall functions

- Finally, this is not the only approach when using wall functions in LES simulations.
- Just to name a few other approaches,
  - The Shumann model.
  - The Grotzbach model.
  - The shifted corrections model.
  - Election model.
  - The optimized ejection model by Marusic.
  - The model of Werner and Wengle [1].
  - The model of Murakami.
  - The suboptimal control-based model.
  - The Deardorff model.
  - Two-layer model by Balaras.
  - Das-Moser embedded wall model
  - Hybrid LES/RANS models.
- For a general review of these models please refer to references [2,3,4].

[1] H. Werner, H. Wengle. Large-Eddy Simulation of Turbulent Flow Over and Around a Cube in a Plate Channel. Eighth Symposium on Turbulent Shear Flows, Munich, Germany. 1991.

[2] P. Sagaut. Large eddy simulation for incompressible flows. An introduction. Springer, 2006.

[3] S. Pope. Turbulent Flows. Cambridge University Press. 2014.

[4] U. Piomelli, E. Balaras. Wall-layer models for large-eddy simulations. Annual Review of Fluid Mechanics. Vol. 35:349-374, 2002

# Introduction to turbulence modeling

## Short description of some LES turbulence models

Model	Short description
<b>Smagorinsky</b>	Simple algebraic model (0-equations). Because of its simplicity and low computational cost, it is widely used. It is not a particularly good choice for wall-bounded flows, but for flows far from solid boundaries it can be quite adequate. The model constants can depend on the flow conditions. This model is a good starting point for complex simulations.
<b>Smagorinsky-Lilly</b>	Simple algebraic model (0-equations). Because of its simplicity and low computational cost, it is widely used. It overcomes some of the limitations of the Smagorinsky model by using damping functions in near-wall regions, therefore, it works better with wall-bounded flows. The model constants can depend on the flow conditions.
<b>Wall-Addaptive Local Eddy Viscosity model (WALE)</b>	Simple algebraic model (0-equations). Retains the simplicity and low computational cost of of the Smagorinsky model. Wall damping effects are accounted for without using the damping function explicitly. It predicts accurately the flow near the walls and transition. The model constants can depend on the flow conditions.
<b>Dynamic Smagorinsky</b>	This model is based on the similarity concept and Germano's identity. It is more universal because the constants are computed dynamically. The model predicts accurately the wall behavior, transition, and allow energy backscatter. The computation of dynamic constants requires additional computational power and fluctuations while computing the constant can cause stability issues.
<b>Dynamic Kinetic Energy Transport</b>	This model overcomes some of the limitations of the Smagorinsky-Lilly and dynamic Smagorinsky models. It solves an additional transport equation for the subgrid scale kinetic energy. The model predicts accurately the wall behavior, transition, and allow energy backscatter, it also allows for history of the kinetic energy. It is more computational expensive as it solves an additional equation and it performs explicit filtering.



# Introduction to turbulence modeling

## DES models

- An alternative to LES models, is the use of detached eddy simulation (DES) models.
- Detached-Eddy Simulation (DES) is a hybrid technique first proposed by Spalart et al. [1] for prediction of turbulent flows at high Reynolds numbers (see also [2,3]).
- Development of the technique was motivated by estimates which indicate that the computational costs of applying Large-Eddy Simulation (LES) to complete configurations such as an airplane, submarine, or road vehicle are prohibitive.
- The high cost of LES when applied to complete configurations at high Reynolds numbers arises because of the resolution required in the boundary layers, an issue that remains even with fully successful wall-layer modeling.
- In Detached-Eddy Simulation (DES), the aim is to combine the most favorable aspects of RANS and LES techniques, *i.e.*, application of RANS models for predicting the attached boundary layers and LES for resolution of time-dependent, three-dimensional large eddies in the core of the flow.
- The scaling cost of the method is then favorable since LES is not applied to resolution of the relatively smaller-structures that populate the boundary layer.

[1] P. Spalart, W. Jou, M. Stretlets, S. Allmaras. Comments on the Feasibility of LES for Wings and on the Hybrid RANS/LES Approach. AFOSR Conference (1997).

[2] P. Spalart. Strategies for turbulence modelling and simulations. International Journal of Heat and Fluid Flow 21 (2000).

[3] K. D. Squires. Detached-eddy simulation: Current status and perspectives. Direct and Large-Eddy Simulation V. ERCOFTAC Series, vol 9. Springer (2004).

# Introduction to turbulence modeling

## DES models

- DES models formulations are relatively simple and can be built on top of any RANS model (usually Spalart-Allmaras or  $k - \omega$  SST).
- In DES models the switch between RANS and LES is based on a criterion similar to,

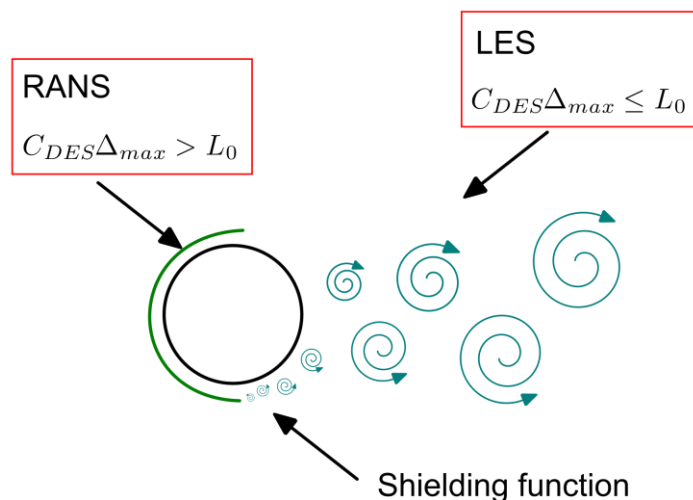
$$C_{DES}\Delta_{max} > L_0 \rightarrow \text{RANS}$$

$$C_{DES}\Delta_{max} \leq L_0 \rightarrow \text{LES}$$

where

$$L_0 = \frac{k^{1.5}}{\epsilon} = \frac{k^{0.5}}{0.09\omega}$$

- In the previous equations,  $\Delta_{max}$  is the maximum edge length of the local cell and  $C_{DES}$  is an empirical constant related to the model.



A shielding function can be used to avoid the resolved structures from entering into the boundary layer regions. This is referred to as delayed DES or DDES.

# Introduction to turbulence modeling

## DES models

- Due to the direct impact of the grid spacing on the RANS model, DES models require more carefully crafted meshes to avoid inappropriate behavior.
- The original DES model is straightforward and simple [1,2].
- Nevertheless, the user requires not only a basic understanding of the model behavior, but also has to follow intricate grid generation guidelines to avoid undefined simulation behavior somewhere between RANS and LES.
- Problematic behavior of standard DES has been reported by Menter and Kuntz [3], who demonstrated that an artificial separation could be produced for an airfoil simulation when refining the max cell edge length ( $h_{\max}$ ) inside the wall boundary layer below a critical value of  $h_{\max}/\delta < 0.5-1$ , where  $\delta$  is the local boundary layer thickness.
- This effect was termed Grid Induced Separation (GIS) [3,4] as the separation depends on the grid spacing and not the flow physics.
- GIS is obviously produced by the effect of a sudden grid refinement which changes the DES model from RANS to LES, without balancing the reduction in eddy-viscosity by resolved turbulence content.
- The switch from the RANS to the LES model inside wall boundary layers is not desirable.

[1] P. Spalart, W. Jou, M. Strelets, S. Allmaras. Comments on the Feasibility of LES for Wings and on the Hybrid RANS/LES Approach. AFOSR Conference (1997).

[2] P. Spalart. Strategies for turbulence modelling and simulations. International Journal of Heat and Fluid Flow 21 (2000).

[3] F. Menter, M. Kuntz, R. Langtry. Ten years of experience with the SST turbulence model. 4th International Symposium on Turbulence Heat and Mass Transfer (2003).

[4] P. Spalart, S. Deck, M. Shur, K. Squires, M. Strelets, A. Travin. A new version of detached-eddy simulation, resistant to ambiguous grid densities. Theor. Comput. Fluid Dyn. 20 (2006).

# Introduction to turbulence modeling

## DES models

- GIS can in principle be avoided by shielding the RANS model from the DES formulation for wall boundary layers.
- This was proposed by Menter and Kuntz [1], who used the blending functions of the SST model for that purpose.
- Later, Spalart et al. [2] proposed a more generic formulation of the shielding function, which depends only on the eddy-viscosity and the wall distance. It can therefore, in principle, be applied to any eddy-viscosity based DES model.
- The resulting formulation was termed **Delayed Detached Eddy Simulation (DDES)** [2].
- While the shielding function developed in reference [2] was considered generic, it was essentially calibrated for the Spalart-Allmaras (SA) one-equation RANS model.
- The **DDES** approach was later extended to a two-equation (Menter  $k - \omega$ ) formulation by Strelets (2001).

[1] F. Menter, M. Kuntz, R. Langtry. Ten years of experience with the SST turbulence model. 4th International Symposium on Turbulence Heat and Mass Transfer (2003).

[2] P. Spalart, S. Deck, M. Shur, K. Squires, M. Strelets, A. Travin. A new version of detached-eddy simulation, resistant to ambiguous grid densities. Theor. Comput. Fluid Dyn. 20 (2006).

[3] M. Strelets. Detached Eddy Simulation of Massively Separated Flows. 39th Aerospace Sciences Meeting and Exhibit, American Institute of Aeronautics and Astronautics (2001).

# Introduction to turbulence modeling

## DES models

- The most recent improvement of the DES formulation is termed **Improved Delayed Detached Eddy Simulation (IDDES)** [1].
- The IDDES formulation incorporates two major enhancements to the original DES method.
  - The first enhancement led to the DDES (Delayed Detached Eddy Simulation) formulation [2] which addresses the issue of GIS by preventing the switch from RANS to LES mode within attached boundary layers due solely to the specifics of the grid design.
  - The second enhancement was the introduction of a wall-modeled LES functionality [1].
  - In general, the IDDES formulation provides a wall-modeled LES (WMLES) response if resolved turbulent content is supplied as an inflow (or initial) condition, and resorts to a DDES response otherwise.
  - The wall-modeled LES behavior is intended to be active only when the inflow conditions include resolved turbulent content, and the grid is fine enough to at least resolve the largest energy containing boundary layer eddies.
  - The IDDES model features several rather intricate length scales definitions, and blending and shielding functions, which allow using this model both in DDES and WMLES mode.

[1] M. Shur, P. Spalart, M. Strelets, A. Travin. A hybrid RANS-LES approach with delayed-DES and wall-modelled LES capabilities. International Journal of Heat and Fluid Flow, 29 (2008).  
[2] P. Spalart, S. Deck, M. Shur, K. Squires, M. Strelets, A. Travin. A new version of detached-eddy simulation, resistant to ambiguous grid densities. Theor. Comput. Fluid Dyn. 20 (2006).

# Introduction to turbulence modeling

## SST DDES Formulation

- The governing equations of the SST DDES model read as [1],

$$\frac{\partial \rho k}{\partial t} + \nabla \cdot (\rho \mathbf{u} k) = \nabla \cdot [(\mu + \sigma_k \mu_t) \nabla k] + P_k - \frac{\rho \sqrt{k^3}}{l_{DDES}}$$

$$\frac{\partial \rho \omega}{\partial t} + \nabla \cdot (\rho \mathbf{u} \omega) = \nabla \cdot [(\mu + \sigma_\omega \mu_t) \nabla \omega] + 2(1 - F_1) \rho \sigma_{\omega 2} \frac{\nabla k \cdot \nabla \omega}{\omega} + \alpha \frac{\rho}{\mu_t} P_k - \beta \rho \omega^2$$

$$\mu_t = \rho \frac{a_1 \cdot k}{\max(a_1 \cdot \omega, F_2 \cdot S)}$$

- This model features several blending and shielding functions.
- Many of the constant used in this model are computed by a blend function between the respective constants of the  $k - \epsilon$  and  $k - \omega$  models.

# Introduction to turbulence modeling

## SST DDES Formulation

- The blending functions, shielding functions, and constants used reads as [1],

$$F_1 = \tanh(\arg_1^4)$$

$$\arg_1 = \min \left( \max \left( \frac{\sqrt{k}}{C_\mu \omega d_\omega}, \frac{500\nu}{d_\omega^2 \omega} \right), \frac{4\rho\sigma_{\omega 2} k}{CD_{k\omega} d_\omega^2} \right)$$

$$CD_{k\omega} = \max \left( 2\rho\sigma_{\omega 2} \frac{\nabla k \cdot \nabla \omega}{\omega}, 10^{-10} \right)$$

$$F_2 = \tanh(\arg_2^2)$$

$$\arg_2 = \max \left( \frac{2\sqrt{k}}{C_\mu \omega d_\omega}, \frac{500\nu}{d_\omega^2 \omega} \right)$$

$$P_k = \min(\mu_t S^2, 10 \cdot C_\mu \rho k \omega)$$

# Introduction to turbulence modeling

## SST DDES Formulation

- The blending functions, shielding functions, and constants used reads as [1],

$$l_{DDES} = l_{RANS} - f_d \max(0, l_{RANS} - l_{LES})$$

$$l_{LES} = C_{DES} h_{\max}$$

$$l_{RANS} = \frac{\sqrt{k}}{C_\mu \omega}$$

$$C_{DES} = C_{DES1} \cdot F_1 + C_{DES2} \cdot (1 - F_1)$$

$$f_d = 1 - \tanh \left[ (C_{d1} r_d)^{C_{d2}} \right]$$

$$r_d = \frac{\nu_t + \nu}{\kappa^2 d_\omega^2 \sqrt{0.5 \cdot (S^2 + \Omega^2)}}$$

$$C_\mu = 0.09, \kappa = 0.41, a_1 = 0.31$$

$$C_{DES1} = 0.78, C_{DES2} = 0.61, C_{d1} = 20, C_{d2} = 3$$



# Introduction to turbulence modeling

## SST DDES Formulation

- Summary of the main characteristics of DES models:
  - DES models [1] are hybrid between RANS and LES.
  - RANS models are used close to the walls, and in the far field LES models are used.
  - The near wall turbulence is not explicitly computed, but fully modeled.
  - The mesh resolution requirements close to the walls are equivalent to those of RANS/URANS.
  - These models work particularly well for detached flows and external aerodynamics.
  - Refrain from using DES models with internal flows.
  - It is strongly recommended to resolve the boundary layer.
  - It is possible to use more stretching in the stream-wise and span-wise direction than with LES because it is not necessary to resolve eddies located in the wall region.

# Introduction to turbulence modeling

## SST DDES Formulation

- Summary of the main characteristics of DES models:
  - In DES, it is critical to analyze the location of the LES/RANS interface.
    - The goal is to be in RANS mode in the wall regions and in LES mode in the free flow.
  - With DES we can use larger CFL numbers in comparison to LES.
  - When using DES models, the user requires not only a basic understanding of the model behavior, but also has to follow intricate grid generation guidelines to avoid undefined simulation behavior somewhere between RANS and LES.
  - If you are planning to use a DES model, it is recommended to use the DDES formulation.
  - DES formulations based on the Spalart-Allmaras and the k- $\omega$  SST turbulence models they both perform well when dealing with external aerodynamics.
  - However, it is recommended to use the Spalart-Allmaras formulation as it has been widely validated, and many of closure coefficients and blending functions have been explicitly calibrated for the Spalart-Allmaras formulation.

# Introduction to turbulence modeling

## A few acronyms that you will find while working with DES and LES models

- **LES**: large eddy simulation (it resolves 80% of the energy spectrum).
- **VLES**: very large eddy simulation (it resolves 50% of the energy spectrum).
- **LES-NWR**: LES with near wall resolution (it resolves the boundary layer).
- **LES-NWM**: LES with near wall modelling (it uses wall functions).
- **WMLES**: wall modeled LES (same as LES-NWM).
- **DES**: detached eddy simulation (hybrid RANS-LES).
- **DDES**: delayed DES (DES with shielding functions).
- **DDES-SA**: DDES or DES based on the Spalart-Allmaras RANS model.
- **DDES-SST**: DDES or DES based on the  $k - \omega$  SST RANS model.
- **IDDES**: improved delayed DES (DES with shielding functions and WMLES).
- **ALES**: adaptive LES (LES with adaptive mesh refinement).
- **GS**: grid scales
- **SGS**: sub-grid scales
- **HRM**: high Reynolds part of the boundary layer (logarithmic layer).
- **LRN**: low Reynolds part of the boundary layer (viscous layer).

# Introduction to turbulence modeling

## Final remarks on DES/LES turbulence models

- The mesh requirements of LES-NWR are close to those of DNS, therefore, it is highly recommended the use of wall functions.
- LES wall functions are valid in the whole boundary layer (including the buffer region).
- Remember, DES/LES methods are intrinsically 3D and unsteady.
- LES simulations are very sensitive to mesh element type; it is highly recommended to use hexahedral meshes.
- For good accuracy, try to keep the mesh growth rate below 1.1.
- If you are dealing with external aerodynamics and detached flows, DES simulations are very affordable.
- In DES, as it is not necessary to resolve eddies located in the wall region, you can use coarser meshes in stream-wise and span-wise directions.
- The WALE and dynamic methods are the best LES choices. However, you can use the Smagorinsky method for simple flows or getting an initial solution.
- For LES simulations, keep the CFL below 1.
- DES simulations have more relaxed time-stepping requirements, but in general you should not go above 5.

# Introduction to turbulence modeling

## Final remarks on DES/LES turbulence models

- Use RANS simulations as starting point for LES/DES simulations.
- When it comes to post-processing turbulence simulations, it can be quite time consuming, especially if we are dealing with unsteady data and large meshes.
- Most of the times we are interested in computing averaged quantities, so do not forget to compute the unsteady statistics.
- Not all discretization schemes are born with LES in mind. In LES simulation we must use low dissipation and non-dispersive discretization methods.
- Same applies for element type. Tetrahedral elements are not very desirable when conducting LES simulations, even if we use high-accuracy and non-dispersive methods.
- Hexes are preferred over the rest of element types. Remember, many LES filters are designed with hexes in mind.
- Low dissipation methods translate in energy preserving methods, that is, the energy spectrum should not increase (or accumulate) with large wave number (small scales).
- It is not recommended to use meshes with hanging nodes.

# Introduction to turbulence modeling

## Final remarks on DES/LES turbulence models

- Summary of the most important requirements of SRS simulations.

	LES	DES	SAS	URANS
Wall resolution $y^+$	1-300	1*	1*	1-1000**
Wall resolution $x^+$	40	No strict requirements		
Wall resolution $z^+$	20	No strict requirements		
CFL number***	1 <	5 <	5 <	10 <
Growth rate	1.1 <	1.1 <****	1.1 <****	1.2 <
Internal flows	YES	NO	NO	YES
External flows	YES	YES	YES	YES

\* Recommended values. It is strongly advised to resolve the boundary layer (LRN approach). The wall modeling approach can also be used. However, the benefits of using a HRN approach are questionable.

\*\* The upper limit depends on the Reynolds number.

\*\*\* Recommended values for good accuracy but you can use higher values at your own risk.

\*\*\*\* Recommended values in the boundary layer region.

# Introduction to turbulence modeling

## Final remarks on DES/LES turbulence models

- LES simulations are about:
  - Resolving the turbulent motion in space and time by using fine meshes and small time-steps.
  - Where we aim at modeling only a small portion of the turbulent spectrum, about 20% or less.
  - We also need to use low dissipative discretization schemes (in space and time).
  - LES simulations are also about computing your unsteady statistics and analyzing time series.

# Roadmap

- ~~1. The turbulent world around us~~
- ~~2. Turbulence, does it matter?~~
- ~~3. Introduction to turbulence modeling~~
- 4. Post-processing turbulence simulations**
- ~~5.  $y^+$  and wall distance units~~
- ~~6. Turbulence modeling in OpenFOAM~~
- ~~7. Some applications using OpenFOAM~~
- ~~8. Final remarks – General guidelines~~



# Post-processing RANS and LES simulations

## General remarks

- Post-processing turbulent simulations can be a very daunting task. Specially if we are dealing with SRS simulations.
- Turbulence simulations can be analyzed using quantitative and qualitative data, we will address the most common methods.
- From a qualitative point of view, most of the times we are interested in visualizing the vortical structures, this can be done using the Q-criterion.
- Sometimes it is interesting to know the sense of rotation of the vortices, this can be done using the vorticity criterion.
- From a quantitative point of, we are interested in plotting the time history of the forces and some other quantities of interest (such as mass flow, heat transfer coefficient, maximum and minimum values of transported quantities and so on).
- It is also of interest plotting the energy spectrum. This kind of plot is useful to determine if we are resolving well the spatial and temporal scales.

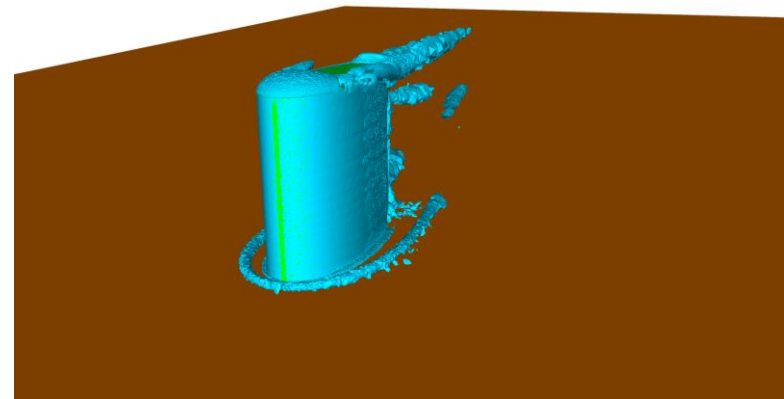
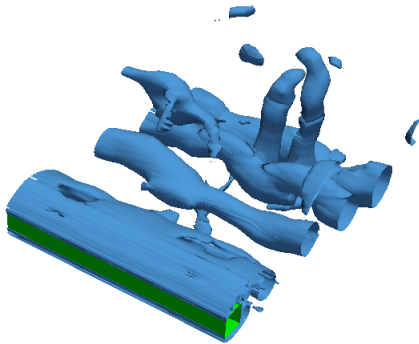
# Post-processing RANS and LES simulations

## Q-criterion

- The Q-criterion is defined as

$$Q = \frac{1}{2} \left[ (\text{tr}(\nabla \mathbf{u}))^2 - \text{tr}(\nabla \mathbf{u} \cdot \nabla \mathbf{u}) \right]$$

- To visualize the vortical structures we plot the iso-surfaces of Q-criterion. The values to plot are positives and several order of magnitudes lower than the maximum value.

