# PROJECTE O TESISNA D’ESPECIALITAT

## Títol
Numerical investigation of the flow around lightweight solar modules. Determination of the uplift forces by means of CFD modeling and wind tunnel validation.

## Autor/a
Sandra Mollet Torrella

## Tutor/a
Allen Bateman Pinzón
Carles Colomer Segura

## Departament
Departament d’Enginyeria Hidràulica, Marítima i Ambiental

## Intensificació
Mecànica de fluids

## Data
Octubre 2011
Abstract

Numerical investigation of the flow around lightweight solar modules. Determination of the uplift forces by means of CFD modeling and wind tunnel validation.

Author: Sandra Mollet Torrella
Tutors: Carles Colomer Segura and Allen Bateman Pinzón

Abstract

Present developments in construction engineering are aiming towards bigger and more slender structures, where wind loading becomes an even more decisive factor in its design. These structures are very often conceived with special geometrical shape that leads to highly difficult (sometimes impossible) approaches to determine the wind load effects. Therefore wind tunnel investigations have been regularly used to get a reliable approach to the real wind conditions on and around the structure.

The appearance in early 1960’s of Computational fluid dynamics (CFD) let us predict the fluid fields and other physics in detail for an application of interest by using numerical methods and algorithms. Before the appearance of CFD the fluid mechanics advancements where done with the combination of experiments and basic theoretical analyses, which have the disadvantage of not including all the required physics of the flow. But now, the role of CFD in engineering predictions has become so strong, that today it can be seen as a new “third dimension” in fluid dynamics. CFD had rapidly become a popular tool in engineering analyses.

CFD calculations in civil engineering are still an immature alternative to well-established methods like wind tunnel. The large computational effort and time-consuming calculations together with the difficulties of modeling the atmospheric boundary layer and its turbulent structures had been the main drawbacks for dealing with wind effects on structures. Nevertheless in the latest years, an always increasing computer power has opened the possibilities of simulation in the field structural aerodynamics, as they start giving accurate results with affordable time expense.

One of the main deficiencies of CFD is the CFD solutions of turbulent flows. These solutions contain turbulence models which are just approximations of the real physics. Therefore, all CFD solutions of turbulent flows are subject to inaccuracy. CFD community is directly attacking this problem in the most basic sense. There is work today on the direct computation of turbulence. This is currently a wide open area of CFD research.

It is the purpose of this study to compare the CFD solutions of a turbulent flow with the solutions obtained in a wind tunnel experiment, in order to validate its use. This study focuses on generating a suitable model for a lightweight solar module and carry out a numerical investigation of the flow around it and determining the uplift forces using computational fluid dynamics (CFD) calculations.

Finally the results will be validated with the results obtained in the experiments done in a wind tunnel by the Institute of Steel Construction of the RWTH University of Aachen.
Numerical investigation of the flow around lightweight solar modules. Determination of the uplift forces by means of CFD modeling and wind tunnel validation.

**Autora:** Sandra Mollet Torrella  
**Tutors:** Carles Colomer Segura i Allen Bateman Pinzón

**Resum**

Els actuals desenvolupaments en enginyeria de la construcció apunten cap a estructures cada vegada més grans i esveltes, on les càrregues del vent es converteixen en un factor encara més determinant en el moment del disseny. Aquestes estructures són sovint concebudes amb geometries especials que dificulten, o inclús fan impossible, el poder determinar els efectes de les càrregues del vent. És per aquest motiu que les investigacions amb túneles de vent han estat utilitzades regularment per a aconseguir una aproximació fiable de les condicions de vent reals, sobre i al voltant de l’estructura.