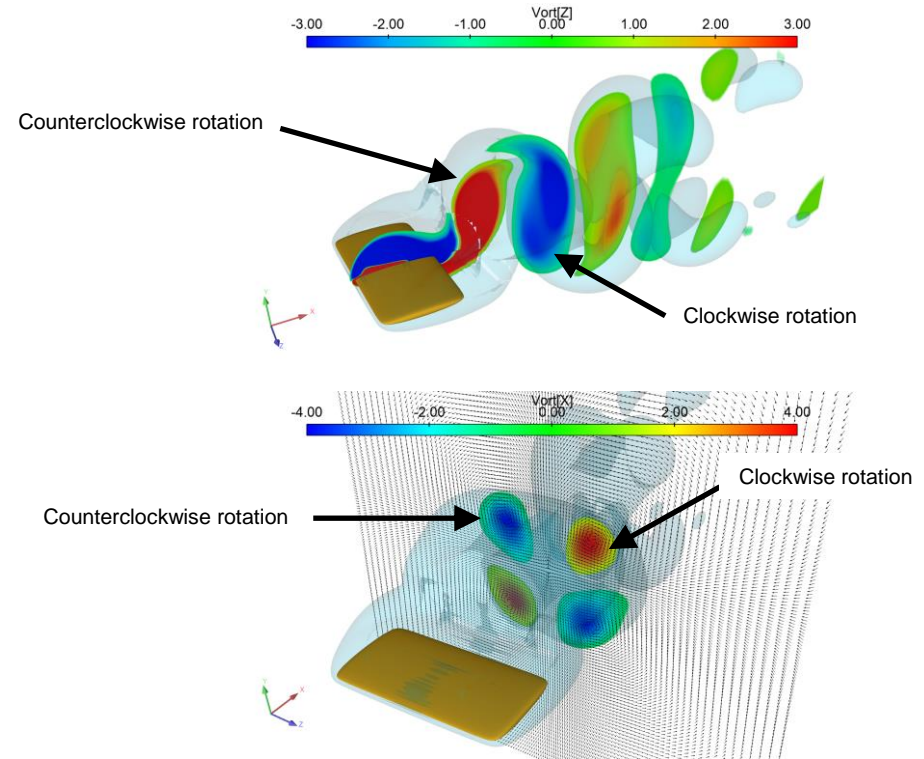
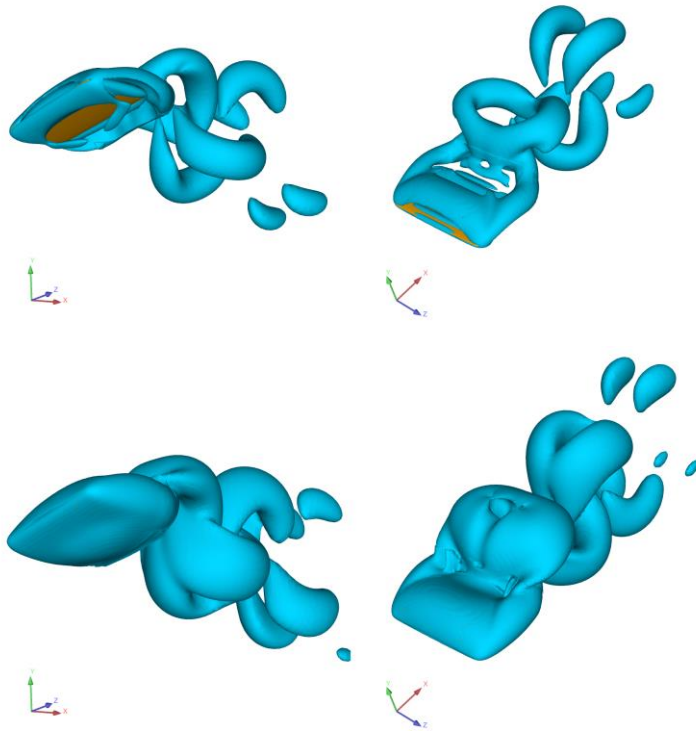


Iso-surfaces of Q-criterion

# Post-processing RANS and LES simulations

## Vorticity criterion

- The vorticity criterion is defined as  $\omega = \nabla \times \mathbf{u}$

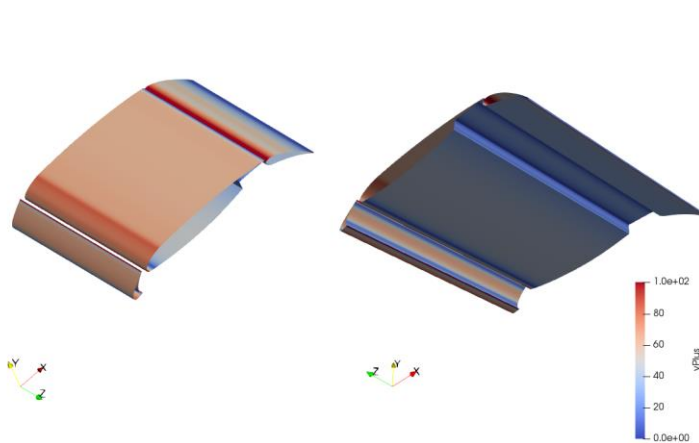


- Comparison of Q-criterion (top figure) and vorticity criterion (bottom image).
- Notice that both methods show the vortical structures, but the vorticity criterion has the disadvantage of also showing the shear layers near the body and between the vortices.
- The vorticity criterion is capable of showing the rotation of the vortices.
- In the top figure contours of  $\omega_z$  are plot. The red vortices are rotating in counter-clockwise sense.
- In the bottom figure contours of  $\omega_x$  are plot. The red vortices are rotating in clockwise sense.

# Post-processing RANS and LES simulations

## $y^+$ value

- The  $y^+$  is computed as discussed in previous slides.
- Hereafter, it is plotted on the walls.
- It is almost impossible to have a uniform  $y^+$  on the walls.
- When evaluating this quantity, we should check that the average value is roughly speaking close to our target value.
- Be sure not to have high peaks in large areas ( $y^+$  values of more than 600).



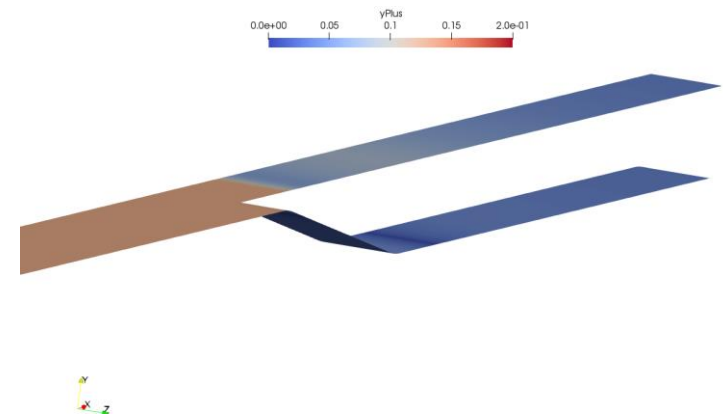
yPlus yplus write:

writing object yPlus

patch airfoil  $y^+$  : min = 3.3170682, max = 122.32767, average = 42.357341

patch flap  $y^+$  : min = 9.6251989, max = 447.31831, average = 47.411466

patch slat  $y^+$  : min = 14.072073, max = 305.59193, average = 93.392662



yPlus yplus write:

writing object yPlus

patch walls  $y^+$  : min = 0.0013513095, max = 0.29017777, average = 0.066419415

# Post-processing RANS and LES simulations

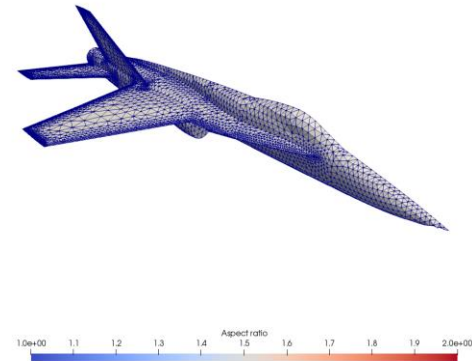
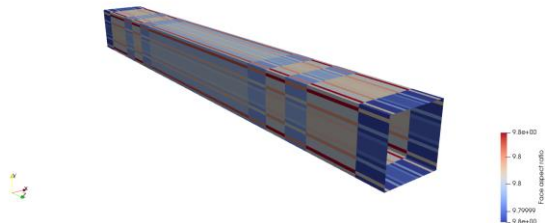
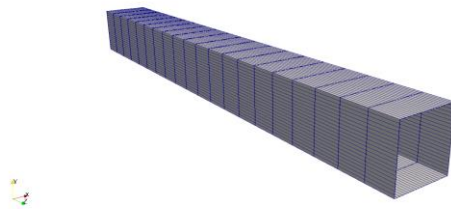
## Surface mesh quality metrics

- Many of the surface mesh quality metrics can be computed directly from the meshing tools or using the postprocessing applications.
- The surface elements can be triangles, quads, general polyhedrons, or a combination of them. This can make the evaluation of the quality metrics rather difficult.
- However, some rough approximations can be made, which most of the times are acceptable.
- When it comes to LES simulations, they have very strict requirements when it comes to mesh quality.
- Some important surface mesh quality metrics to look at:
  - Face aspect ratio.
  - Face area.
  - Longest edge.
  - Growth rate.

# Post-processing RANS and LES simulations

## Surface mesh aspect ratio

- Roughly speaking, this quantity represents the ratio between the shortest and longest edge.
- For quad elements, this quantity can be computed easily.
- For other shapes (triangles, polyhedrons, and so on), it is not very easy to compute. However, approximations can be used.
- In LES simulations, the surface mesh aspect ratio should be below five. However, values up to ten are acceptable.
- Surface mesh aspect ratio has a direct implication when computing the wall distance units  $x^+$  and  $z^+$ .



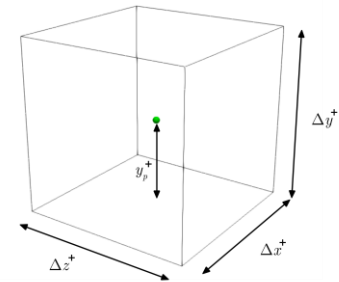
# Post-processing RANS and LES simulations

## Wall distance units $x^+$ and $z^+$

- Similar to  $y^+$ , the wall distance units can be computed in the stream-wise (  $\Delta x^+$  ) and span-wise (  $\Delta z^+$  ) directions.

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

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



- For simplicity, let us assume that they are the same in both directions (which is not a bad approximation).
- Then, we can compute  $\Delta x^+$  and  $\Delta z^+$  as follows,

$$\Delta x^+ = \Delta z^+ = \frac{\sqrt{\text{Face area}} \times y^+}{y_{\text{cell center}}}$$

- Where we approximated  $\Delta x$  and  $\Delta z$  as follows,

If the surface mesh is uniform and quadrilateral, you can measure the edge length directly from the surface cells

$$\rightarrow \Delta x = \Delta z = \sqrt{\text{Face area}} \leftarrow$$

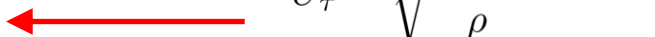
Again, this is a relatively good approximation. Have in mind that there are many more methods.

# Post-processing RANS and LES simulations

## Wall distance units $x^+$ and $z^+$

- In the previous equations,  $y_{\text{cell center}}$  is the distance normal from the surface to the first cell center.
- The quantity  $y_{\text{cell center}}$  can be directly obtained from the mesh (this is recommended if the mesh is uniform), or it can be inferred from the  $y^+$  value (for non-uniform or industrial meshes), as follows,

$$y_{\text{cell center}} = \frac{y^+ \times \nu}{U_\tau}$$



$$U_\tau = \sqrt{\frac{\tau_{\text{wall}}}{\rho}}$$

Wall shear stresses.  
Computed by the solver.

- Summarizing, these are the steps to follow to compute  $\Delta x^+$  and  $\Delta z^+$ ,
  - Compute  $y^+$  (usually computed by the solver).
  - Compute surface mesh face area.
  - Compute  $y_{\text{cell center}}$  using  $y^+$ ,  $\nu$  and  $U_\tau$ .
  - Compute  $\Delta x^+$  and  $\Delta z^+$  using the following relationship,

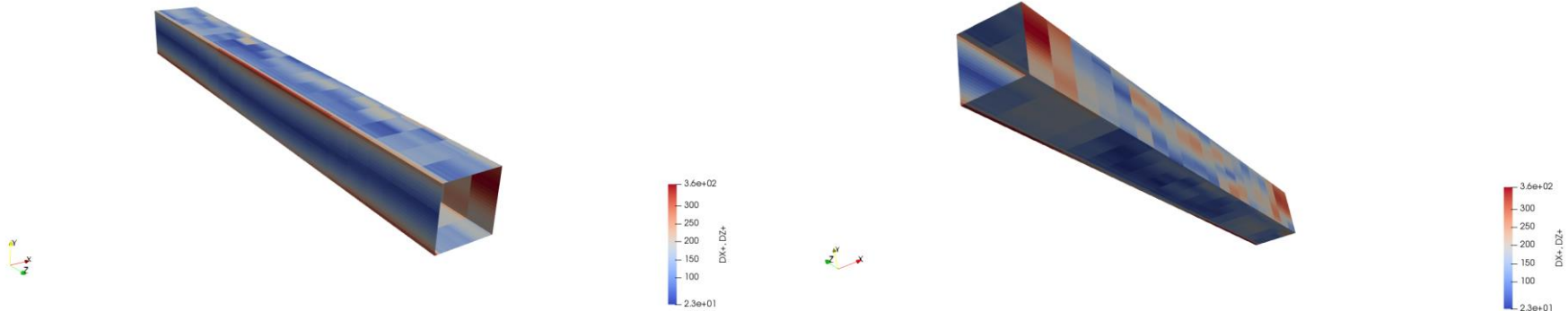
$$\Delta x^+ = \Delta z^+ = \frac{\sqrt{\text{Face area}} \times y^+}{y_{\text{cell center}}} = \frac{\sqrt{\text{Face area}} \times U_\tau}{\nu}$$



# Post-processing RANS and LES simulations

## Wall distance units $x^+$ and $z^+$

- Surface representation of  $\Delta x^+$  and  $\Delta z^+$ .



- Typical requirements for LES simulations are (these are recommendations based on different references):

$$\Delta x^+ < 50, \Delta z^+ < 50 \quad \text{for} \quad y^+ < 5$$

Wall resolving

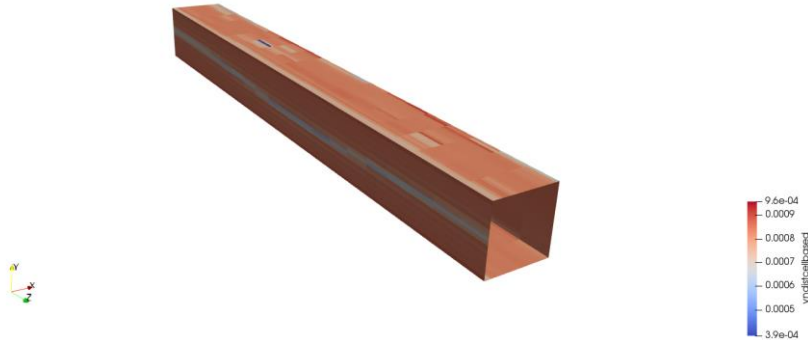
$$\Delta x^+ < 4\Delta y^+, \Delta z^+ < 4\Delta y^+ \quad \text{for} \quad 30 \leq y^+ \leq 300$$

Wall modeling

# Post-processing RANS and LES simulations

## Wall distance units $x^+$ and $z^+$

- Surface representation of  $y_{\text{cell center}}$  or the distance normal from the surface to the first cell center,



- If you used the previous equation for computing  $y_{\text{cell center}}$ , this field is based on the wall shear stresses.
- Therefore, even if the mesh is uniform, the surface representation is not uniform.
- If the information of the first cell center normal to the wall is available (geometrical information), it might be a better approximation.

# Post-processing RANS and LES simulations

## Integral length scales and grid length scales

- The integral length scales and grid length scales are computed as previously discussed, and their values are plotted in the domain.
- These metrics are usually computed from a precursor RANS simulation. We use these metrics to identify where to put more cells.
- For accurate LES simulations, the integral length scales must be sufficiently resolved.
- In RANS simulations, it is enough to use 5 cells across integral length scales.
- LES simulations have higher requirements (about 10 cells across integral length scales).

IO  
0.0e+00 0.025 0.05 0.075 1.0e-01



Integral length scales

RL  
0.0e+00 1.25 2.5 3.75 5.0e+00

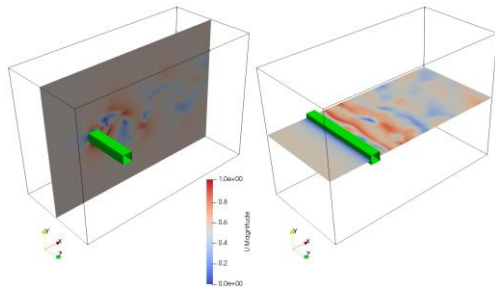


Ratio of integral length scales to grid length scales

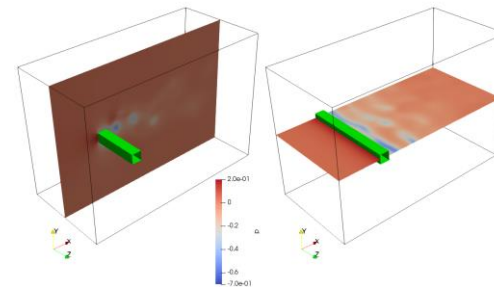
# Post-processing RANS and LES simulations

## Mean values

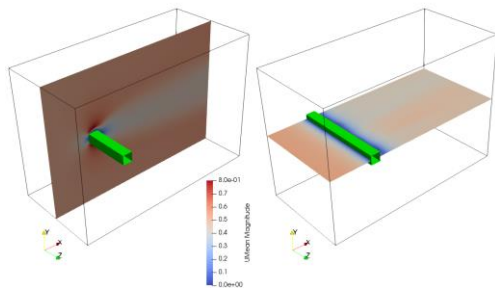
- For most engineering applications, we are interested in computing the mean quantities.
- When running unsteady simulations, do not forget to compute the mean values of the quantities of interest (QoI).



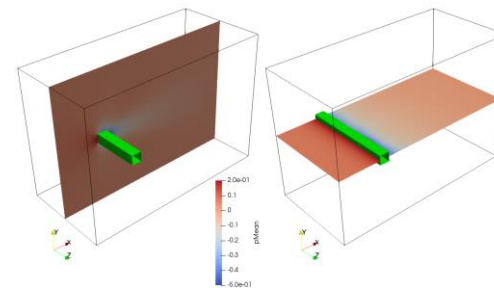
Instantaneous velocity



Instantaneous pressure



Mean velocity



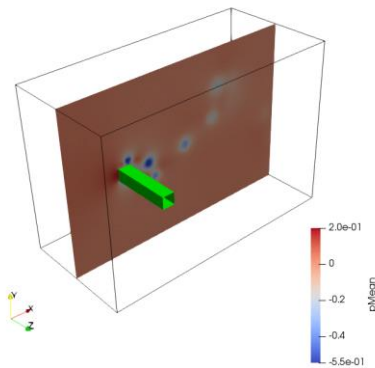
Mean pressure

# Post-processing RANS and LES simulations

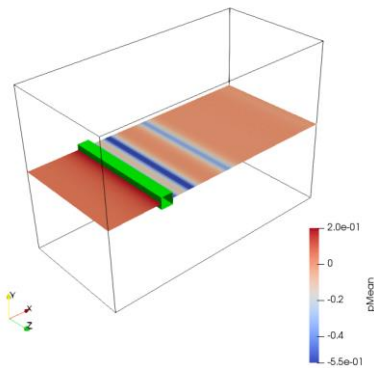
## Mean values

- You can also use the mean values to determine when to stop the simulation (unsteady simulations).
- When the mean quantities stop changing (qualitative and quantitatively), you can say that you have reach a statistically steady state.

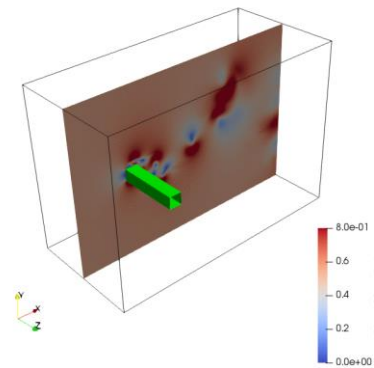
Time: 2.000000



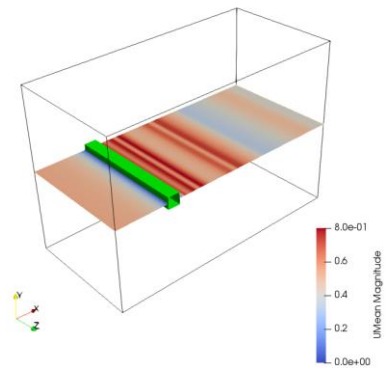
Mean pressure field – Qualitative check



Time: 2.000000



Mean velocity field – Qualitative check

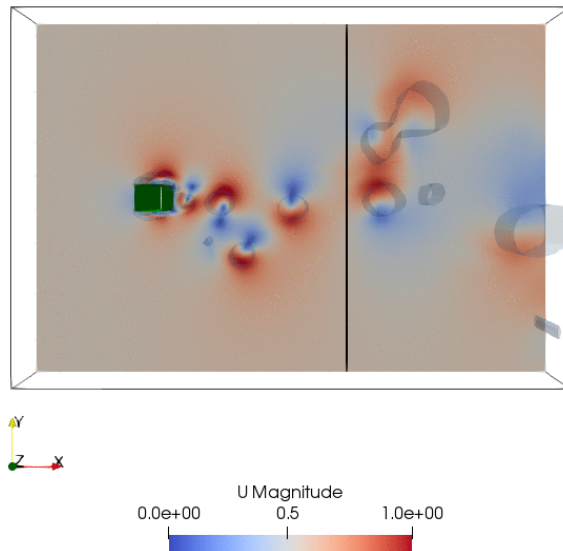


# Post-processing RANS and LES simulations

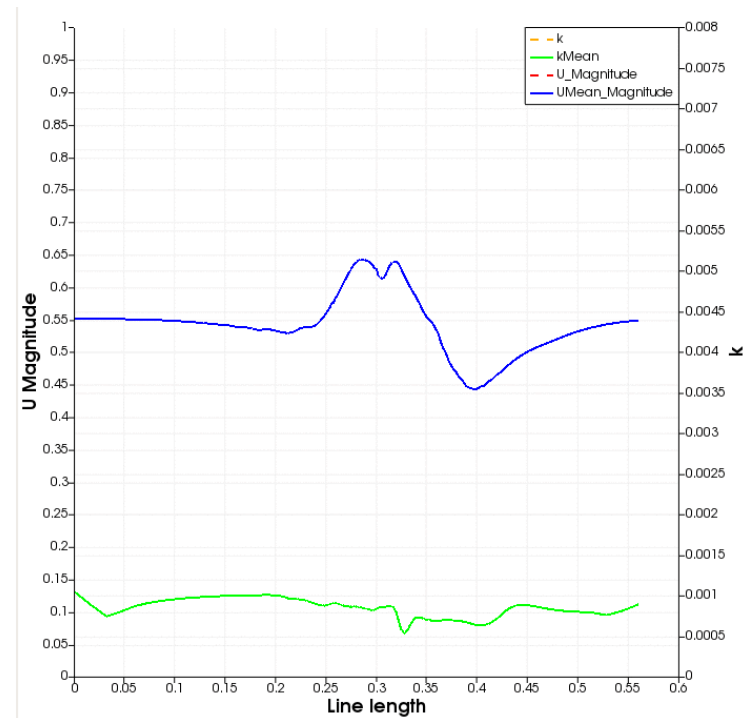
## Mean values and sampling of local quantities

- We can also do local sampling of field variables in the domain and monitor their instantaneous and averaged behaviors.
- When the mean quantities stop changing (qualitative and quantitatively), you can say that you have reach a statistically steady state.

Time: 2.000000



Instantaneous velocity – Qualitative check



Sampled QoI along a line – Quantitative check

# Post-processing RANS and LES simulations

## Pope criterion

- A good LES simulation aims at resolving 80% of the turbulent energy spectrum.
- A way to measure the quality of a LES simulation is by using the Pope criterion [1, 2], which is a simple measure of the fraction of the turbulent kinetic energy in the resolved motions. This criterion is defined as follows,

$$M(\mathbf{x}, t) = \frac{k_{SGS}(\mathbf{x}, t)}{k_{SGS}(\mathbf{x}, t) + k_{RES}(\mathbf{x}, t)} \quad \text{where} \quad k_{RES} = 0.5 (u'^2 + v'^2 + w'^2)$$

where  $k_{SGS}$  is the turbulent kinetic energy of the SGS eddies and  $k_{RES}$  is the kinetic energy of the resolved eddies (determined by the grid size).

- To compute M, a methodology to find  $k_{SGS}$  is required. Therefore, the use of one equation kinetic energy transport models is recommended.
- If you are using the Smagorinsky model, you can approximate  $k_{SGS}$  as follows,

$$k_{SGS} = \frac{(C_s \Delta |\tilde{\mathbf{S}}|)^2}{0.3} = \frac{1}{0.3} \left( \frac{\nu_t}{C_s \Delta} \right)^2$$

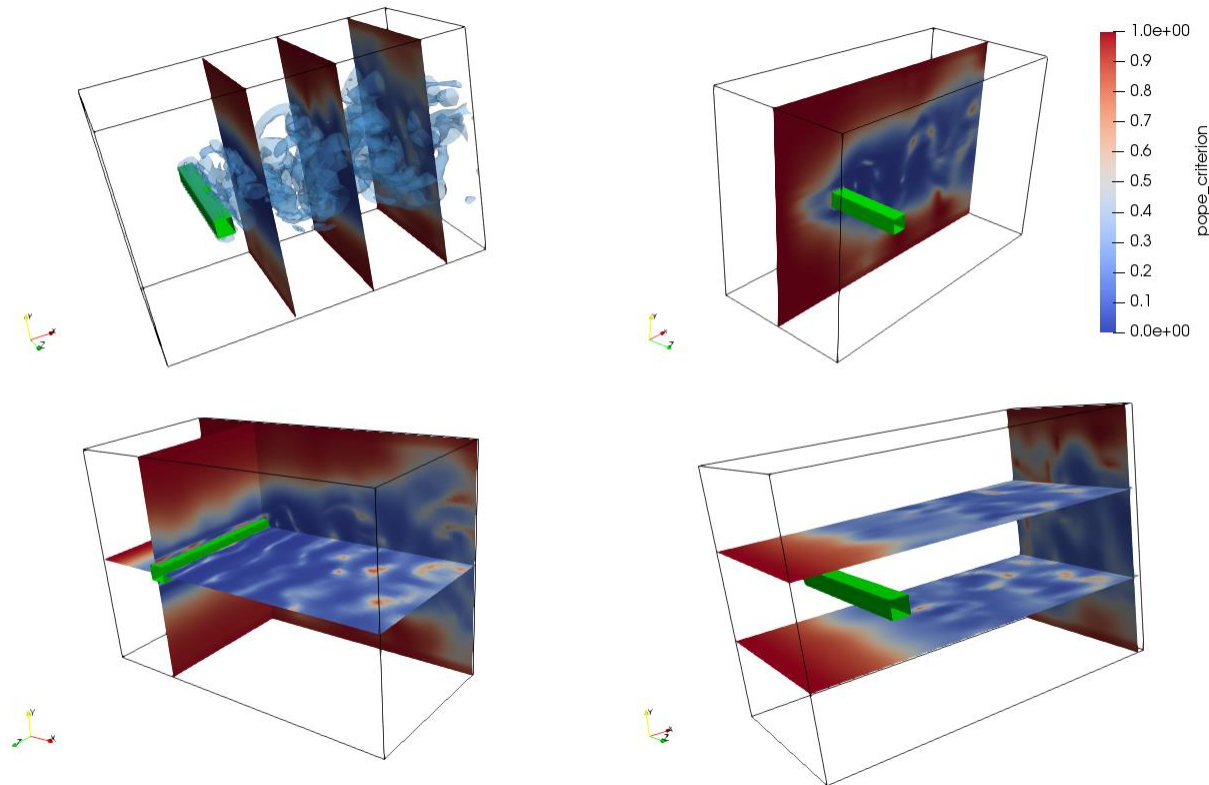
[1] S. Pope. Turbulent Flows. Cambridge University Press. 2014.

[2] S. Pope. Ten questions concerning the large-eddy simulation of turbulent flows. New J. of Physics. 2004.

# Post-processing RANS and LES simulations

## Pope criterion

- The values of  $M$  are between 0 and 1. A value of  $M = 0$  corresponds to a DNS simulation, and a value of  $M = 1$  corresponds to a RANS simulation.
- The Pope criterion can be visualized by plotting it in several cut planes of the domain.





# Post-processing RANS and LES simulations

## Celik LES index quality

- Another method to determine the quality of a LES simulation, is the LES index quality proposed by Celik [1].
- This metric is defined as follows:

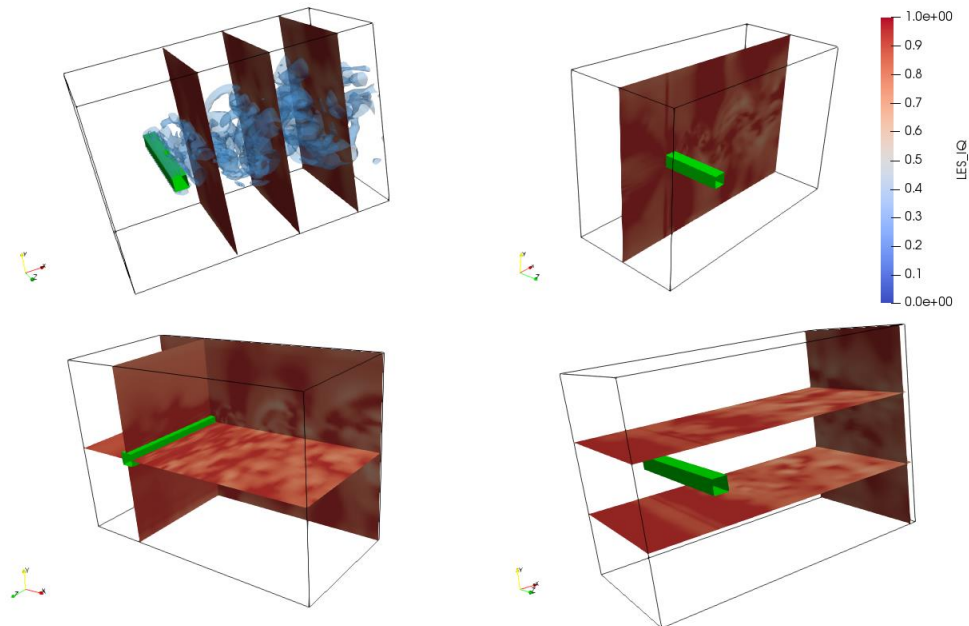
$$LESIQ = \frac{1}{1 + 0.05 \left[ \frac{(\nu + \nu_{SGS})}{\nu} \right]^{0.53}}$$

- This metric is a good alternative to the Pope criterion if you do not have reliable information regarding the modelled turbulent kinetic energy.
  - Which usually is the case when using the Smagorinsky or WALE models.

# Post-processing RANS and LES simulations

## Celik LES index quality

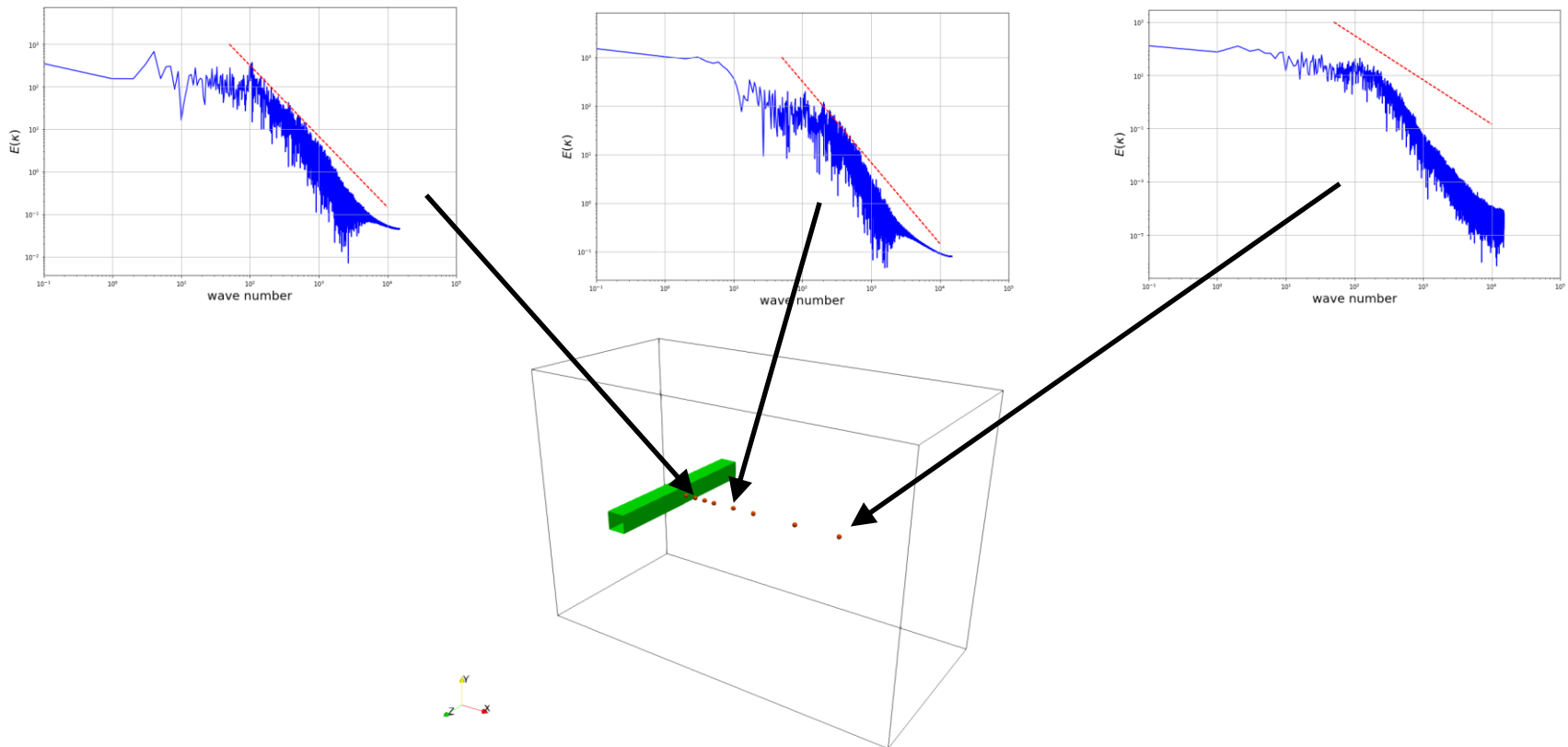
- When using the LES index quality proposed by Celik [1], the metric can be interpreted as follows:
  - The LESIQ is defined between 0 and 1 (but it can be higher than one).
  - The higher the value of LESIQ, the better the resolution.
  - Recommended values are 0.8 and above.
  - This metric can be visualized by plotting it in several cut planes of the domain.



# Post-processing RANS and LES simulations

## Turbulent power spectrum

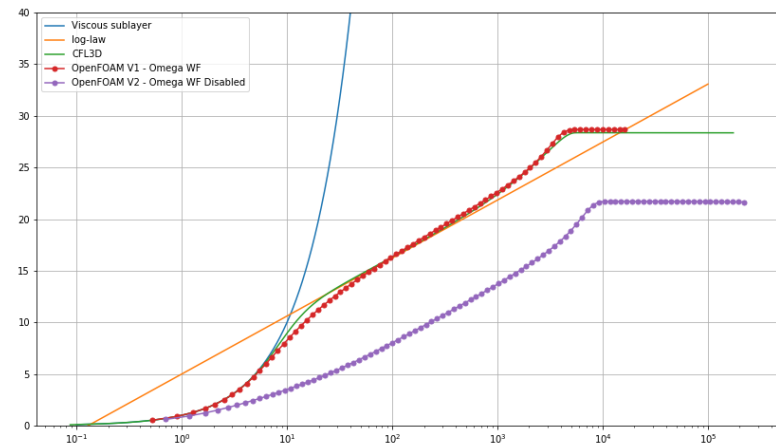
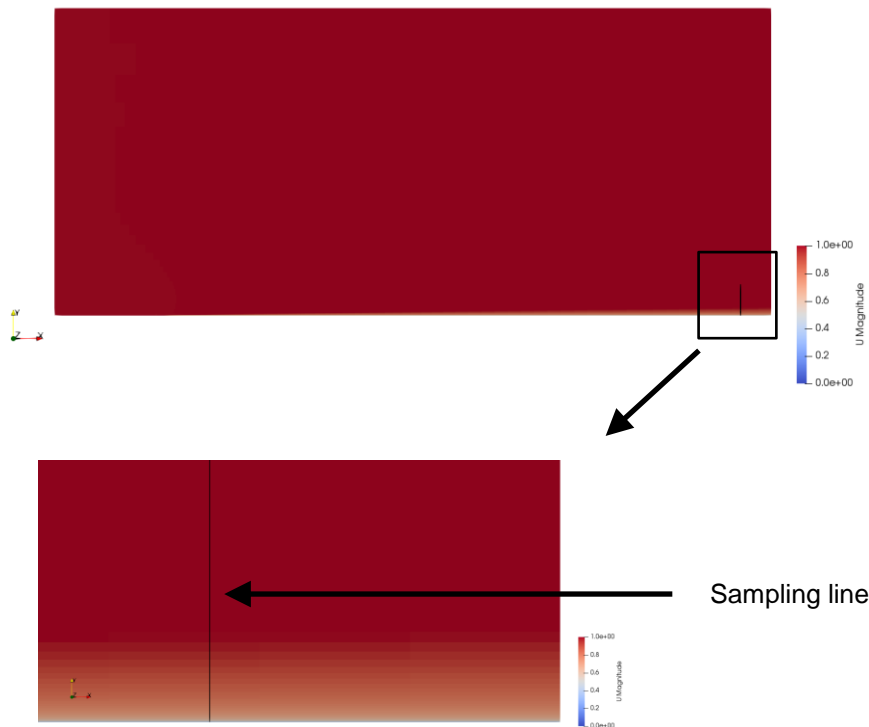
- The turbulent power spectrum represents the distribution of the kinetic energy across the various length scales.
- It is a direct indication of how energy is dissipated with eddies size.
- These plots are local and are obtained by sampling in time the kinetic energy in various locations of the domain (a lot of data needs to be gathered) and signal processing methods.



# Post-processing RANS and LES simulations

## Plot of the law-of-the wall

- We can plot the law-of-the wall in any arbitrary sampling line.
- We know that 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.
- By plotting the velocity in terms of the non-dimensional variables  $u^+$  and  $y^+$ , we can compare the profiles obtained from the simulations with the theoretical profiles.

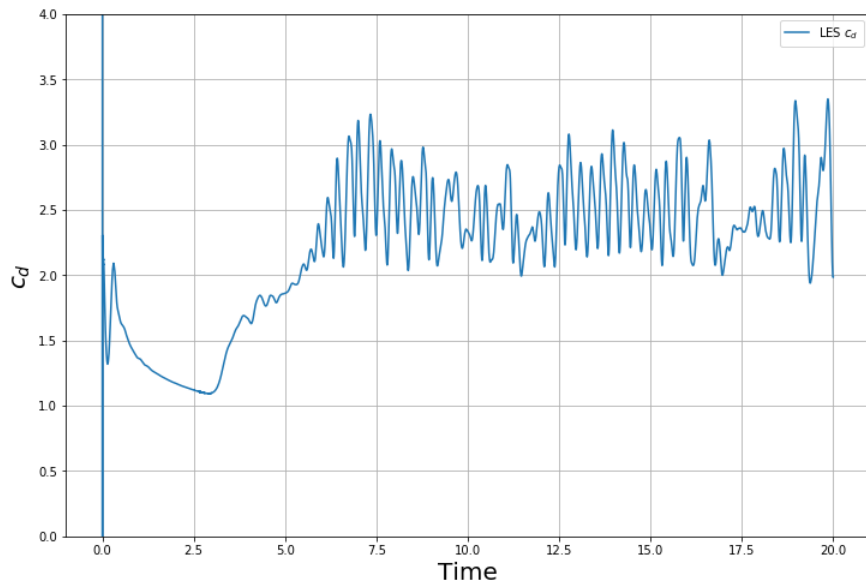


- This is hardcore validation, and it is done for very academic cases or when it is requested by the application.
- For industrial cases, most of the times you do not need to do it, and if you do it, be sure to do the sampling in a region where the flow is attached and far from recirculation areas or other walls.

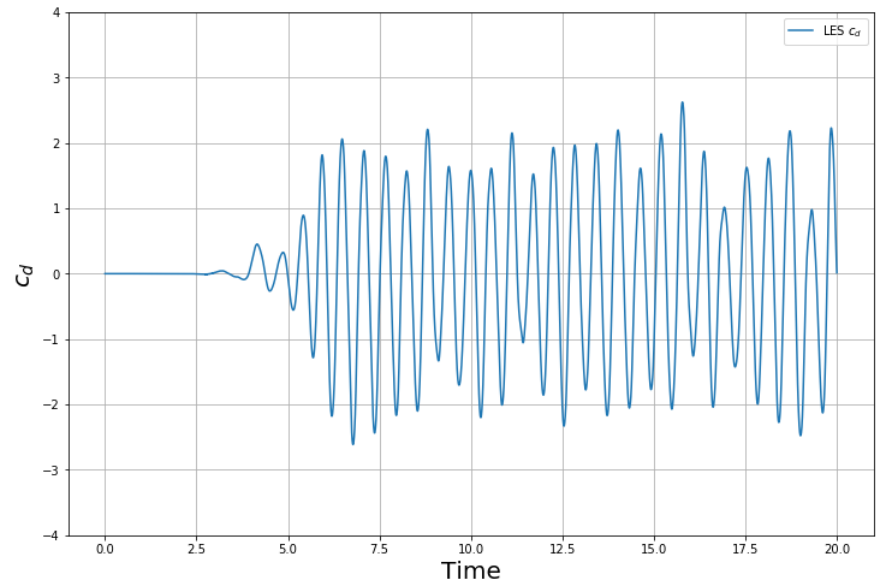
# Post-processing RANS and LES simulations

## Sampling of integral quantities

- It is important to compute integral quantities when running turbulent simulations.
- You can sample the integral quantities in time and compute the descriptive statistics of the signal.
- Many integral quantities can be sampled, such as, mass flow, heat transfer rate, shear stresses, friction coefficient and so on.