L’aparició de *Computational fluid dynamics (CFD)* a principis dels anys 60 ens ha permès predir els flux d’un fluid i d’altres propietats en detall usant mètodes numèrics i algoritmes. Abans de l’aparició de CFD els avanços en la mecànica de fluids eren duts a terme mitjançant la combinació d’experiments i teoria bàsica, teoria que sovint no pot incloure tots els requisits físics del fluid. Avui en dia la utilització dels CFD en l’enginyeria ha adoptat un paper important, tant que es pot veure com una tercera dimensió en la dinàmica de fluids. CFD s’ha convertit ràpidament en una eina molt popular en anàlisi d’enginyeria.

Els càlculs amb CFD en l’enginyeria civil són encara una alternativa poc desenvolupada comparada amb mètodes com els túneles de vent. El gran cost computacional i temps requerit juntament amb la complexitat de modelar la capa límit atmosfèrica han estat un dels principals inconvenients a l’hora d’afrontar els efectes del vent sobre les estructures. Tot i així, en els últims anys, sempre tenint en compte la creixent evolució de la potència dels ordinadors, s’han augmentat les possibilitats de la simulació en camps d’aerodinàmica estructural, ja que comencen a donar resultats força precisos amb una despesa de temps raonable.

Una de les principals deficiències dels CFD, és la seva utilització en fluxos turbulents. Aquestes solucions contenen models de turbulència que aproximen les condicions físiques reals. És per això, que totes les solucions de CFD amb fluxos turbulents estan subjectes a imprecisions. La comunitat de CFD està actualment intentant resoldre aquest problema, és un camp en el que encara s’està investigant.

L’objectiu d’aquesta tesina és resoldre un problema amb flux turbulent mitjançant les eines que ens proporcionen el CFD i després comprovar la seva validesa comparant aquests resultats amb els obtinguts amb assajos en un túnel de vent. L’estudi es centra en la creació d’un model per a un panell solar, estudiar el flux al voltant d’aquest i determinar les forces que actuen sobre ell utilitzant CFD.

Finalment els resultats seran validats amb els obtinguts en els experiments duts a terme per el Institut of Steel Construction de la universitat RWTH d’Aachen.
Acknowledgments

I would like to thank all people who have helped and inspired me during this thesis.

I especially want to thank, my supervisor in Germany, the engineer Carles Colomer, who helped me with his knowledge, guidance and support. Without him this thesis would not have been possible.

I am grateful to the Institute of Steel Structures of the RWTH University of Aachen, for letting me use their installations and helping me whenever I needed.

I am very grateful to all the professors that during these 5 years of study, have thought me, and brought me all the knowledge that helped me write this thesis, especially to the professor Allen Bateman, tutor of the present thesis.

I warmly thank my family for all their support during all these years.

Finally I owe special gratitude to all my friends, for their continuous and unconditional support, for all those long days studying together and the happy times we spent. I would like to especially thank Robert Fontecha and Alejandro Tornay, who have become a family for me during this year in Aachen.
Table of contents

Abstract ........................................................................................................................................... 2
Acknowledgments .......................................................................................................................... 4
List of figures .................................................................................................................................. 9
1. Introduction and objectives ........................................................................................................ 11
2. Previous concepts ..................................................................................................................... 13
3. Turbulence ................................................................................................................................ 17
4. Turbulence models ...................................................................................................................... 22
   4.1. Direct numerical simulation (DNS) ...................................................................................... 22
   4.2. Reynolds-averaged Navier-Stokes (RANS) ......................................................................... 23
       4.2.1. Linear eddy viscosity models ...................................................................................... 24
           4.2.1.1. Zero equation models .......................................................................................... 25
           4.2.1.2. One equation models .......................................................................................... 25
           4.2.1.3. Two equation models .......................................................................................... 25
           4.2.1.3.1.  $k - \varepsilon$ models ......................................................................................... 26
           4.2.1.3.2.  $k - \omega$ models ............................................................................................. 27
       4.2.2. Nonlinear eddy viscosity models or Reynolds Stress models (RSM) .......................... 30
   4.3. Large Eddy Simulation (LES) .............................................................................................. 31
       4.3.1. Smagorinsky model ..................................................................................................... 33
       4.3.2. WALE model ............................................................................................................. 33
       4.3.3. Dynamic Smagorinsky-Lilly model ............................................................................ 33
   4.4. Detached eddy simulation model (DES) .............................................................................. 34
   4.5. SST-DES formulation ......................................................................................................... 34
   4.6. Summary ............................................................................................................................. 35
5. Boundary layer ............................................................................................................................ 36
   5.1. Boundary layer equations .................................................................................................... 36
   5.2. Boundary layer thickness ................................................................................................... 37
   5.3. Boundary sub-layers .......................................................................................................... 39
   5.4. Separation of the boundary layer ....................................................................................... 40
   5.5. Example of a fluid flow over a cylinder .............................................................................. 41
6. Meshing ....................................................................................................................................... 47
   6.1. Meshing types ..................................................................................................................... 47
Table of contents