Mean value	2.495
Standard deviation	0.286
Variance	0.0822
RMS	2.512



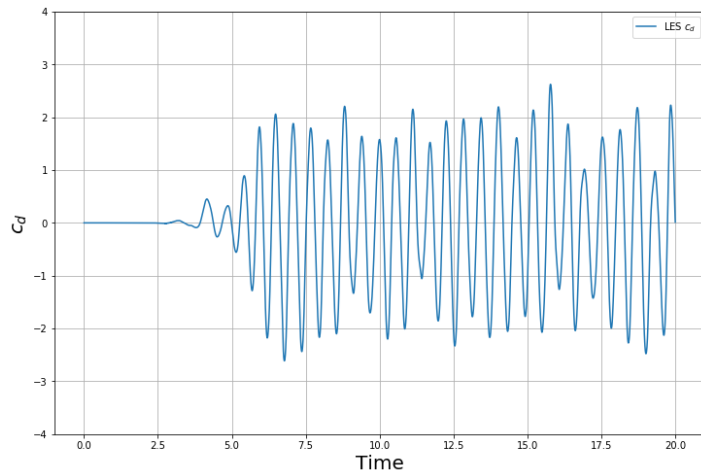
Mean value	-0.010
Standard deviation	1.355
Variance	1.837
RMS	1.355

# Post-processing RANS and LES simulations

## Sampling of integral quantities – Dominant frequency

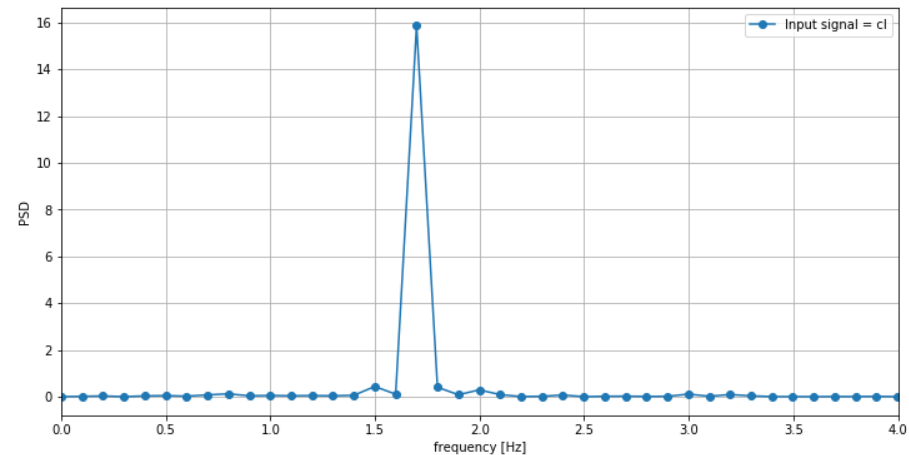
- In many unsteady simulations there is vortex shedding.
- The shedding frequency can be computed from the time signal of a sampled integral quantity (e.g., forces).
- The dominant frequency can be computed using signal processing methods (e.g. periodogram).

Input signal



Signal processing

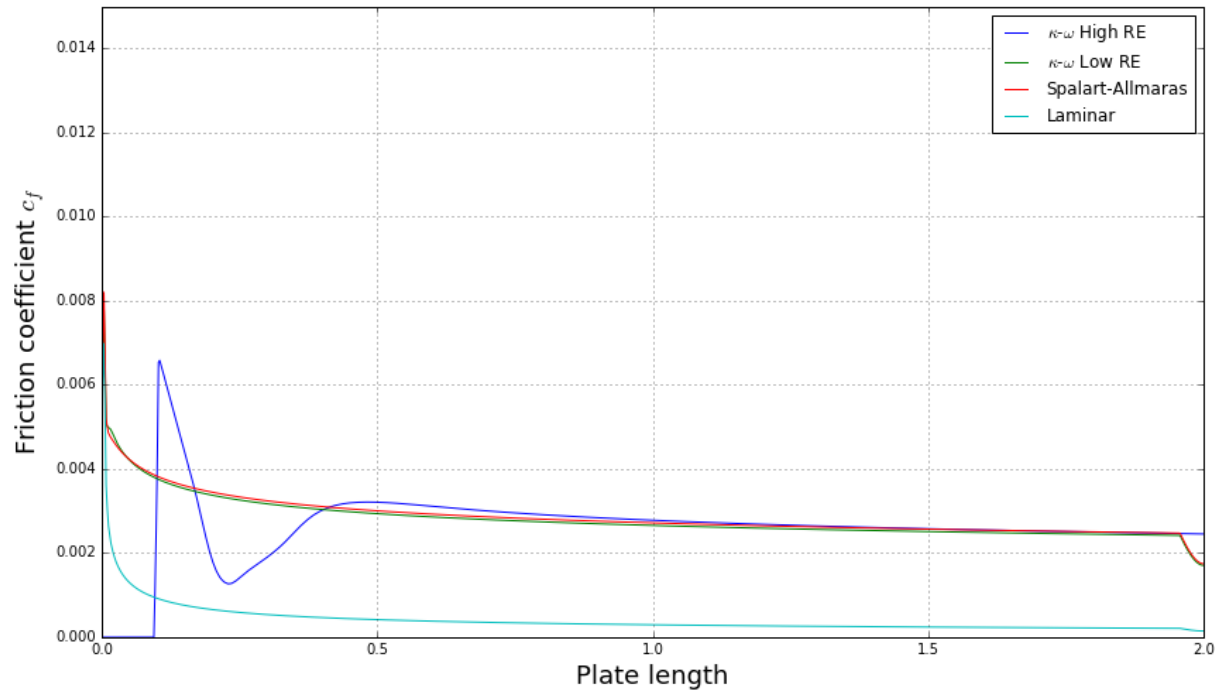
Power spectral density of the input signal



# Post-processing RANS and LES simulations

## Plot of local quantities at the walls

- We can also compute local quantities at the walls and plot its behavior.
- Quantities that can be computed: friction coefficient, shear stresses,  $y^+$ , pressure distribution, pressure coefficient, temperature distribution, and on.



Friction coefficient plot along a surface – Comparison with other numerical results and empirical correlations.

# Roadmap

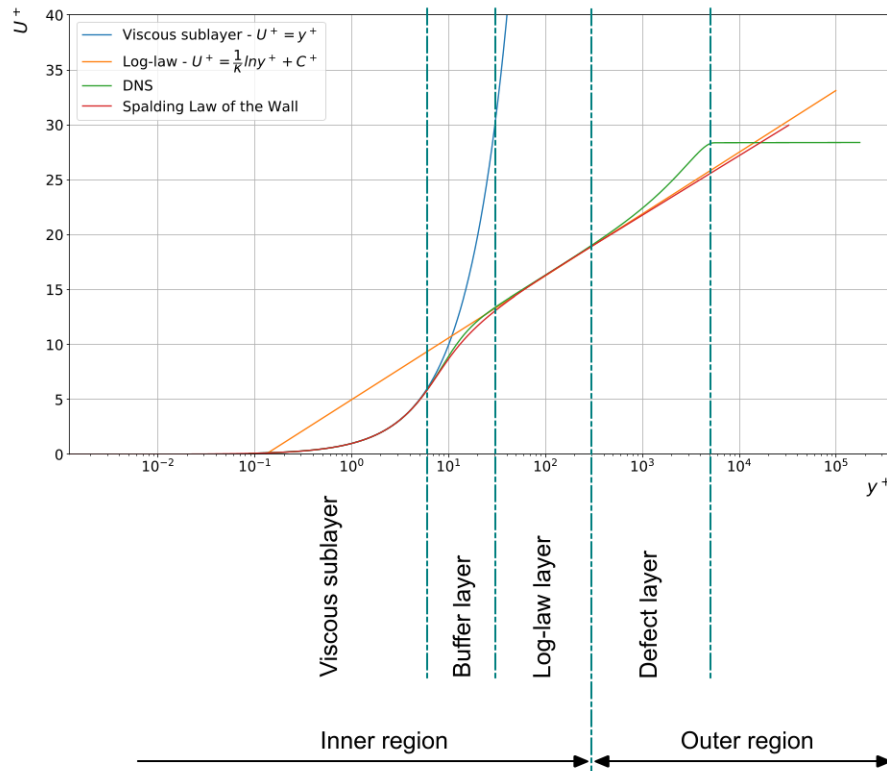
- ~~1. The turbulent world around us~~
- ~~2. Turbulence, does it matter?~~
- ~~3. Introduction to turbulence modeling~~
- ~~4. Post-processing turbulence simulations~~
- 5.  $y^+$  and wall distance units**
- ~~6. Turbulence modeling in OpenFOAM~~
- ~~7. Some applications using OpenFOAM~~
- ~~8. Final remarks – General guidelines~~



# $y^+$ and wall distance units

## 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.
- Do not forget this definition as it is used a lot in turbulence modeling.



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

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

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

Where  $y$  is the distance normal to the wall,  $U_\tau$  is the shear velocity, and  $u^+$  relates the mean velocity to the shear velocity

# $y^+$ and wall distance units

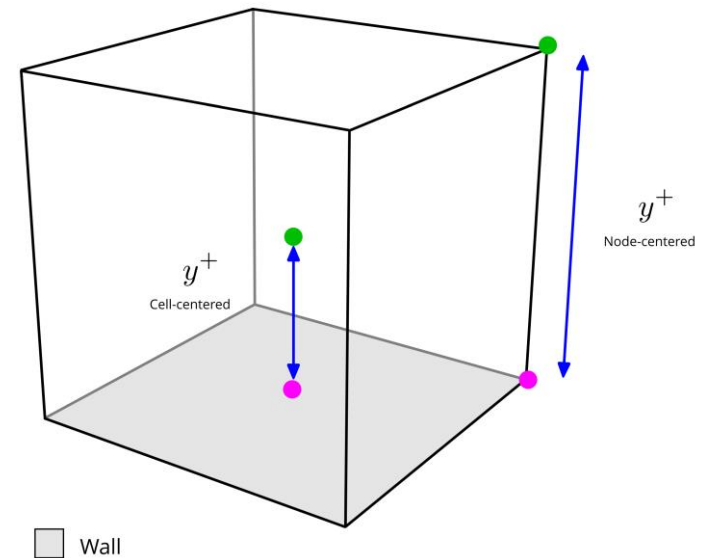
## $y^+$ wall distance units normal to the wall

- $y^+$  is very important quantity in wall bounded turbulence modeling.
- We can use  $y^+$  to estimate the mesh resolution near the wall before running the simulation.
  - Have in mind that we do not know a-priori the wall shear stresses at the walls.
  - Therefore, we need to use correlations to get a rough estimate and generate the initial mesh.
  - The initial mesh is then generated according to the chosen near the wall treatment, namely, wall resolving, wall functions, or  $y^+$  insensitive.
  - Then, we run a precursor simulation to validate the mesh or to get a better estimate of  $y^+$  and determine if we need a finer or coarser mesh.
  - It is an iterative process, and it can be very time consuming, as it might require remeshing and rerunning the simulation to satisfy the near the wall treatment.
- It is quite difficult to get a uniform  $y^+$  value at the walls.
- Try to get a  $y^+$  mean value as close as possible to your target value.
- Also, check that you do not get very high maximum values of  $y^+$  (more than a 1000)
- Values of  $y^+$  up to 300 are fine.
- $y^+$  values larger than 300 and up to a 1000 are acceptable if they do not cover a large surface (no more than 10% of the total wall area), or if they are not located in critical zones.
- Use common sense when accessing  $y^+$  value.

# $y^+$ and wall distance units

## $y^+$ wall distance units normal to the wall

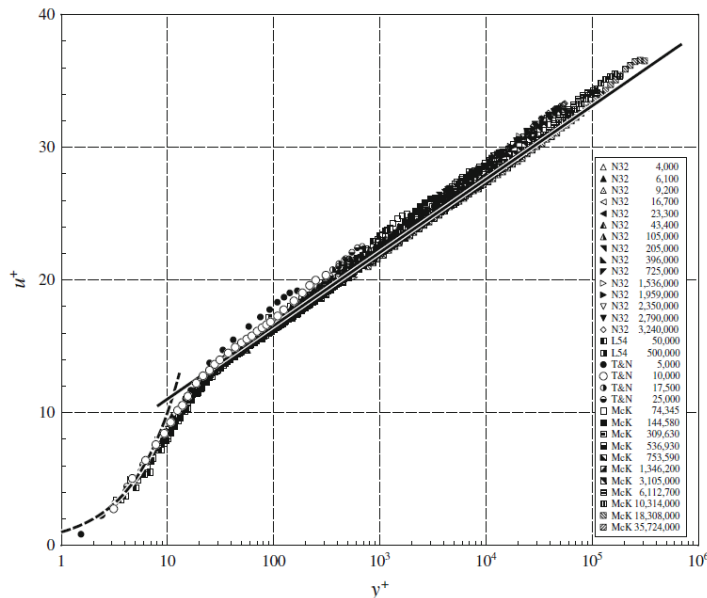
- It is worth mentioning that  $y^+$  can be defined in reference to the cell center or the cell nodes.
- From now on, we are going to assume that we are using a cell-centered solver.
  - Ansys Fluent, StarCCM+, OpenFOAM, code Saturne, SU2, are all cell-centered.
  - It is up to you to check with the solver's documentation to find out with what kind of formulation are you using.
- By the way,  $y^+$  is computed in the same way disregarding of the cell type used.
  - In the cell-centered formulation you need to know the location of the cell center.
  - In the node-centered formulation you need to know the location of the reference node. Usually, the node closest to the surface.



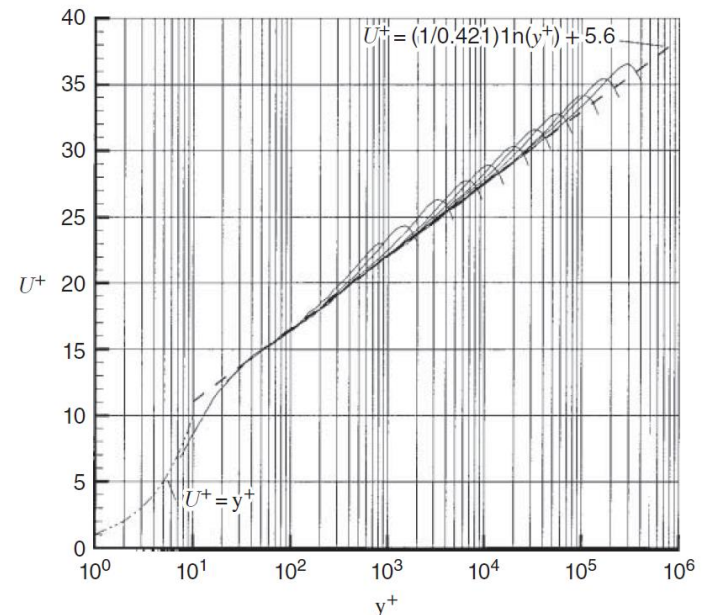
# $y^+$ and wall distance units

## $y^+$ wall distance units normal to the wall

- Most of the times you will find in the literature a  $y^+$  upper limit of 300 for the applicability of wall functions.
- In reality, this upper limit depends on the Reynolds number, as shown in the figure below. For high Reynolds number you can reach large  $y^+$  values.
- However, if the  $y^+$  upper limit is below or close to 100, you should not use wall functions



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 [1].



Mean velocity profiles in pipe flow showing the collective approach to a log law. The curves are for Reynolds numbers between  $Re = 31 \times 10^3$  and  $Re = 18 \times 10^6$  [2].

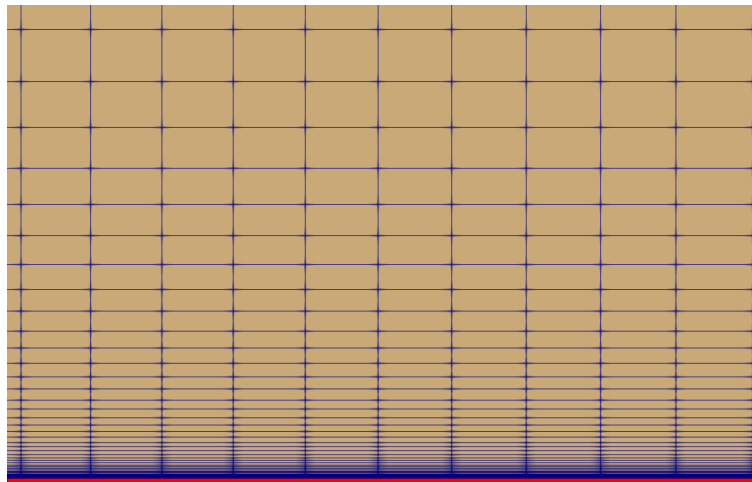
[1] F. Nieuwstadt, B. Boersma, J. Westerweel. Turbulence. Introduction to Theory and Applications of Turbulent Flows. Springer. 2016.

[2] B. McKeon, J. Li, W. Jiang, J. Morrison, A. Smits. Further observations on the mean velocity distribution in fully developed pipe flow. 2004.

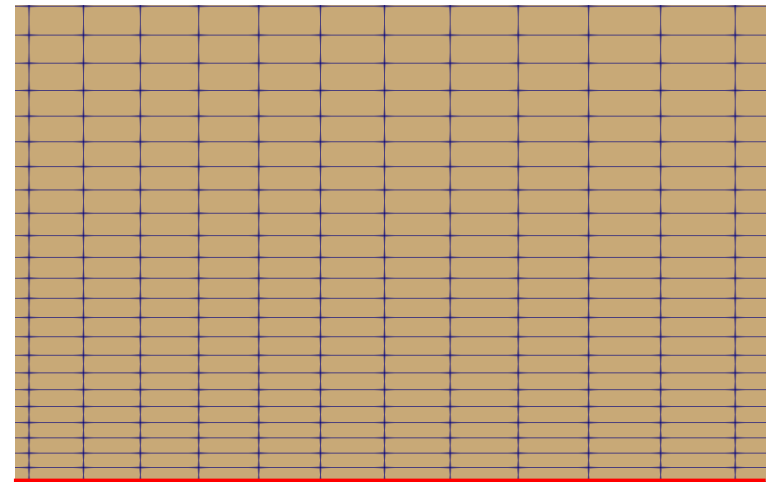
# $y^+$ and wall distance units

## $y^+$ wall distance units normal to the wall

- At meshing time and estimate the normal distance from the wall to the first cell center ( $y$ ), we use the well-known  $y^+$  definition.
- Where we set a target  $y^+$  value and then we solve for the quantity  $y$ .
  - If you choose a low target  $y^+$  (e.g., less than 10), you will have a mesh that is clustered towards the wall (small value of  $y$ ).
  - If you choose a large  $y^+$  value (e.g., more than 100), you will have a coarse mesh close to the walls (large value of  $y$ ).



Fine mesh towards the walls



Coarse mesh towards the walls

# $y^+$ and wall distance units

## Estimation of $y^+$ and normal wall distance

- At meshing time, to estimate the normal distance from the wall to the first cell center, we use the well-known  $y^+$  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?
  - We can use correlations, as described in the next slide.

# $y^+$ and wall distance units

## Estimation of $y^+$ and normal wall distance

- At meshing time, to estimate the normal distance from the wall to the first cell center, you can proceed as follows,

- $Re = \frac{\rho \times U \times L}{\mu}$   Compute the Reynolds number using the characteristic length of the problem.
- $C_f = 0.058 \times Re^{-0.2}$   Compute the friction coefficient using any of the correlations available in the literature. There are many correlations available ranging from pipes to flat plates, for smooth and rough surfaces.  
This correlation corresponds to a smooth flat plate case, ideal for external aerodynamics.
- $\tau_w = \frac{1}{2} \times C_f \times \rho \times U_\infty^2$   Compute the wall shear stresses using the friction coefficient computed in the previous step.
- $U_\tau = \sqrt{\frac{\tau_w}{\rho}}$   Compute the shear velocity using the wall shear stresses computed in the previous step.
- $y = \frac{\mu \times y^+}{\rho \times U_\tau}$   Set a target  $y^+$  value and solve for  $y$  using the flow properties and previous estimates.  
This value represents the distance from the wall to the first cell center.  
At this point, you can generate your mesh trying to respect this value.