6.2. Mesh quality ........................................................................................................... 49

7. Boundary conditions ........................................................................................................ 51
  7.1. Inlet .................................................................................................................... 52
  7.2. Outlet ................................................................................................................... 53
  7.3. Opening ............................................................................................................... 54
  7.4. Wall .................................................................................................................... 54
  7.5. Symmetry plane ................................................................................................. 55

8. Validation: experiment on solar modules ........................................................................... 56
  8.1. Motivation .......................................................................................................... 56
  8.2. Experiment .......................................................................................................... 57
  8.2.1. Experiment set-up ....................................................................................... 57
  8.2.2. Experimental results .................................................................................... 58
  8.3. CFD modeling ..................................................................................................... 60
  8.3.1. Creation of the geometry ............................................................................. 60
  8.3.2. Meshing ......................................................................................................... 60
  8.3.3. Boundary conditions .................................................................................... 63
  8.3.4. Results and comparison ............................................................................... 65

Conclusions ................................................................................................................................. 72

Bibliography ................................................................................................................................ 73

Annex A ...................................................................................................................................... 75
List of most commonly used symbols

\(e\): internal energy

\(F(x, y, z)\): body force

\(g\): acceleration due to gravity

\(k\): kinetic energy

\(L\): length-scale

\(m\): mass

\(Ma\): Mach number

\(p\): pressure

\(\Pi\): wake strength parameter

\(Re\): Reynolds number

\(S_E\): source of energy

\(S_{ij}\): mean strain rate

\(t\): time

\(T\): temperature

\(\bar{u}\): velocity

\(u, v, w\): velocity components

\(u_0\): free stream velocity

\(v_{sgr}\): subgrid-scale viscosity

\(x, y, z\): cartesian coordinates

\(y\): distance from wall

\(y^+\): dimensionless distance from wall

\(\delta\): boundary layer thickness

\(\varepsilon\): turbulent dissipation

\(\mu\): viscosity

\(\mu_t\): eddy viscosity
\( \nu \): kinematic viscosity  
\( \nu_t \): kinematic eddy viscosity  
\( \tau \): shear stress  
\( \tau_w \): wall shear stress  
\( \rho \): density  
\( \omega \): specific dissipation
List of figures

Fig. 3.1: The Reynolds experiment: (a) laminar flow, (b) transitional flow and (c) turbulent flow .................................................................................................................................................. 17
Fig. 3.2: Creation of eddies behind an object ................................................................................................................. 19
Fig. 3.3: Exemple of eddies formed by clouds behind an island ...................................................................................... 19
Fig. 4.1: Extend of modelling for some turbulent models ................................................................................................. 22
Fig. 4.2: Plots of parts of Reynolds decomposition ........................................................................................................... 23
Fig. 4.3: Representation of turbulent motion .............................................................................................................................. 31
Fig. 5.1: Comparison of laminar and turbulent boundary layers .................................................................................. 36
Fig. 5.2: Boundary layer thickness and velocity profile ........................................................................................................... 37
Fig. 5.3: Representation of the sub-layers in a turbulent boundary layer ........................................................................... 39
Fig. 5.4: Law of the wall .................................................................................................................................................. 40
Fig. 5.5: Boundary layer separation .................................................................................................................................. 41
Fig. 5.6: Mesh for fluid flow over a cylinder ........................................................................................................................... 42
Fig. 5.7: Streamlines for Re=0,112 ....................................................................................................................................... 42
Fig. 5.8: Streamlines for Re=22,4 ........................................................................................................................................ 43
Fig. 5.9: Streamlines for Re=50,41 ..................................................................................................................................... 44
Fig. 5.10: Flow past a cylinder at Re=2000 (Photograph by Werle and Gallon, ONERA) ................................................................. 44
Fig. 5.11: Flow past a cylinder at Re=10000 (Photograph by Thomas Corke and Hasan Najib, Illinois Institute of Technology, Chicago) ............................................................................................................. 45
Fig. 5.12: Flow regimes at a circular cylinder ....................................................................................................................... 46
Fig. 6.1: Elements used in a mesh ........................................................................................................................................ 47
Fig. 6.2: Example of structured mesh ................................................................................................................................. 48
Fig. 6.3: Example of unstructured mesh ............................................................................................................................... 48
Fig. 6.4: Example of hybrid mesh ........................................................................................................................................ 48
Fig. 6.5: Example of inflation ................................................................................................................................................ 50
Fig. 7.1: Types of boundary conditions ............................................................................................................................... 51
Fig. 8.1: Solar module ....................................................................................................................................................... 56
Fig. 8.2: Sensor K3D120 ..................................................................................................................................................... 57
List of figures

Fig. 8.3: Base of the solar module in the wind tunnel ................................................ 57
Fig. 8.4: Wind velocity profile .................................................................................. 58
Fig. 8.5: Forces in the x direction measured by the wind tunnel ............................. 58
Fig. 8.6: Forces in the z direction for each support measured by the wind tunnel ...... 59
Fig. 8.7: Forces in the z direction measured by the wind tunnel ............................. 59
Fig. 8.8: Solar module geometry ............................................................................... 60
Fig. 8.9: Automatic mesh cut .................................................................................... 61
Fig. 8.10: Automatic mesh cut in sections with smaller element size ..................... 61
Fig. 8.11: Automatic mesh with smaller element size in the near-wall regions ........ 62
Fig. 8.12: Detail of automatic mesh with smaller element size in the near-wall regions 62
Fig. 8.13: Automatic mesh with smaller element size in the near-wall regions .......... 63
Fig. 8.14: Mesh with sweep and automatic method .................................................... 63
Fig. 8.15: Boundary conditions .................................................................................. 64
Fig. 8.16: Inlet velocity profile .................................................................................. 64
Fig. 8.17: Guidelines for checking mesh quality ....................................................... 66
Fig. 8.18: Mesh quality parameters .......................................................................... 66
Fig. 8.19: Streamlines around the solar module for $v_{ref} = 4 m/s$ ....................... 67
Fig. 8.20: Streamlines around the solar module for $v_{ref} = 4 m/s$ ....................... 68
Fig. 8.21: Velocity of the fluid around the solar module for $v_{ref} = 4 m/s$ .............. 68
Fig. 8.22: Velocity of the fluid around the solar module for $v_{ref} = 4 m/s$ .............. 69
Fig. 8.23: Pressure over the solar module for $v_{ref} = 4 m/s$ .................................... 69
Fig. 8.24: Forces in the x direction ............................................................................ 70
Fig. 8.25: Forces in the z direction ............................................................................ 70
Fig. 8.26: Forces in the x direction ............................................................................ 71
Fig. 8.27: Forces in the z direction ............................................................................ 71
1. Introduction and objectives

The actual developments in construction engineering make essential the study of the fluid flow around the structures. As they are becoming bigger and more slender, the wind loading has become an important decisive factor in the design of structures. By now, these studies have been carried out using wind tunnel experiments, which is a great tool in structure design.