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

# $y^+$ and wall distance units

## Estimating normal wall distance – Meshing requirements

- Remember, the thickness of the thermal sublayer for a high Prandtl number fluid (e.g., water) is much less than the momentum sublayer thickness.
- Therefore, the mesh requirements close to the walls need to be corrected for the thinner thermal boundary layer.
- The  $y^+$  value can be corrected as follows,

$$y^+ \approx \frac{1}{\sqrt{Pr}}$$

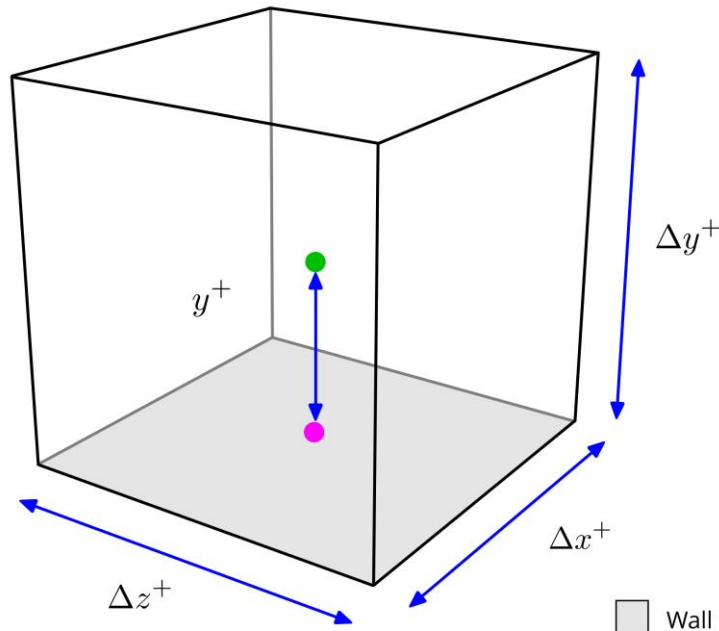
- A similar situation exists when working with multispecies transport and using the Schmidt number ( $Sc$ ).
- If the Schmidt number is considerably larger than unity, then the thickness of the thermal diffusion sublayer is much less than the momentum sublayer thickness.
- In this situation, the  $y^+$  value can be corrected as follows,

$$y^+ \approx \frac{1}{\sqrt[3]{Sc}}$$



# $y^+$ and wall distance units

## Wall distance units $x^+ - y^+ - z^+$ and some rough estimates



- Similar to  $y^+$ , the wall distance units can be computed in the stream-wise ( $\Delta x^+$ ) and span-wise ( $\Delta z^+$ ) 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 (in cell-centered solvers),

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

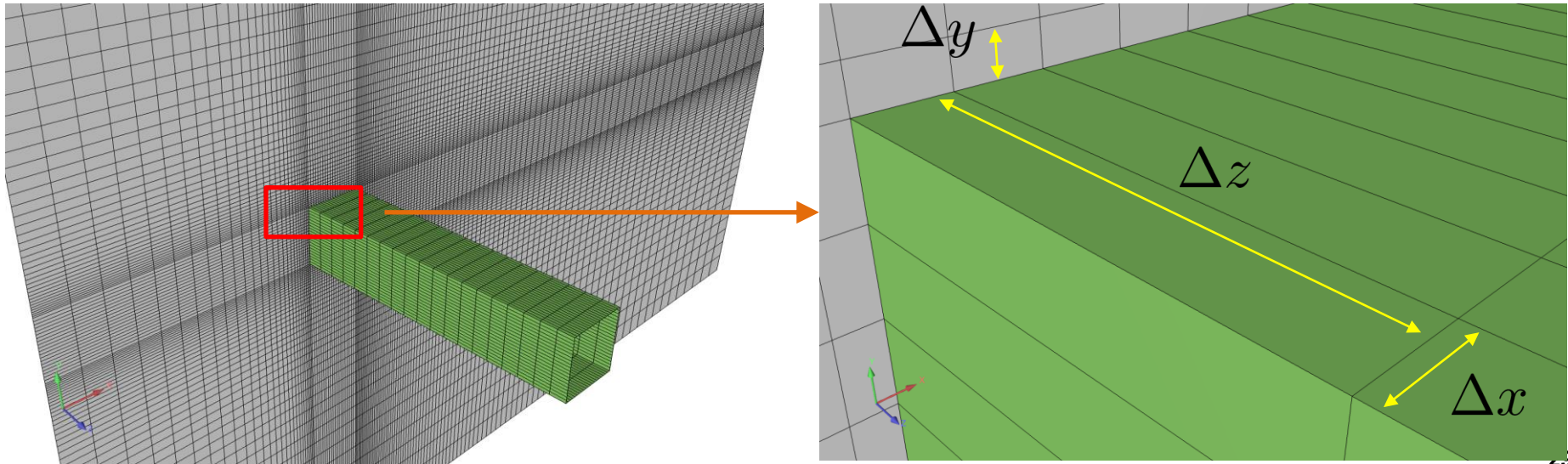
$$(\Delta x^+, \Delta y^+, \Delta z^+) = \left( \frac{x}{l_\tau}, \frac{y}{l_\tau}, \frac{z}{l_\tau} \right)$$

where  $l_\tau = \frac{\nu}{U_\tau}$   
Viscous length

# $y^+$ and wall distance units

## Wall distance units $x^+$ – $y^+$ – $z^+$ and some rough estimates

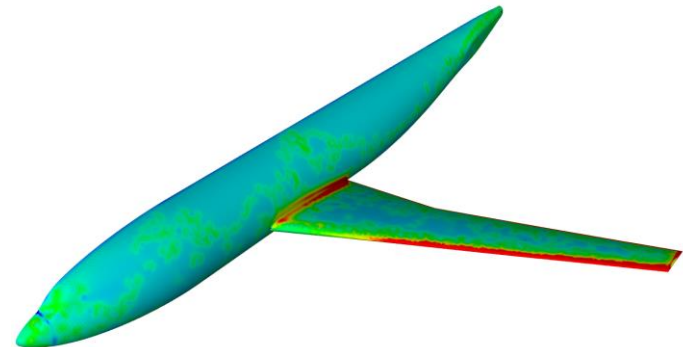
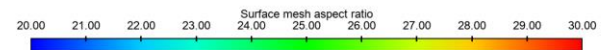
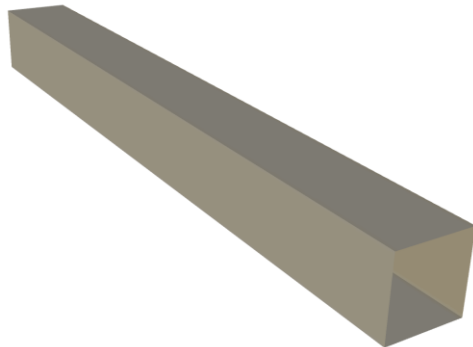
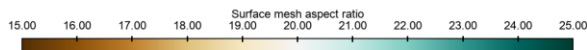
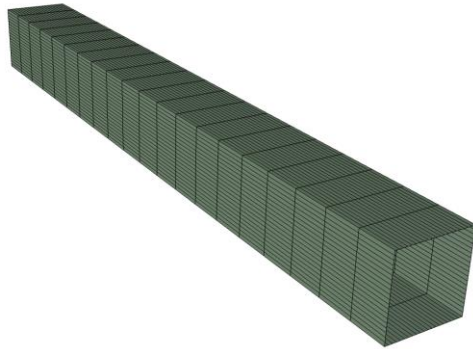
- It is not only about the  $y^+$  value.
- In SRS simulations  $x^+$  and  $z^+$  are also important.
- The  $x^+$  and  $z^+$  values strongly depend on the surface mesh aspect ratio.
- The aspect ratio is defined as the ratio between the longest edge and shortest edge.
- Recommended values of surface mesh aspect ratio for SRS simulations are 10 or less.
- However, have in mind that this is a rough estimate based on experience and good practices.
- Meshing applications and solvers usually have adequate tools for mesh diagnostics.
- Notice that this metric is entirely based on geometrical information.



# $y^+$ and wall distance units

## Wall distance units $x^+ - y^+ - z^+$

- For quad elements, the surface aspect ratio can be computed easily.
- For other shapes (triangles, polyhedrons, and so on), it is not very easy to compute.
- Surface mesh aspect ratio has a direct implication when computing the wall distance units  $x^+$  and  $z^+$ .



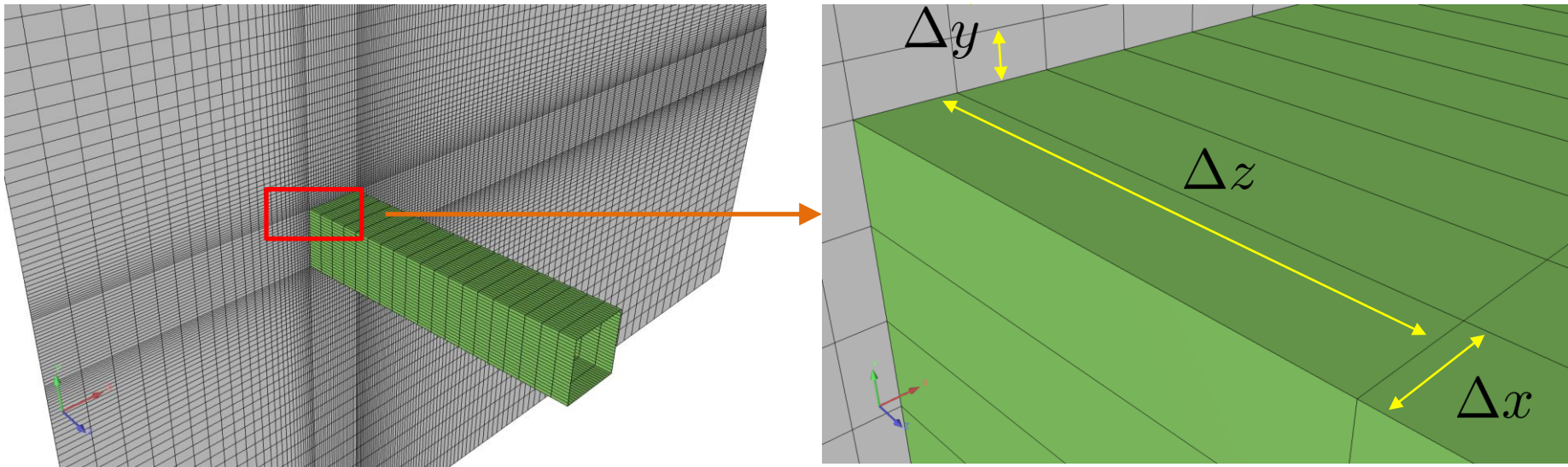
# $y^+$ and wall distance units

## Wall distance units $x^+ - y^+ - z^+$

- Besides the surface aspect ratio, you should also check the ratio between surface cell base length and the cell height. This aspect ratio can be approximated as follows,

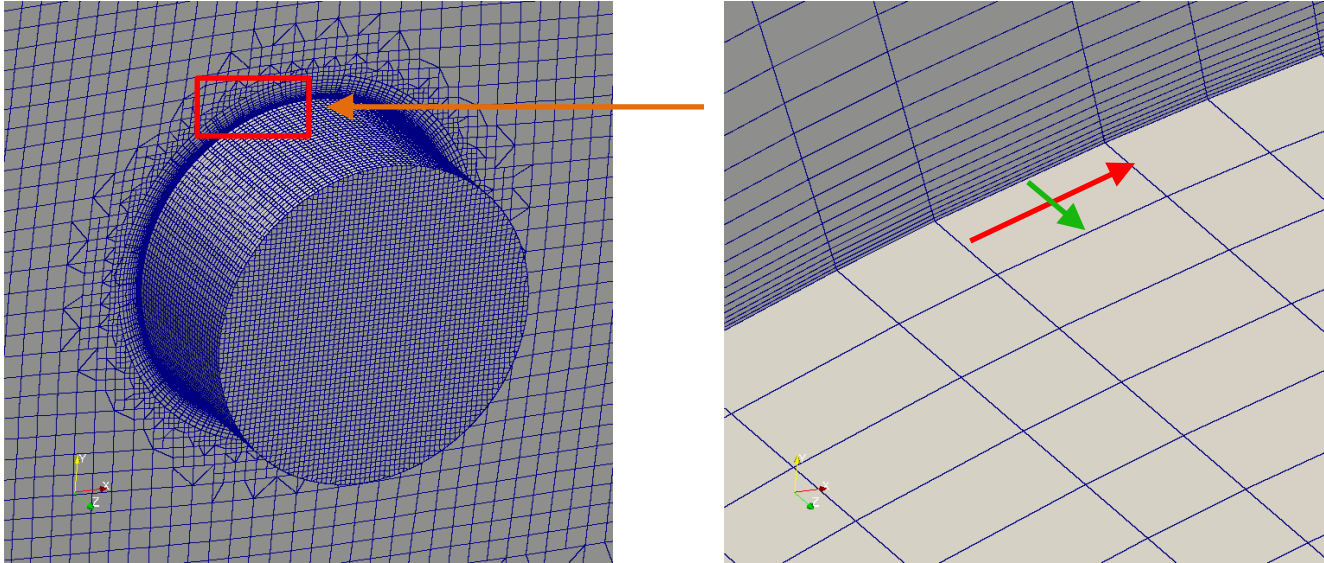
$$AR = \frac{\sqrt{\text{Surface cell area}}}{\Delta y}$$

- Recommended values are 20 or less.
- As you can see, this aspect ratio can impose strong requirements on the mesh.
- Again, this is a rough estimate based on experience, and that should be used when conducting SRS simulations



# $y^+$ and wall distance units

## Wall distance units $x^+ - y^+ - z^+$ and some rough estimates



- Similar to  $y^+$ , the wall distance units can be computed in the span-wise ( $\Delta z^+$ ) and stream-wise ( $\Delta x^+$ ) directions.
- Typical requirements for LES are (these are recommendations based on different references):

$$\Delta x^+ < 50, \Delta z^+ < 50 \quad \text{for} \quad y^+ < 5$$

Wall resolving

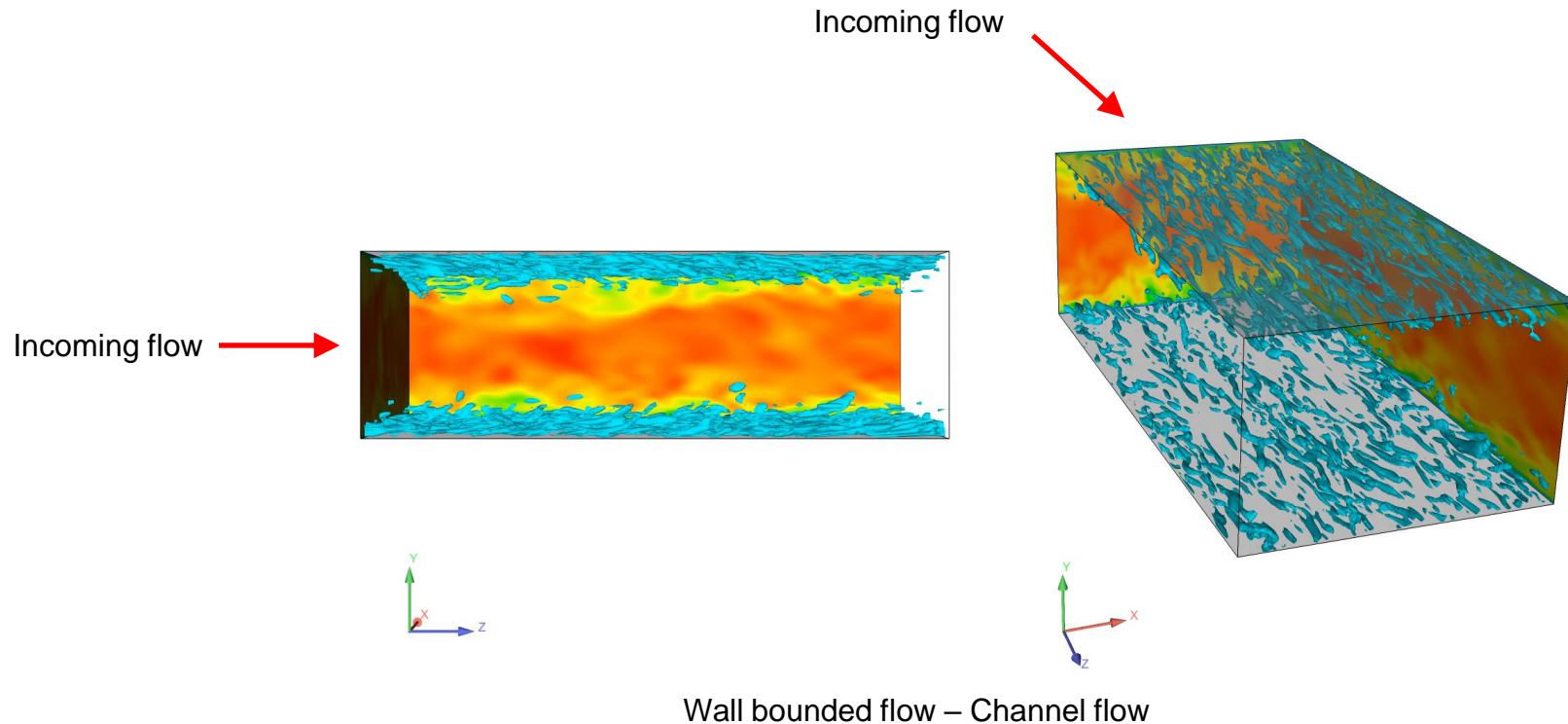
$$\Delta x^+ < 4\Delta y^+, \Delta z^+ < 4\Delta y^+ \quad \text{for} \quad 30 \leq y^+ \leq 300$$

Wall modeling



# $y^+$ and wall distance units

## Wall distance units and some rough estimates



Wall bounded flow – Channel flow

- In wall resolving LES simulations, typical wall distance units requirements to resolve the structures close to the walls (streaks), are as follows,

$$\Delta x^+ < 50, \Delta z^+ < 50 \quad \text{for} \quad y^+ < 5$$

Spanwise direction

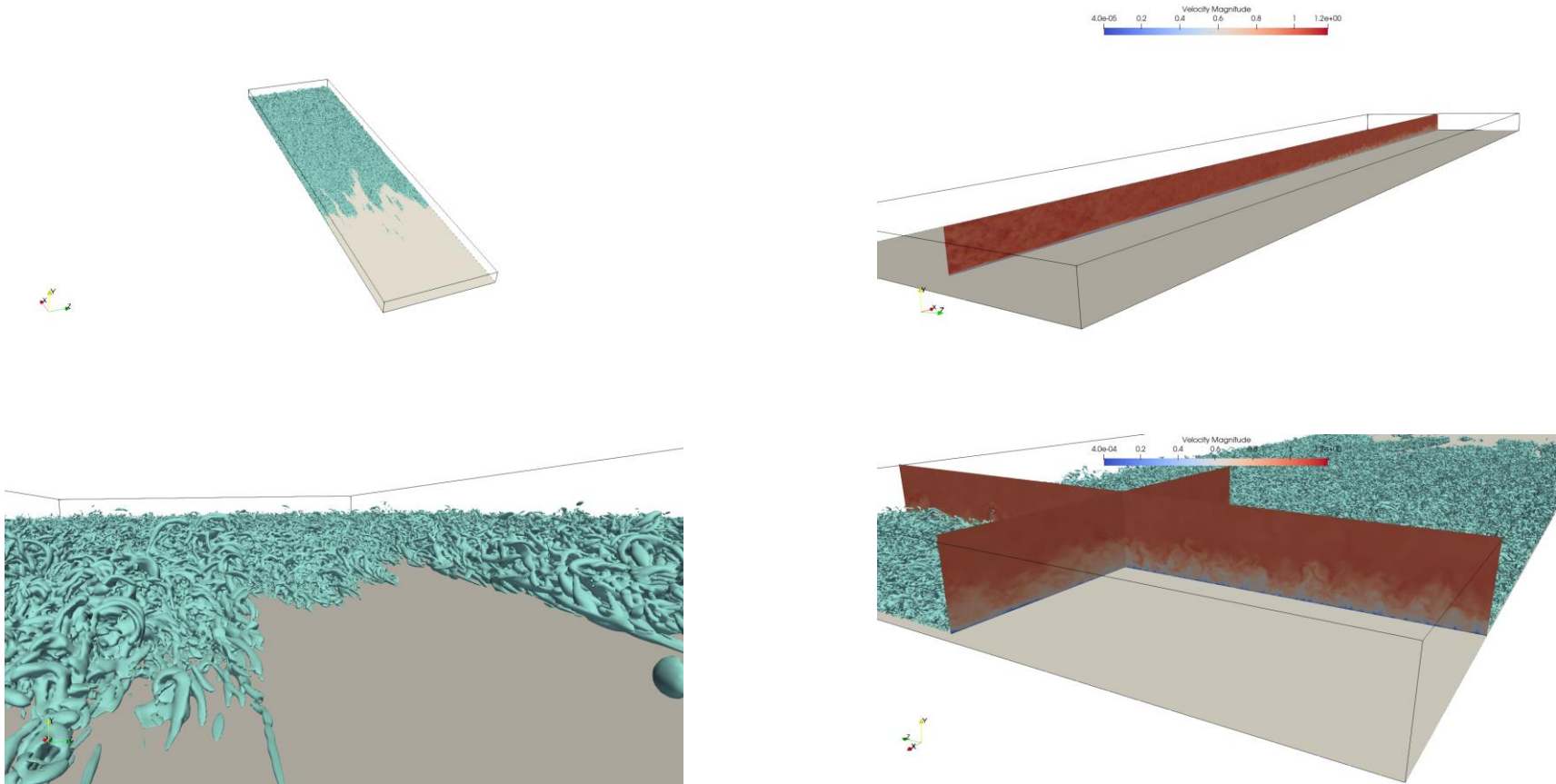
Streamwise direction

Wall normal direction

Wall resolving

# $y^+$ and wall distance units

## Wall distance units and some rough estimates



- In proper resolved DNS, the wall distance units requirements in all directions (spanwise, streamwise and normal) are close or lower than one.

# $y^+$ and wall distance units

## 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 recommended to resolve 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.



# $y^+$ and wall distance units

## 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  $30 < y^+ < 300$
- But values of  $y^+$  up to 1000 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,  $1 < y^+ < 300$ .
- 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  $\Delta x^+$  and  $\Delta z^+$  values as high as 300 times the value of  $\Delta y^+$ .

# $y^+$ and wall distance units

## 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-priori 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  $\Delta x^+$  and  $\Delta z^+$  values as high as 300 the value of  $\Delta y^+$ .