The objective to achieve in computational fluid dynamics is to calculate an entire flow field either around an arbitrary obstacle or through a channel of any shape. The equations to describe that are the Navier-Stokes equations, but, at present, no computer has the capacity or the calculation speed necessary to fulfill this task. Thereby, the governing equations have to be simplified.

Before CFD, the fluid dynamics used to use pure theory and pure experiments. Until 1960, the advancements in fluid mechanics were made with a combination of experiments and theoretical analyses, analyses which always required the use of a simplified model to obtain a closed-form solution of the governing equations. These solutions have the disadvantage of not including all the requisite physics of the flow.

In the early 60’s the appearance of Computational Fluid Dynamics (CFD), let us predict the fluid fields by using numerical methods and algorithms. This method seems to be a great advance in fluid mechanics studies, as it offers us the possibility of study the fluid flow around and object only by means of computational calculations obtaining more exact solutions without the need of expensive wind tunnel experiments. The use of Computational Fluid Dynamics to predict the flow has risen dramatically in the last years.

Of course, the instrument which has allowed the practical growth of CFD is the high speed digital computer. CFD solutions generally require the repetitive manipulation of thousands, or even millions, of numbers, a task that is humanly impossible without the aid of a computer. Therefore, advances in CFD, and its application to problems of more and more detail and sophistication, are intimately related to advances in computer hardware, particularly in regard to storage and execution speed.

One of the most important problems in all classical physics is turbulence. Turbulence is a complex process difficult to solve. The solutions of turbulent flows are one of the main deficiencies of CFD. CFD goes through it by using turbulence models, which are just approximations of the real physics, that is why the CFD solutions of turbulent flows are subject to inaccuracy. There is a big list of different turbulence models which have been introduced by experimented scientists. It is then important to get to know which one to use in each case. But, can we rely on the solutions obtained by means of CFD?
CFD is nowadays, a method in which studies are still going on. To make studies done by CFD reliable as wind tunnel experiments it is important to validate its results by comparing them with wind tunnel results.

The objective of this study is to present the key aspects of computational fluid mechanics from a standard user point of view. There will be no focus on long theoretical explanations, but rather a comprehensive listing of the main milestones in the process of achieving a reliable solution.

After a short introduction of the physics of the problem to be solved (see chapter 2) the focus of the study will be set on the decisive aspects of any CFD calculation: turbulence modeling (chapter 3 and 4), boundary layer and wall treatment (chapter 5), meshing (chapter 6), definition of boundary conditions (chapter 7) and experimental validation (chapter 8).

Out of the scope of this work remains another aspect of CFD such as the solver. Different methodologies such as finite differences, finite volumes and finite elements exist. For this work a finite-element based solver (Ansys CFX) will be used.

Special attention will be paid to the problem of the turbulence and its modeling. A short presentation of the most important turbulence models used by CFD will be done, as well as a comparison between them in order to get to know which one should be used in each case.

After some practical comments regarding meshing, wall treatment and boundary conditions, validation will be performed based on wind tunnel experiments.

For that purpose the study of the uplift forces in a lightweight solar module will be carried out. The process to generate a suitable model of it will be explained. A comparison between different meshing options offered by the CFD will be performed and the results will be compared to the ones obtained by wind tunnel experiments.
2. Previous concepts

In this chapter some basic concepts will be introduced in order to understand the physics of the present work. The basic equations governing the fluid flow will be presented as well as the basic conservation principles and laws used to obtain them.