# $y^+$ and wall distance units

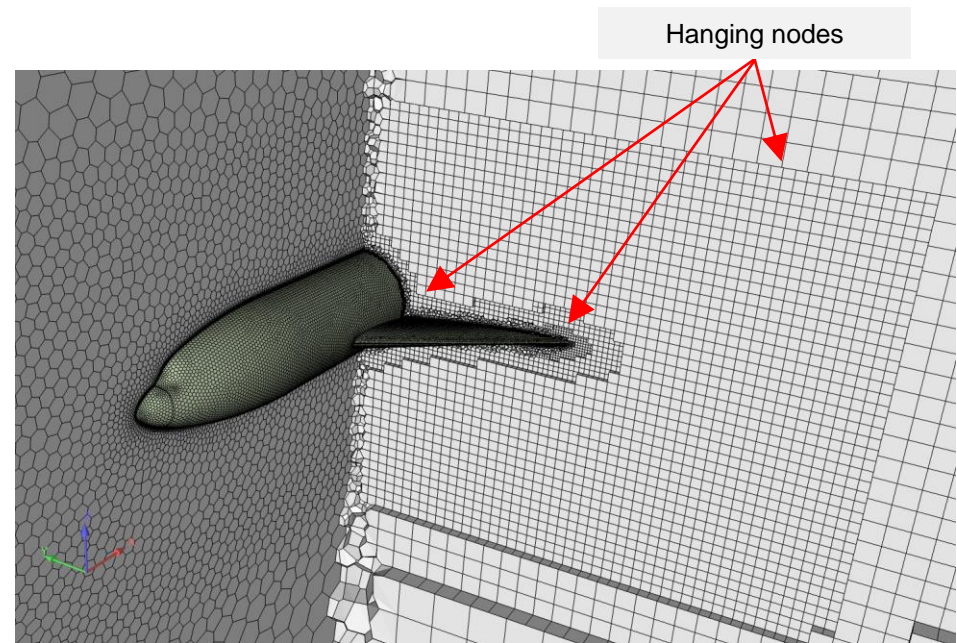
## A few mesh resolution guidelines and rough estimates

- 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,  $\Delta x^+ < 4\Delta y^+$ ,  $\Delta z^+ < 4\Delta y^+$
  - LES simulations are intrinsically 3D.
  - Try to avoid the use of symmetry (axial and planar).
  - Use hexahedral meshes.

# $y^+$ and wall distance units

## A few mesh resolution guidelines and rough estimates

- On meshes with hanging nodes
  - Hanging nodes are often used by meshers and supported by solvers to ease the mesh generation process.
  - In SRS simulations, it is extremely recommended to avoid hanging nodes.
  - Hanging nodes are source of fast transition between small and large cells and can add a lot numerical diffusion to the solution.
  - In particular, avoid hanging nodes close to the walls.



Hexahedral dominant mesh with polyhedral surface mesh  
Hanging nodes allowed – Octree subdivision

**Guided tutorial 4**  
**Turbulence flow over a backward facing  
step**

**Guided tutorial 5**  
**Buice-Eaton 2D Diffuser**

# Roadmap

- ~~1. The turbulent world around us~~
- ~~2. Turbulence, does it matter?~~
- ~~3. Introduction to turbulence modeling~~
- ~~4. Post-processing turbulence simulations~~
- ~~5.  $y^+$  and wall distance units~~
- 6. Turbulence modeling in OpenFOAM**
- ~~7. Some applications using OpenFOAM~~
- ~~8. Final remarks – General guidelines~~

# Turbulence modeling in OpenFOAM

## RANS/URANS Turbulence models available in OpenFOAM

- In OpenFOAM, you will find the following family of RANS turbulence models:
  - LRR
  - LamBremhorstKE (incompressible)
  - LaunderSharmaKE
  - LienCubicKE (incompressible)
  - LienLeschziner (incompressible)
  - SSG
  - ShihQuadraticKE (incompressible)
  - SpalartAllmaras
  - qZeta (incompressible)
  - V2f
  - kEpsilonLopesdaCosta
  - kOmega
  - kOmega2006
  - kOmegaSST
  - kOmegaSSTLM
  - kOmegaSSTSAS
  - kkLOmega (incompressible)
  - kEpsilon
  - realizableKE
  - RNGkEpsilon
  - buoyantKEpsilon (compressible)
- The name of the models is given using OpenFOAM jargon.
- Our task is to choose the less wrong model!

# Turbulence modeling in OpenFOAM

## DES/LES Turbulence models available in OpenFOAM

- In OpenFOAM, you will find the following family of DES/LES turbulence models:

- DeardorffDiffStress
- Smagorinsky
- SpalartAllmarasDDES
- SpalartAllmarasDES
- SpalartAllmarasIDDES\*
- WALE
- dynamicKEqn
- dynamicLagrangian
- kEqn
- kOmegaSSTDES

\* It is an improved of DDES that can be used for WMLES

- And if you are interested in doing laminar simulations (simulations with no turbulence model or what people call DNS), you can use the following model:
  - laminar (DNS)
- The name of the models is given using OpenFOAM jargon.
- Our task is to choose the less wrong model!



# Turbulence modeling in OpenFOAM

## DES/LES Turbulence models available in OpenFOAM

- In you are working with DES/LES turbulence models, you will find the following delta functions:
  - cubeRootVolDelta
  - IDDESDelta
  - maxDeltaxyz
  - PrandtlDelta
  - SmoothDelta
  - vanDriestDelta
- In you are working with LES dynamic turbulence models, you will find the following filter functions:
  - anisotropicFilter
  - laplaceFilter
  - simpleFilter

# Turbulence modeling in OpenFOAM

## Multiphase Turbulence models available in OpenFOAM

- If you are dealing with multiphase flows (dispersed systems), you will find the following family of RANS/URANS turbulence models:
  - continuousGasKEpsilon
  - kOmegaSSTSato
  - LaheyKEpsilon
  - mixtureKEpsilon
- If you are dealing with multiphase flows (dispersed systems), you will find the following family of LES turbulence models:
  - continuousGasKEqn
  - Niceno
  - SmagorinskyZhang
- The name of the models is given using OpenFOAM jargon.
- Our task is to choose the less wrong model!

# Turbulence modeling in OpenFOAM

## Turbulence models available in OpenFOAM

- As you can see, there is a cornucopia, plethora, fullness, abundance of turbulence models implemented in OpenFOAM, from RANS to LES.
- You can also implement yours!
- You can find the turbulence models implementations in the following directories:
  - `$WM_PROJECT_DIR/src/MomentumTransportModels`
- You can find the turbulence models hardwired for multiphase flows (dispersed systems), in the following directories:
  - `$WM_PROJECT_DIR/src/MomentumTransportModels/phaseCompressible/RAS`
  - `$WM_PROJECT_DIR/src/MomentumTransportModels/phaseCompressible/LES`
- If you are dealing with atmospheric boundary layer, you can find specific turbulence models in the following directory:
  - `$WM_PROJECT_DIR/src/atmosphericModels`

# Turbulence modeling in OpenFOAM

## Turbulence models available in OpenFOAM

- The source code of the DES/LES delta functions is located in the following directory:
  - `$WM_PROJECT_DIR/src/MomentumTransportModels/  
momentumTransportModels/LES/LESdeltas`
- The source code of the LES dynamic turbulence models filter functions is located in the following directory:
  - `$WM_PROJECT_DIR/src/MomentumTransportModels/  
momentumTransportModels/LES/LESfilters/`

# Turbulence modeling in OpenFOAM

## Turbulence models available in OpenFOAM

- If you have absolutely no idea of what model to use, we highly recommend you the family models  $k - \omega$  or the realizable  $k - \epsilon$ .
- Remember, when a turbulent flow enters a domain, turbulent boundary conditions and initial conditions must be specified.
- Also, if you are dealing with wall bounded turbulence you will need to choose the near-wall treatment.
- You can choose to solve the viscous sub-layer (low-Re approach) or use wall functions (high-Re approach).
- You will need to give the appropriate boundary conditions to the wall treatment.
- Always define the wall patches as **wall** type in the dictionary file `constant/polyMesh/boundary`, otherwise you will not be able to activate the wall functions (this applies for fixed and moving walls).
- Remember, every turbulence model has different requirements when it comes to choosing the boundary conditions and initial conditions, so read the theory and get familiar with the models.

# Turbulence modeling in OpenFOAM

## Turbulence models available in OpenFOAM

- The initial conditions and boundary conditions change from model to model and depends on the physics involve, so get familiar with the theory.
- Remember, there is no universal turbulence model.
- To set the initial conditions and boundary conditions, our best advise is to find and read the original reference used to implement the turbulence model.
- For instance, if you are interested in the  $k - \omega$  turbulence model, you can go to the directory:
  - `$WM_PROJECT_DIR/src/MomentumTransportModels/  
momentumTransportModels/RAS/kOmega`
- Open the file `kOmega.H`, in the header file you will find the references used to implement the model in OpenFOAM and a short description, in this case:

Standard high Reynolds-number k-omega turbulence model for incompressible and compressible flows.

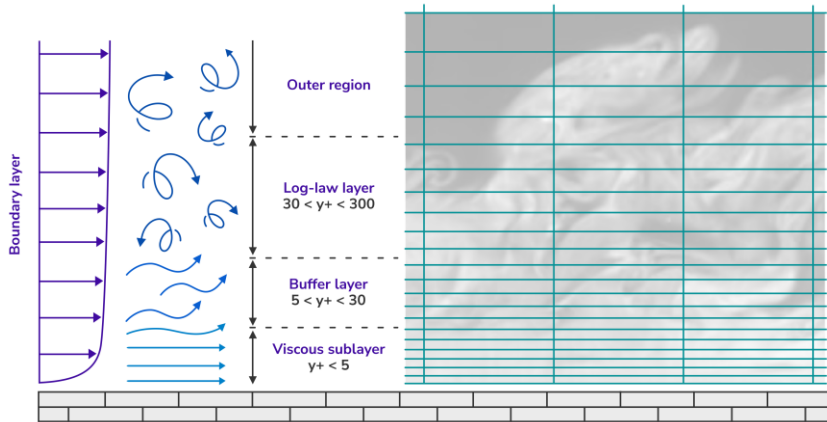
References:

Wilcox, D. C. (1998).  
Turbulence modeling for CFD  
(Vol. 2, pp. 103-217). La Canada, CA: DCW industries.

- Remember, you have the source code so take your time and try to read it.

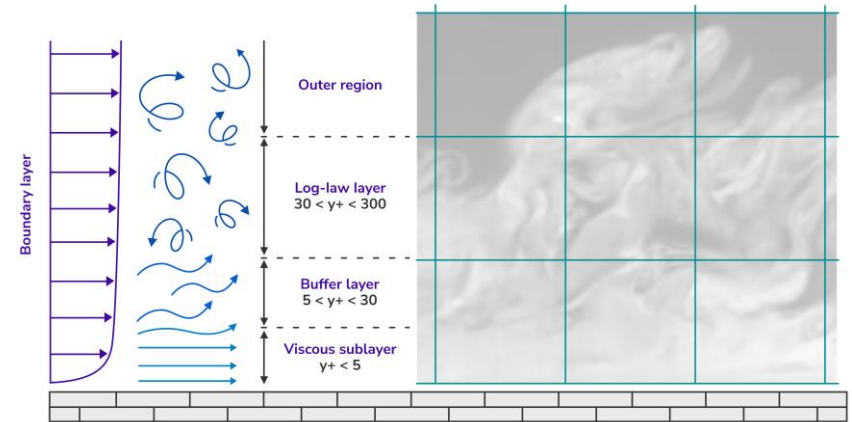
# Turbulence modeling in OpenFOAM

## Near-wall treatment and wall functions



### Wall resolving approach

No wall-functions used  
Also known as Low-RE or LRN



### Wall modeling approach

Wall-functions used  
Also known as High-RE or HRN

- Wall functions is the approach to use if you are more interested in the mixing in the outer region, rather than on a very accurate prediction of the forces on the wall or transition.
- 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.
  - In this case, it is better to take the wall resolving approach.

# Turbulence modeling in OpenFOAM

## Near-wall treatment and wall functions

- To deal with wall bounded turbulence, several wall functions are available in OpenFOAM (wallFunctions),
  - atmBoundaryLayer
  - alphasatWallFunction
  - alphasatJayatilkeWallFunction
  - epsilonWallFunction
  - fWallFunction
  - kqRWallFunction
  - kLowReWallFunction
  - nutWallFunction
  - omegaWallFunction
  - v2WallFunction
- The source code of the wall functions is located in the following directories:
  - `$WM_PROJECT_DIR/src/MomentumTransportModels/  
momentumTransportModels/derivedFvPatchFields/wallFunctions`



# Turbulence modeling in OpenFOAM

## Near-wall treatment and wall functions

- You will find the following turbulent viscosity boundary conditions (nutWallFunctions):
  - nutkRoughWallFunction
  - nutkWallFunction
  - nutLowReWallFunction
  - nutUSpaldingWallFunction
  - nutURoughWallFunction
  - nutUWallFunction
- The source code of the nut wall functions is located in the following directories:
  - `$WM_PROJECT_DIR/src/MomentumTransportModels/momentumTransportModels/derivedFvPatchFields/wallFunctions/nutWallFunctions`
- You will find the following turbulent thermal diffusivity boundary conditions (alphatWallFunctions):
  - alphasJayatillekeWallFunction
  - alphasWallFunction
- The source code of the thermal diffusivity boundary conditions is located in the following directories:
  - `$WM_PROJECT_DIR/src/ThermophysicalTransportModels/derivedFvPatchFields/alphatWallFunctions`

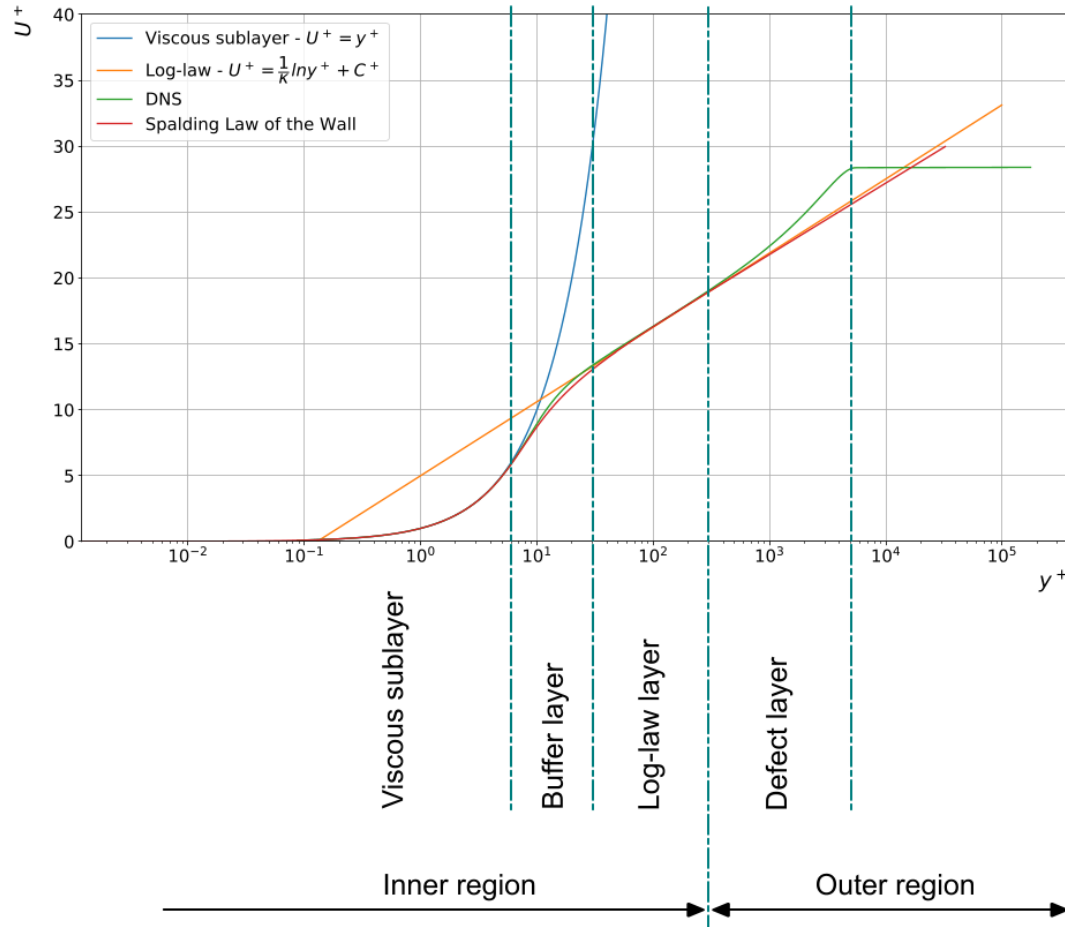
# Turbulence modeling in OpenFOAM

## Near-wall treatment and wall functions

- You will find the following turbulent viscosity boundary conditions (atmBoundaryLayer):
  - atmBoundaryLayerInletEpsilon
  - atmBoundaryLayerInletK
  - atmBoundaryLayerInletVelocity
  - nutkAtmRoughWallFunction
- The source code of the atmospheric boundary layer wall functions is located in the following directories:
  - `$WM_PROJECT_DIR/src/atmosphericModels/derivedFvPatchFields/`
- You will find the following surface film models boundary conditions (wallFunctions):
  - alphasatFilmWallFunction
  - nutkFilmWallFunction
- The source code of the wall functions is located in the following directories:
  - `$WM_PROJECT_DIR/src/regionModels/surfaceFilmModels/derivedFvPatchFields/wallFunctions`

# Turbulence modeling in OpenFOAM

## Near-wall treatment and wall functions



- If the first node normal to the wall is in the viscous sub-layer region, you do not need to use wall functions.
- Have in mind that this approach is computationally expensive (in particular for unsteady simulations).
- Instead, if the first node normal to the wall is in the log-law layer, you need to use wall functions.
- When positioning the first node normal to the wall, try to avoid as much as possible the buffer layer, this is the transition region and things are not very clear there.

# Turbulence modeling in OpenFOAM

## Near-wall treatment and wall functions

- Let us be really loud on this. To enable wall functions, you need to define the **base type** boundary condition of the patch as a **wall**.
- The **base type** boundary conditions are defined in the dictionary file *constant/polyMesh/boundary*.

Name of the patch

```
bottomWall
{
    type
    nFaces
    startFace
}
```

The **base type** of the patch. If the physical patch is a wall it must be defined as a **wall** here.

- The specific wall function or **primitive type** boundary condition to be used, is defined in the field dictionaries located in the directory 0, or the directory where you set the initial and boundary conditions.

```
bottomWall
{
    type nutkWallFunction;
    value uniform 1e-12;
}
```

```
bottomWall
{
    type kqRWallFunction;
    value uniform 0.0001;
}
```

```
bottomWall
{
    type omegaWallFunction;
    value uniform 100000;
}
```

# Turbulence modeling in OpenFOAM

## Initial estimates – Boundary conditions and initial conditions

- OpenFOAM does not come with any utility to compute the initial estimates for the boundary conditions and initial conditions (at least that we are aware of).
- Therefore, the user needs to compute these values manually.
- You can follow the guidelines given hereafter or you can refer to the original reference of the turbulence model you are planning to use.
- The following link is a very good source for turbulence models information:
  - <https://turbmodels.larc.nasa.gov/>
- At this link, you can find a calculator to compute the free-stream estimates:
  - <http://www.wolfdynamics.com/tools.html?id=110>
- If you use commercial software, you do not compute these initial estimates because the solver comes with built-in functions to compute these estimates.
  - But you always have the option to input these values manually.

# Turbulence modeling in OpenFOAM

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

- Let us use the  $k - \omega$  turbulence model to illustrate how to choose the free-stream initial conditions.
- The initial value for the turbulent kinetic energy  $k$  can be found as follows

$$k = \frac{3}{2}(UI)^2$$

- The initial value for the specific rate of dissipation  $\omega$  can be found as follows

$$\omega = \frac{\rho k}{\mu} \left( \frac{\mu_t}{\mu} \right)^{-1}$$

- Where  $\mu_t/\mu$  is the viscosity ratio and  $I = u'/\bar{u}$  is the turbulence intensity.
- The specific kinetic energy can be related to the turbulent dissipation and the turbulent kinetic energy as follows,

$$\omega = \frac{\epsilon}{C_\mu k}$$

# Turbulence modeling in OpenFOAM

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

- 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 %
$\mu_t / \mu$	1	10	100

- By the way, use these guidelines for external aerodynamics only.
- Follow these guidelines if you are struggling to find the boundary conditions for the near-wall treatment.
- We highly recommend you read the source code and find the references used to implement the model.

# Turbulence modeling in OpenFOAM

## $k - \omega$ 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 (depending on the Reynolds number, values up to 1000 are acceptable if they do not cover more than 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):

$$y^+ \leq 6$$



# Turbulence modeling in OpenFOAM

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

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

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

- For  $\omega$  you can use the wall function **omegaWallFunction** with the following value:

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

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.

# Turbulence modeling in OpenFOAM

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

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

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

- For  $\omega$  you should use the wall function **omegaWallFunction** with the following value:

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

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.

# Turbulence modeling in OpenFOAM

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

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

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

- For  $\omega$  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.

# Turbulence modeling in OpenFOAM

## Setting nut boundary conditions for the $k - \omega$ Turbulence model

- As we are solving a boundary condition – initial condition problem (**BCIC**), we need to define boundary and initial conditions for **nut** (turbulent viscosity) as well.
- Remember, all this fancy treatment for the closure equations is done only to compute **nut**, or the turbulent viscosity (which is not a physical property).
- We only define **nut** boundary conditions at the walls.
- The free stream values are computed from  $k$  and  $\omega$  (**calculated** boundary condition using OpenFOAM jargon).
- The source code of the **nut** wall functions is located in the directory:
  - `$WM_PROJECT_DIR/src/MomentumTransportModels/  
momentumTransportModels/derivedFvPatchFields/  
wallFunctions/nutWallFunctions`

# Turbulence modeling in OpenFOAM

## Setting nut boundary conditions for the $k - \omega$ Turbulence model

- If you are interested in using  $y^+$  insensitive wall functions with the  $k - \omega$  family models, you can use any of the following wall functions for the turbulent viscosity:
  - **nutUSpaldingWallFunction** (continuous Spalding function).
  - **nutkWallFunction** ( $y^*$  based – It uses the turbulent kinetic energy).
  - **nutUWallFunction** ( $y^+$  based).
- In our experience, all these wall functions can be used with low-Re and high-Re turbulence models.
- However, the **nutkWallFunction** wall function is the most stable one.
- But attention, **nutkWallFunction** can only be used if the turbulent kinetic energy is computed by the turbulence model.
- Other **nut** wall function options are:
  - **nutkRoughWallFunction** (high-Re with roughness)
  - **nutURoughWallFunction** (high-Re with roughness)
  - **nutLowReWallFunction** (low-Re)

# Turbulence modeling in OpenFOAM

## 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
<b>nut</b>	<b>nut(-)WallFunction*</b> or <b>nutUSpaldingWallFunction**</b> (with 0 or a small number)	<b>nutUSpaldingWallFunction**</b> , <b>nutkWallFunction</b> , <b>nutUWallFunction</b> , <b>nutLowReWallFunction</b> or <b>fixedValue***</b> (with 0 or a small number)
<b>k, q, R</b>	<b>kqRWallFunction**</b> (with inlet value or a small number) $k_{wall} = k$	<b>kqRWallFunction**</b> or <b>kLowReWallFunction</b> (with inlet value, 0, or a small number) or <b>fixedValue***</b> (with 0 or a small number)
<b>epsilon</b>	<b>epsilonWallFunction</b> (with inlet value) $\epsilon_{wall} = \epsilon$	<b>epsilonWallFunction</b> (with inlet value) or <b>zeroGradient***</b> or <b>fixedValue***</b> (with 0 or a small number)
<b>omega</b>	<b>omegaWallFunction**</b> (with a large number) $\omega_{wall} = 10 \frac{6\nu}{\beta y^2} \quad \beta = 0.075$	<b>omegaWallFunction**</b> or <b>fixedValue***</b> (both with a large number)
<b>nuTilda</b>	–	<b>fixedValue</b> (one to ten times the molecular viscosity, a small number, or 0)

\* nutUWallFunction or nutkWallfunction

\*\* Recommended options for  $y^+$  insensitive treatment (continuous wall functions)

\*\*\* Will disable wall functions. The equations will be integrated down to the viscous sublayer with no damping or corrections.

# Turbulence modeling in OpenFOAM

## Setting the right numerics – The $k - \omega$ turbulence model

- When we use a turbulence model, we are solving for additional transport equations for the turbulent variables, therefore we need to choose a discretization method for the new variables.
- If you are working with the  $k - \omega$  turbulence model, you can use the following discretization method for the convective terms (*fvSchemes*),

```
divSchemes
{
    div(phi,U)          Gauss linearUpwind default;
    div(phi,k)          Gauss linearUpwind grad(k);
    div(phi,omega)      Gauss linearUpwind grad(omega);
    div((nuEff*dev2(T(grad(U)))) Gauss linear;
}
```

Second order accurate discretization (for turbulence variables)

```
divSchemes
{
    div(phi,U)          Gauss linearUpwind default;
    div(phi,k)          Gauss upwind;
    div(phi,omega)      Gauss upwind;
    div((nuEff*dev2(T(grad(U)))) Gauss linear;
}
```

First order accurate discretization (for turbulence variables)

- Remember, turbulence is a diffusive process so using a first order accurate discretization method for the turbulence variables is acceptable (if you are doing RANS/URANS).
- If you are working with SRS models, it is highly recommended to use second order accurate methods for the turbulent variables.
- Depending on the physics involved and method used, there will be different entries.

# Turbulence modeling in OpenFOAM

## Setting the right numerics – The $k - \omega$ turbulence model

- When we use a turbulence model, we are solving for additional transport equations for the turbulent variables, therefore we need to choose a discretization method for the new variables.
- If you are working with the  $k - \omega$  turbulence model, you can use the following discretization method for the gradient terms (*fvSchemes*),

Blending coefficient (between 0 and 1). If it is 1, limiter enabled

```
gradSchemes
{
    default          cellLimited Gauss linear 1;
    grad(k)          cellLimited Gauss linear 1;
    grad(omega)      cellLimited Gauss linear 1;
}
```

Bounded gradient discretization (including turbulence variables)

Gradient limiter disabled

```
gradSchemes
{
    default          cellLimited Gauss linear 0;
    grad(k)          cellLimited Gauss linear 0;
    grad(omega)      cellLimited Gauss linear 0;
}
```

Unbounded gradient discretization (including turbulence variables)

- When it comes to gradient discretization, it is a good idea to use a bounded method for the turbulent variables (if you are doing RANS/URANS).
- You can start with a very aggressive method (blending coefficient equal to 1), and then change to a more accurate and less dissipative method (blending coefficient equal to 0.5). However, always monitor the solution, if the turbulent variables start to oscillate too much, go back to the aggressive formulation.
- If you are working with SRS models, it is highly recommended to use an unbounded method as it is less dissipative (blending coefficient between 0 and 0.5).
- Depending on the physics involved and method used, there will be different entries.



# Turbulence modeling in OpenFOAM

## Setting the right numerics – The $k - \omega$ turbulence model

- Finally, if you are using the  $k - \omega$  family models (or any other  $y^+$  insensitive method), you will need to define a method for the calculation of the approximate distance to the nearest patch for all cells and boundaries (*fvSchemes*),

This entry is needed for any model that relies on the calculation of the distance to the nearest patch for all cells

```
wallDist
{
    method meshWave;
}
```

- The following options are available: advectionDiffusion, meshWave, Poisson.
- For good quality meshes (low skewness and low non-orthogonality), the meshWave method is a good choice. Otherwise, use the Poisson method.
- The advectionDiffusion method is the most accurate, but it is computationally expensive.
- You will find the source code in the following directory:
  - `$WM_PROJECT_DIR/src/finiteVolume/fvMesh/wallDist/patchDistMethods`

# Turbulence modeling in OpenFOAM

## Searching for information in the source code

- If you want to find where the directories containing some specific information are located (e.g., the wall functions), type in the terminal:
  - `$> find $FOAM_SRC -iname wallFunctions`
- To find a string of text in any files, go to the desired directory and type in the terminal:
  - `$> grep -r nutUWallFunction`
- Remember, you can use wild cards, e.g.,
  - `$> grep -r .*WallFunct.*`
- You can use these commands to search for general information in the source code.

## **Guided tutorial 6 RANS multi-element airfoil**

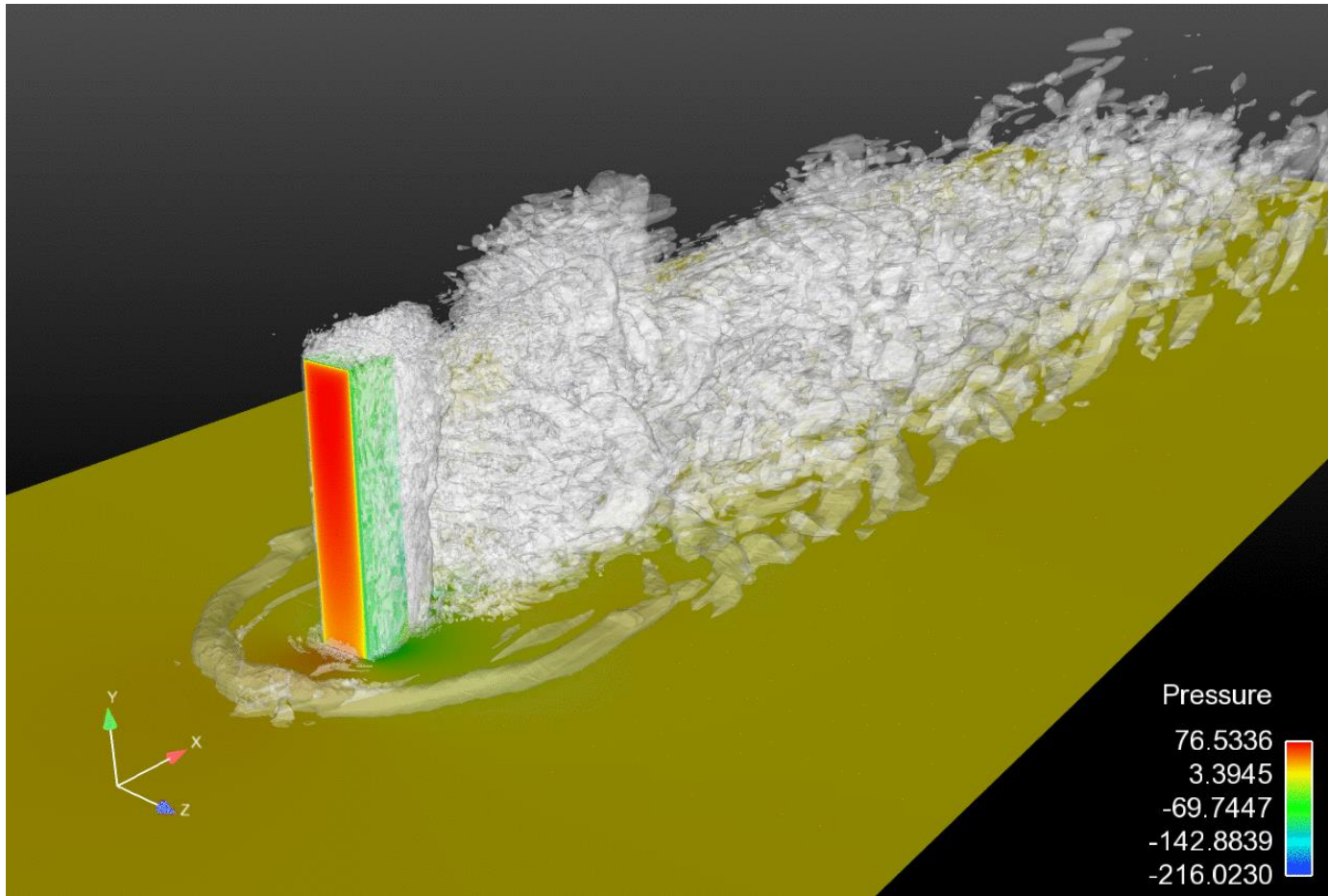
## **Guided tutorial 7 Static mixer – RSM**

# Roadmap

- ~~1. The turbulent world around us~~
- ~~2. Turbulence, does it matter?~~
- ~~3. Introduction to turbulence modeling~~
- ~~4. Post-processing turbulence simulations~~
- ~~5.  $y^+$  and wall distance units~~
- ~~6. Turbulence modeling in OpenFOAM~~
- 7. Some applications using OpenFOAM**
- ~~8. Final remarks – General guidelines~~

# Some applications using OpenFOAM

## Some applications – LES simulation



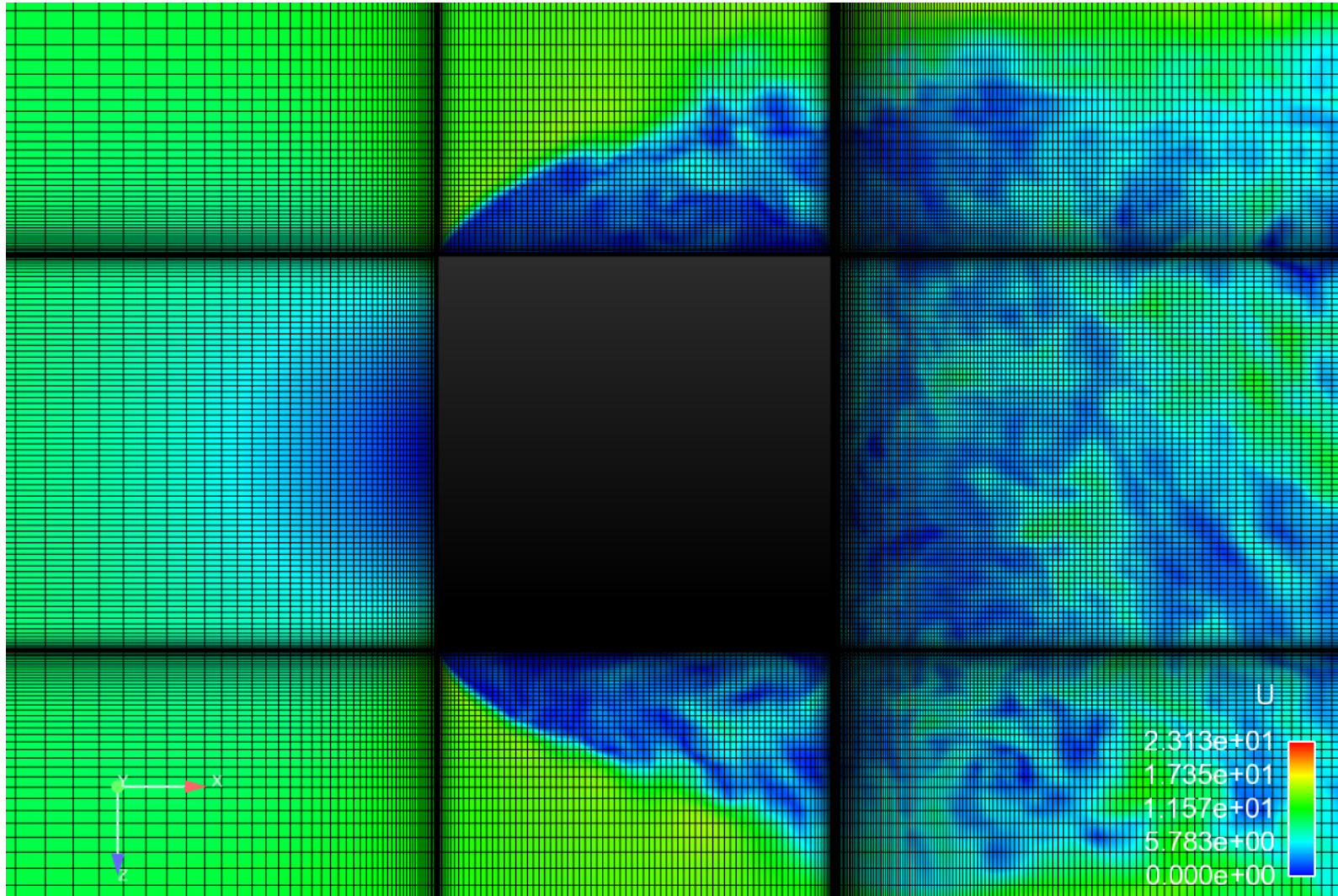
Iso-surfaces of Q criterion. Walls colored by instantaneous pressure.

The mesh size is about

<http://www.wolfdynamics.com/training/turbulence/image12.gif>

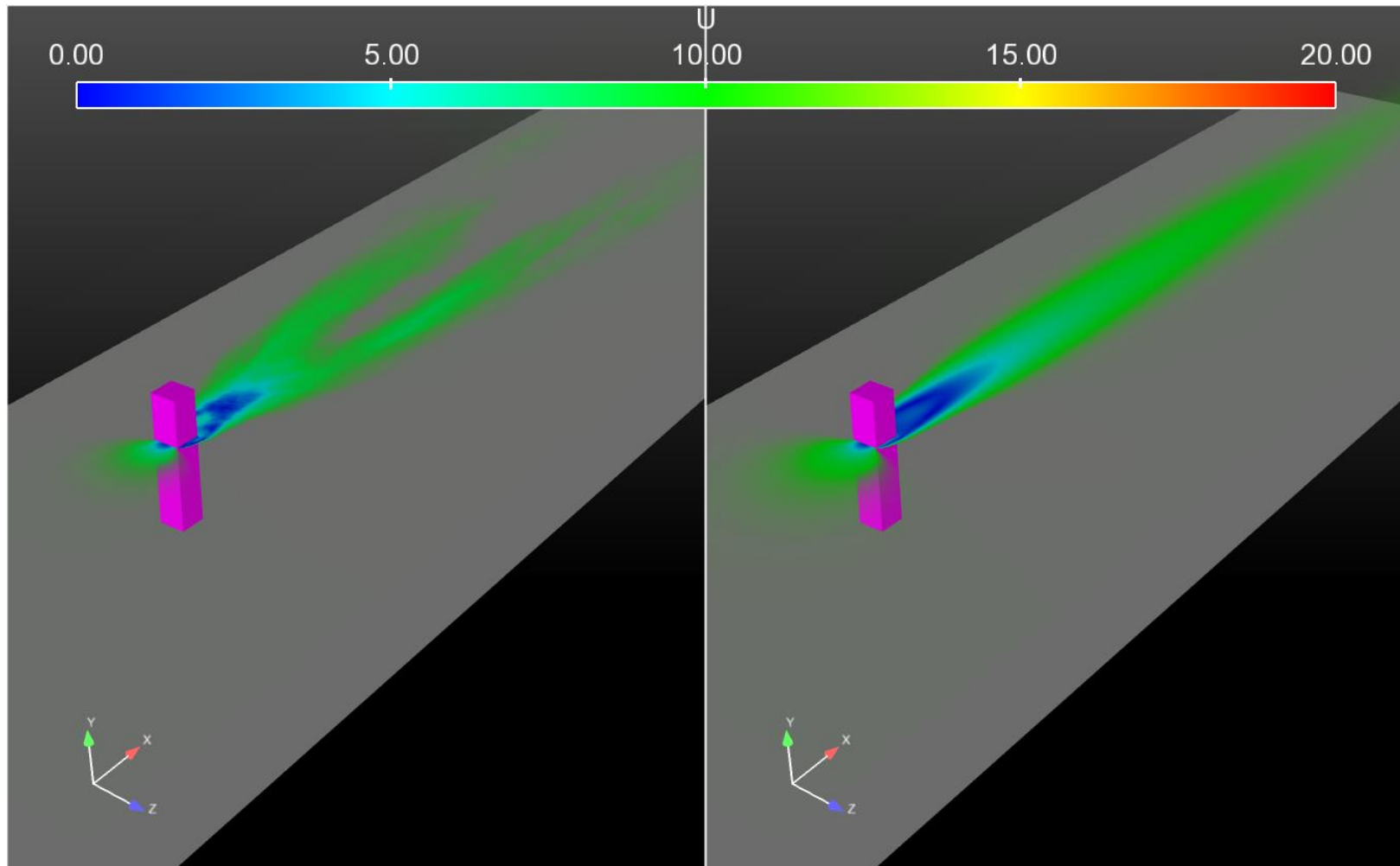
# Some applications using OpenFOAM

## Some applications – LES simulation



# Some applications using OpenFOAM

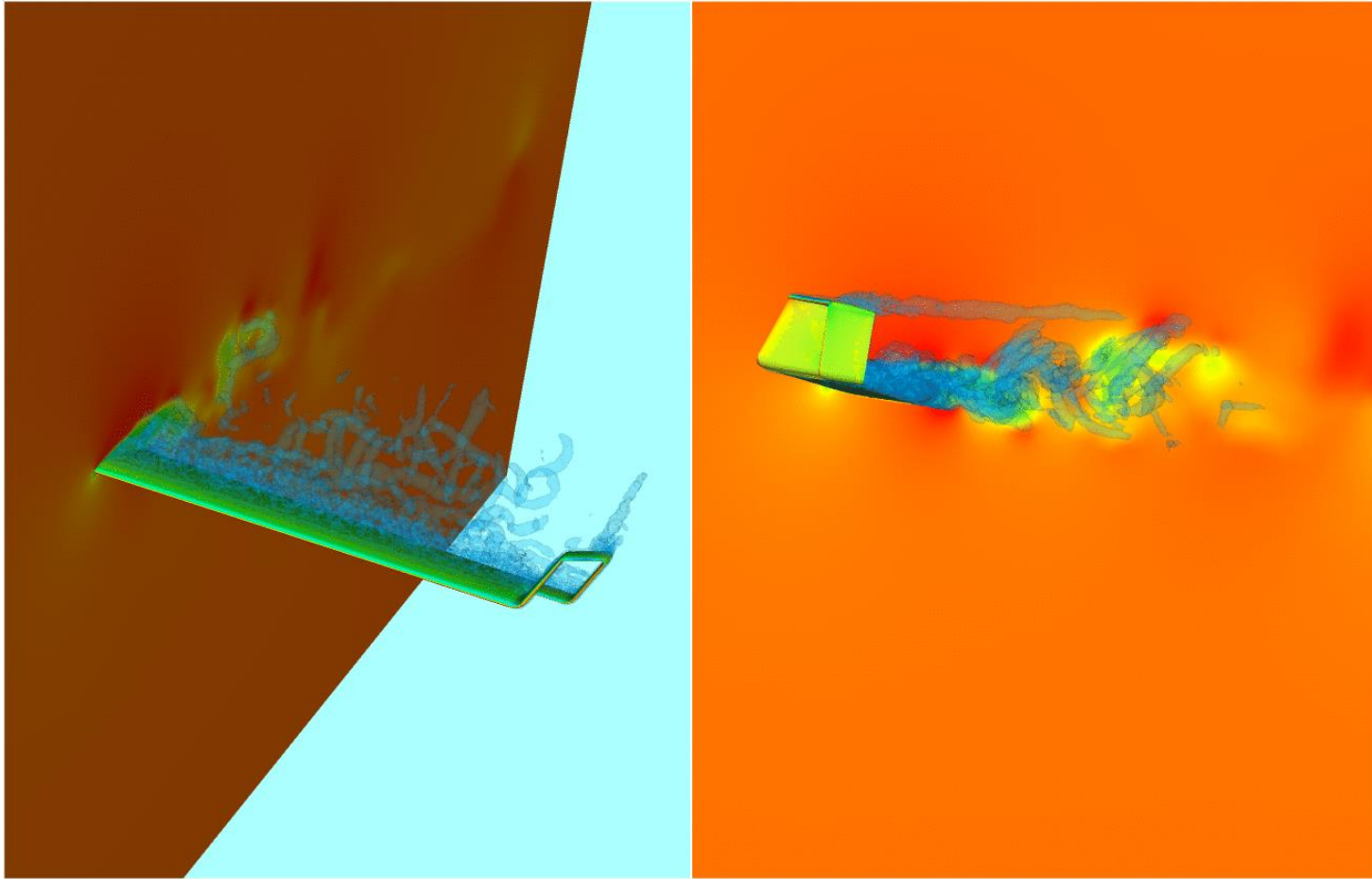
## Some applications – LES simulation



LES simulation Vs. RANS simulation

# Some applications using OpenFOAM

## Some applications – DES simulation



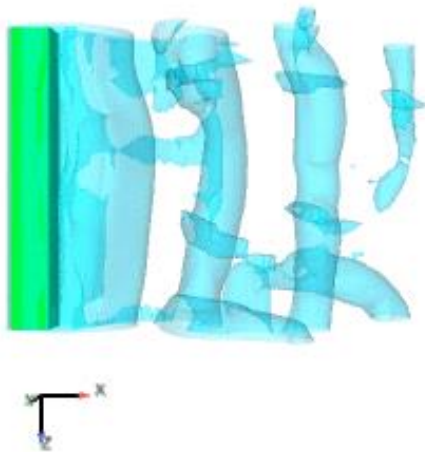
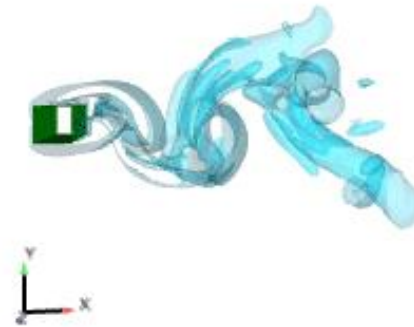
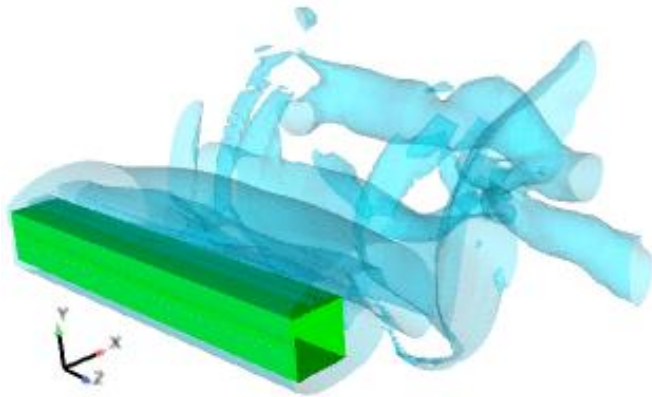
Iso-surfaces of Q criterion.