A fluid is a substance which deforms continuously under an applied shear stress, no matter how small this is. In this study a Newtonian incompressible fluid will be used. In the case of Newtonian fluids there is a linear relationship between the stress and rate of strain so that:

\[ \tau = \mu \frac{du}{dy} \]  

(2.1)

Where

\( \tau \) : shear stress
\( \mu \) : viscosity
\( \frac{du}{dy} \) : velocity gradient

\( \nu = \frac{\mu}{\rho} \) : kinematic viscosity
\( \rho \) : density

One of the most important properties of a fluid is the viscosity. The viscosity is a property of the fluid, which measures the resistance of the fluid of being deformed by the shear stress. The fluid adheres to the wall, so that its velocity in the wall’s surface is zero. In many flows, the effects of viscosity are important only near walls, so that the flow in the largest part of the domain is considered as inviscid.

Due to the three fundamental physic principles which all fluid dynamics satisfy,

1. The mass of a fluid is conserved
2. The momentum is conserved (Newton’s second law)
3. The Energy is conserved

the governing equations of fluid dynamics can be obtained. (Ferziger and Peric, 2002) (Versteeg and Malalasekera, 1995).
1) Conservation of mass:

\[
\frac{Dm}{Dt} = 0 \tag{2.2}
\]

The mass balance for a fluid element can be written down as:

| Rate of increase of mass in fluid element | Net rate of flow of mass into fluid element |

After doing some mathematical calculations we get to the **continuity equation**:

\[
\frac{D\rho}{Dt} + \rho \nabla \cdot \vec{u} = 0 \tag{2.3}
\]

For an incompressible fluid, where \( \rho \) is a constant, the equation becomes

\[
\nabla \cdot \vec{u} = 0 \tag{2.4}
\]

2) Conservation of momentum:

Newton’s second law states that the rate of change of momentum of a fluid particle is equal to the sum of the forces on the particles.

| Rate of increase of momentum of fluid particle | Sum of forces on fluid particle |

The momentum equations can be written as:

\[
\frac{\partial (\rho \vec{u})}{\partial t} + \rho (\vec{u} \cdot \nabla) \cdot \vec{u} = -\nabla p + \nabla \cdot \tau + F \tag{2.5}
\]

These are the well-known Navier-Stokes equations which describe how the velocity, pressure, temperature and density of a moving fluid are related. One the left-hand side of the equation are the derivative over time and the internal forces (convection term). In the right-hand side we can see the gradient of the pressure, the viscous forces and the gravitational forces.

Very often the term of Navier-Stokes equations is applied as well to the group of equations containing the Navier Stokes equations themselves and the continuity equation and the energy equation (presented in the following).
3) Conservation of energy:

The first law of thermodynamics states that the rate of change of energy of fluid particle is equal to the rate of heat addition to the fluid particles plus the rate of work done on the particle.

\[
\frac{DE}{Dt} = -\nabla \cdot (p\tilde{u}) + \left[ \frac{\partial (u\tau_{xx})}{\partial x} + \frac{\partial (u\tau_{yx})}{\partial y} + \frac{\partial (u\tau_{zx})}{\partial z} + \frac{\partial (v\tau_{xy})}{\partial x} + \frac{\partial (v\tau_{yy})}{\partial y} + \frac{\partial (v\tau_{zy})}{\partial z} \right] + \nabla \cdot (k\nabla T) + S_E \tag{2.6}
\]

Where \( E = e + \frac{u^2}{2} \), with \( e \) the sum of internal energy, \( k \) is the thermal conductivity and \( S_E \) represents a source of energy.

Taking a look to all of the equations written above some comments and observations can be made.

The governing equations are a system of nonlinear partial differential equations, and they are very difficult to solve analytically. As is well known, there are few exact solutions, and all of these have been obtained introducing simplifying assumptions. To date, there is no general closed-form solution to these equations.