<http://www.wolfdynamics.com/training/turbulence/image13.gif>



# Some applications using OpenFOAM

## Some applications – URANS simulation

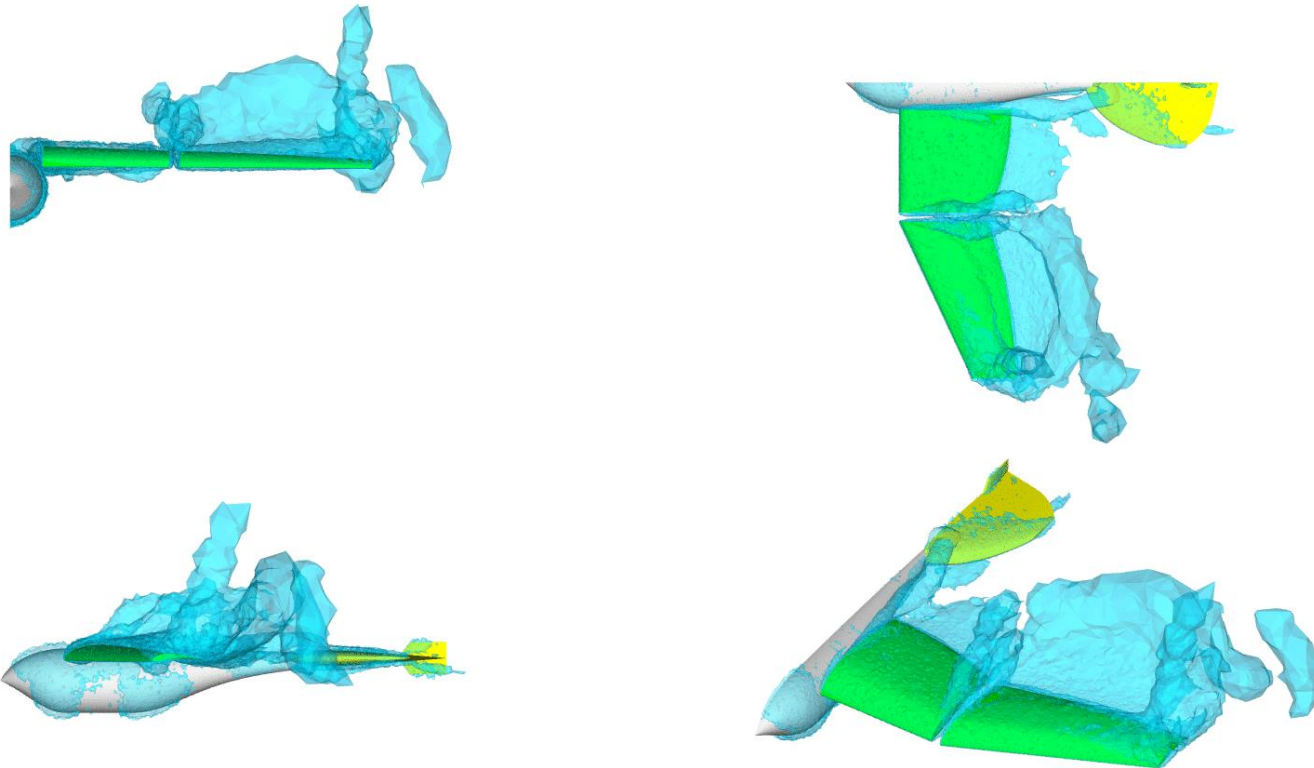


Iso-surfaces of vorticity magnitude.

<http://www.wolfdynamics.com/training/turbulence/image14.gif>

# Some applications using OpenFOAM

Some applications – URANS simulation with dynamic meshes

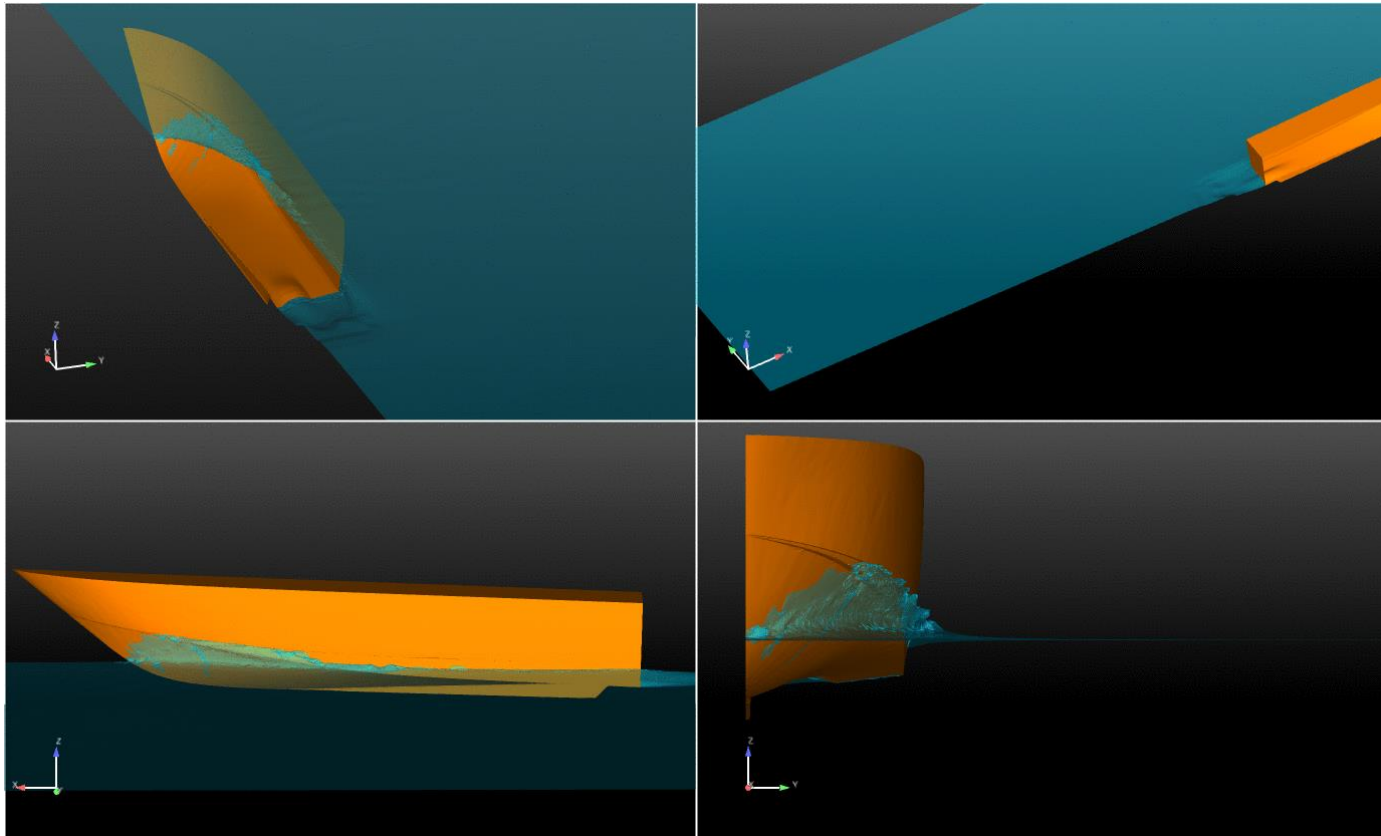


URANS simulation with moving walls. Vorticity structures visualized using iso-surfaces of Q criterion.

<http://www.wolfdynamics.com/training/turbulence/image15.gif>

# Some applications using OpenFOAM

Some applications – Multiphase flow simulation (VOF)

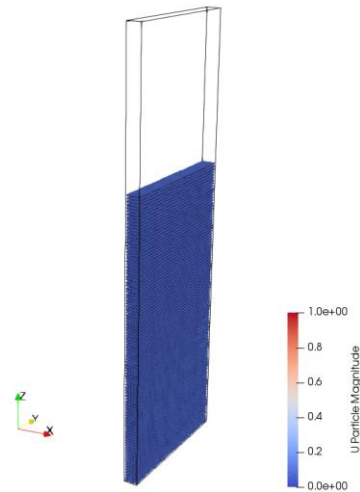
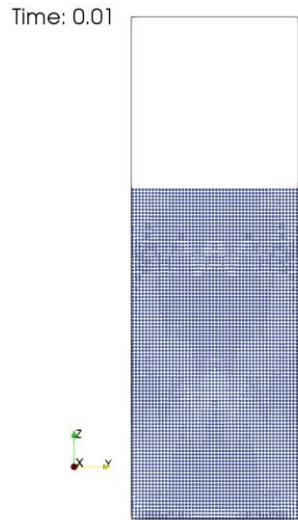


Free surface – Sea keeping

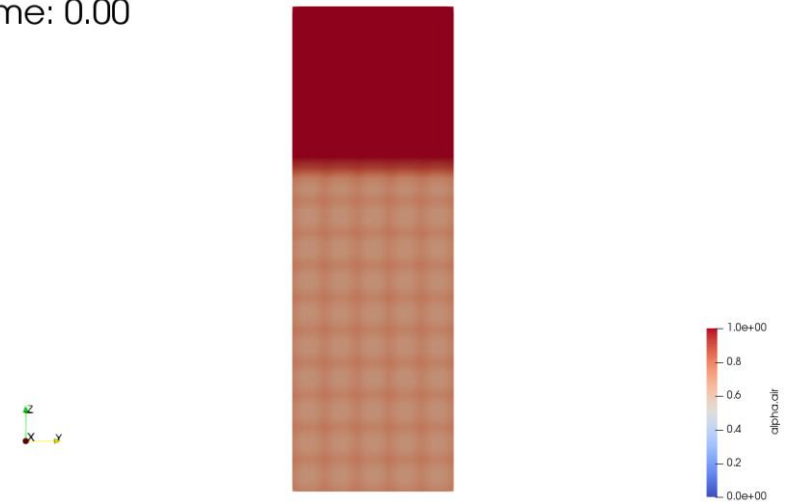
<http://www.wolfdynamics.com/training/turbulence/image18.gif>

# Some applications using OpenFOAM

## Some applications – Multiphase flow simulation (Dispersed flow)



Time: 0.00



<http://www.wolfdynamics.com/training/turbulence/image16.gif>

<http://www.wolfdynamics.com/training/turbulence/image17.gif>

Fluidized bed hydrodynamics – Gas-particles interaction with hydrodynamic coupling

# Roadmap

- ~~1. The turbulent world around us~~
- ~~2. Turbulence, does it matter?~~
- ~~3. Introduction to turbulence modeling~~
- ~~4. Post-processing turbulence simulations~~
- ~~5.  $y^+$  and wall distance units~~
- ~~6. Turbulence modeling in OpenFOAM~~
- ~~7. Some applications using OpenFOAM~~
- 8. Final remarks – General guidelines**

# Final remarks – General guidelines

## Turbulence modeling guidelines and tips

- Compute Reynolds number and determine whether the flow is turbulent.
- Try to avoid using turbulent models with laminar flows.
- Choose the near-wall treatment and estimate  $y$  normal distance before generating the mesh.
- Run the simulation for a few time steps and get a better prediction of  $y^+$  and correct your initial prediction of  $y$ .
- The realizable  $k - \epsilon$  or  $k - \omega$  SST models are good choices for general applications.
- The standard  $k - \epsilon$  model is very reliable, you can use it to get initial values for more sophisticated models. Remember, OpenFOAM implementation force you the use of wall functions.
- If you are interested in resolving the large eddies and the inertial range, and modeling the smallest eddies, DES or LES are the right choice.
- If you do not have any restriction in the near wall treatment method, use wall functions.

# Final remarks – General guidelines

## Turbulence modeling guidelines and tips

- Choose your near-wall modeling strategy ahead of time and check  $y^+$  and  $y$  values to make sure the near-wall mesh is suitable.
- The utility `refineWallLayer`, can be used to refine the cells next to patches. This can save you a lot time, as you do not have to go back to the meshing stage. But use it with care, because it can increase a lot the skewness of the mesh.
- Be aware of the limitations of the turbulence model chosen, find and read the original references used to implement the model in OpenFOAM.
- Use the functionObject `yPlus` to check/plot the  $y^+$  value.
- Use the default model constants unless you know what are you doing or you are confident that you have better values.
- Set reasonable boundary and initial conditions for the turbulence model variables.
- Always monitor the turbulent variables, some of them are positive bounded.

# Final remarks – General guidelines

## Turbulence modeling guidelines and tips

- Avoid strong oscillations of the turbulent variables.
- If you are doing LES or DES, remember that these models are intrinsically 3D and unsteady.
- In LES you should choose your time-step in such a way to get a CFL of less than 1 and preferably of about 0.5 for LES. DES simulations can use larger CFL values (up to 5 for reasonable accuracy).
- If you are doing RANS, it is perfectly fine to use upwind to discretize the turbulence closure equations. After all, turbulence is a dissipative process. However, some authors may disagree with this, make your own conclusions.
- On the other hand, if you are doing LES you should keep numerical diffusion to the minimum, so you should use second order methods.
- LES methods can be sensitive to mesh element type; it is highly recommended to use hexahedral meshes.
- Mesh quality is of paramount importance, try to avoid bad elements near the inlets (as they can introduce numerical diffusion) or at the walls (as they can affect the boundary layer or wall functions).



# Final remarks – General guidelines

## Turbulence modeling guidelines and tips

- Reynolds stress models are expensive and require a good initial guess. However, they perform well with swirling flows.
- If you are doing unsteady simulations, always remember to compute the average values (ensemble average).
- Avoid the use of adaptive time-stepping and adaptive save intervals, as they may introduce oscillations in your solution.
- If you are dealing with multiphase flows, remember that some turbulence models are hardwired to the multiphase models.
- If you are working with combustions and aero-acoustics, you will get best results using LES models but at the cost of higher computational requirements.
- If you are dealing with external aerodynamics and detached flows, DES simulations are very affordable, and surprisingly, they give good results most of the times.
- If you are doing LES and using the Smagorinsky model, remember that it does not perform very well with wall bounded flows, so you might need to change the value of the Smagorinsky constant.

# Final remarks – General guidelines

## Turbulence modeling guidelines and tips

- DNS requires no modeling, but it demands high mesh resolution for the large scales all the way through at least the beginning of the dissipation scales. This requires an incredible amount of mesh cells (in the order of  $Re^3$  or worse).
- LES requires modeling a part of the inertial subrange and into the beginning of the dissipation scales. The amount of required modeling is set by the mesh resolution that can be afforded (at worst in the order of  $Re^2$  which is much less than the mesh resolution for DNS but still is a high requirement).
- In general, LES models are less expensive than DNS, but much more expensive than RANS/URANS.
- RANS/URANS requires modeling of everything from the integral scales into the dissipation range and only mean quantities are computed. Despite this, they perform very well.
- The hybrid method DES, model everything close to the walls and resolves all the scales in the far field (as in LES). DES methods have mesh resolution requirements between RANS and LES.
- The work-horse of turbulence modeling in CFD: RANS

# Final remarks – General guidelines

## Future of Turbulence Modelling in Industrial Applications

- Many authors state that the future trends are quite clear: moving from RANS models to LES models.
- However, I politely disagree with this as many industrial applications are quite complex in order to simulate them using LES models.
- LES simulations are unattractive in industry due to the excessive amount of computational resources needed (which cost money), and the amount of time needed to get the outcomes (and time is money).
- RANS models are the work horse of industrial applications and will continue to be until a big leap in computing hardware or solution strategy happens.
- However, academia is moving slowly towards LES and new computing platforms (which hopefully will consume less energy than wind tunnels).
- DES simulations are starting to become more affordable and are slowly replacing URANS.
- DNS remains out of reach for all engineering use. However, it provides a very good base for model development and testing.

# Final remarks – General guidelines

## References

- S. Pope. Turbulent Flows. Cambridge University Press, 2000.
- D. Wilcox. Turbulence Modeling for CFD. DCW Industries ,Inc., 2010.
- P. A. Davidson. Turbulence: An Introduction for Scientists and Engineers. Oxford University Press, 2015.
- M. Leschziner. Statistical Turbulence modelling for Fluid Dynamics. Imperial College Press, 2016.
- P. Bernard. Turbulent Fluid Flow. Wiley, 2019.
- S. Rodriguez. Applied Computational Fluid Dynamics and Turbulence Modeling. Springer, 2020.
- H. Tennekes and J. L. Lumley. A First Course in Turbulence. MIT Press, 1972.
- P. Sagaut. Large Eddy Simulation for Incompressible Flows. Springer, 2006.
- H. Schlichting, K. Gersten. Boundary-Layer Theory. 9<sup>th</sup> Edition, Springer, 2017.
- J. Mathieu, J. Scott. An Introduction to Turbulent Flow. Cambridge University Press, 2000.
- P. Libby. Introduction to Turbulence. Taylor Francis, 1996.
- U. Frisch. Turbulence. The Legacy of A. N. Kolmogorov. Cambridge University Press, 2004.
- J. Hinze. Turbulence. McGraw-Hill, 1975.
- K. Hoffmann, S. Chiang. Computational Fluid Dynamics. Volume III. ESSBooks, 2000.
- Turbulence Modelling by Lars Davidson [http://www.tfd.chalmers.se/~lada/comp\\_turb\\_model/](http://www.tfd.chalmers.se/~lada/comp_turb_model/)
- NASA turbulence modeling resources - <https://turbmodels.larc.nasa.gov/>
- Turbulence and CFD models by J. Guerrero [http://www3.dicca.unige.it/guerrero/teaching\\_turbulence.html](http://www3.dicca.unige.it/guerrero/teaching_turbulence.html)

## **Guided tutorial 8**

**Vortex shedding past square cylinder  
RANS – DES – LES – DNS**

## **Guided tutorial 9**

**Bonus case – Ahmed body**

# Thank you for your attention

- We hope you have found this training useful, and we hope to see you in one of our advanced training sessions:
  - OpenFOAM® – Multiphase flows
  - OpenFOAM® – Naval applications
  - OpenFOAM® – Turbulence Modeling
  - OpenFOAM® – Compressible flows, heat transfer, and conjugate heat transfer
  - OpenFOAM® – Advanced meshing
  - DAKOTA – Optimization methods and code coupling
  - Python – Programming, data visualization, and exploratory data analysis
  - Python and R – Data science and big data
  - ParaView – Advanced scientific visualization and python scripting
  - And many more available on request
- Besides consulting services, we also offer '**Mentoring Days**' which are days of one-on-one coaching and mentoring on your specific problem.
- For more information, ask your trainer, or visit our website  
<http://www.wolfdynamics.com/>

**Be collaborative, be innovative, be cloud**



**wolf**dynamics

multiphysics simulations,  
optimization & data analytics

**Let's connect**



[guerrero@wolfdynamics.com](mailto:guerrero@wolfdynamics.com)



[www.wolfdynamics.com](http://www.wolfdynamics.com)