**Summary of equations in fluid mechanics:**

**Continuity equation:**

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \tilde{u}) = 0 \tag{2.7}
\]

**Navier –Stokes equations:**

\[
\rho \left( \frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + w \frac{\partial u}{\partial z} \right) = -\frac{\partial p}{\partial x} + \mu \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right) + F_x \tag{2.8}
\]

\[
\rho \left( \frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + w \frac{\partial v}{\partial z} \right) = -\frac{\partial p}{\partial y} + \mu \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2} \right) + F_y \tag{2.9}
\]

\[
\rho \left( \frac{\partial w}{\partial t} + u \frac{\partial w}{\partial x} + v \frac{\partial w}{\partial y} + w \frac{\partial w}{\partial z} \right) = -\frac{\partial p}{\partial z} + \mu \left( \frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2} \right) + F_z \tag{2.10}
\]
Energy equation:

\[
\rho \frac{DE}{Dt} = -\nabla \cdot (p \vec{u}) + \left[ \frac{\partial (u \tau_{xx})}{\partial x} + \frac{\partial (u \tau_{yx})}{\partial y} + \frac{\partial (u \tau_{zx})}{\partial z} + \frac{\partial (v \tau_{xy})}{\partial x} + \frac{\partial (v \tau_{yy})}{\partial y} \\
+ \frac{\partial (v \tau_{zy})}{\partial z} + \frac{\partial (w \tau_{xz})}{\partial x} + \frac{\partial (w \tau_{yz})}{\partial y} + \frac{\partial (w \tau_{zz})}{\partial z} \right] + \nabla \cdot (k \nabla T) + S_E \tag{2.11}
\]

This is a system of 5 equations with 7 unknowns \((\rho, p, T, e, u, v, w)\). To close the entire system of equations two more equations must be added. The equations we must add are the equation of state which relates the pressure, temperature and density of the fluid and a thermodynamic relation between state variables. These equations depend on each case.
3. Turbulence

The problem of turbulence is one of the most intriguing and important problems in all classical physics. During the 19th and 20th Centuries, this problem has been studied by many physicists and engineers. But we do not understand yet how and why turbulence occurs. That is the reason why a prediction of the turbulent behavior cannot be done with reliability. Turbulence is a subject on which still studies are going on.

There are many different definitions of turbulence, but one of the most specific was said by Chapman and Tobak in 1985:

“Turbulence is any chaotic solution to the 3-D 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” (McDonough, 2004).

In 1883, experiments by Reynolds on flow in a pipe showed that the behavior of the flow was characterized by a dimensionless parameter, the Reynolds number. This Reynolds number is a measure of the ratio of inertia forces to viscous forces.

\[
Re = \frac{\text{inertia force}}{\text{viscous force}} = \frac{\rho u^2 L^2}{\mu L}
\]

(3.1)

Where L is the system length-scale.

In the figure (2.1) an eschematic representation of the experiment carried out by Reynolds is shown. There three different states of the fluid flow can be easily determined.

Fig. 3.1: The Reynolds experiment: (a) laminar flow, (b) transitional flow and (c) turbulent flow
The general expression of Reynolds number is:

\[ Re = \frac{\rho u D}{\mu} \]  (3.2)

In the experiment it was observed that at values below the critical Reynolds number the flow is smooth and fluid layers past each other in an orderly way. On the other hand, at values above the critical Reynolds number some complicated events take place and that leads to a change of the flow character. The flow behavior becomes random and chaotic, and the motion becomes unsteady.

Due to this Reynolds number a classification of the fluid flow can be made:

- Laminar flow (for \( Re < 2000 \)): The fluid flows in parallel lines.
- Transitional state (for \( 2000 < Re < 4000 \)): It is a mixture of laminar and turbulent flow,
- Turbulent flow (for \( Re > 4000 \)): The fluid flows in a very irregular way, changing its direction erratically. The flow is unpredictable due to the appearance of eddies.

The turbulent flow can be characterized by the following features:

- Highly unsteady
- Diffusion: there is a rapid process of mixing of the swirling eddies of fluid.
- Three dimensional
- Dissipation: Due to the action of viscosity the kinetic energy of the flow reduces and it is converted into internal energy.
- High Reynolds number
- Vorticity: the vortex stretching mechanism is one of the principal mechanisms by which the intensity of turbulence is increased.

The characteristics of a laminar fluid flow change when a perturbation is introduced and eddies appear. (Ferziger and Peric, 2002).
A laminar flow can be transformed in a turbulent flow as the Reynolds number increases. This happens because small disturbances to the flow are no longer damped by the flow, they begin to grow by taking energy from the original laminar flow. Instability in the flow create eddies.

Fig. 3.2: Creation of eddies behind an object

Fig. 3.3: Example of eddies formed by clouds behind an island
The fluid is stirred and produces large eddies. These large eddies are unstable and interact generating smaller eddies. This smaller eddies break into even more smaller eddies, and so on. This is known as energy cascade.

This energy cascade continues until the Reynolds number is sufficiently small that the eddy motion is stable, and molecular viscosity is effective in dissipating the kinetic energy.

Turbulence can be considered to consist of eddies of different sizes, these eddies are a turbulent motion localized over a region of size $l$, with a velocity $u(l)$ and a timescale $\tau(l) = \frac{l}{u(l)}$. At eddies with a large Reynolds number the direct effects of viscosity are negligibly small. Nevertheless these eddies can become so small that molecular diffusion becomes important and viscous dissipation of energy takes place. The scale at which this happens is the Kolmogorov length scale.

The scales in the energy cascade are uncontrollable, but a division into three categories based on these length scales can be done:

- **Integral length scale:**

  This is the largest scales in the energy spectrum. The eddies there obtain the energy from the mean flow and from each other. They are the eddies which contain the most of the energy, they have a large velocity fluctuation and a low frequency. The length scale of this eddies, $l_0$, is comparable to the flow length scale $L$, and a relation between it, the energy dissipation, $\varepsilon$, and the turbulent kinetic energy, $k$, can be done as follows:

  $$l_0 \propto \frac{k^2}{\varepsilon}$$  \hspace{1cm} (3.3)

  And the Reynolds number associated with these large eddies is defined as:

  $$Re_l = \frac{k^2}{\varepsilon v}$$  \hspace{1cm} (3.4)

- **Kolmogorov length scales:**

  These are the smallest scales and form the viscous sub-layer of the boundary layer (which will be explained in chapter 5). Kolmogorov, in 1941, introduce the idea that the smallest scales of turbulence are similar for every turbulent flow and depend only on the energy dissipation $\varepsilon$ and the kinematic viscosity $v$. Finally, at the Kolmogorov length scales, the turbulence kinetic energy is dissipated into heat through the action of molecular viscosity.
• Taylor micro scales:

These are the intermediate scales between the largest and the smallest scales. In these micro scales there is no dissipation, the energy passes from the largest to the smallest without dissipation.

The studies of the Kolmogorov theory and hypothesis, its length scales and how small do eddies get are quite complicated. For more information see (McDonough, 2004) and (Pope, 2000).
4. Turbulence models

Due to the wide range of length and time scales associated with turbulent flow, the creation of a model for this kind of flows is very difficult or even impossible to carry out. Turbulence models are used to predict the effects of turbulence in fluid flow without resolving all scales of the smallest turbulent fluctuations. These models can be classified by the range of length and time scales that are modeled and the range of length and time scales that are resolved.

![Diagram of turbulence models]

**Fig. 4.1: Extend of modelling for some turbulent models**

4.1. Direct numerical simulation (DNS)

This is not properly a turbulence model, in a DNS the Navier-Stokes equations are numerically solved using a computer. Obviously, this is the most accurate approach to turbulence simulation, since it does not need an averaging or approximation. In this simulation, all of the motions contained in the flow are resolved. But current computers are not sufficiently large and fast to permit the necessary resolution if Re is high or the problem is too complicated, that is the reason why this simulation is only used for small Reynolds numbers.

It can be demonstrated that the number of operations grows as $Re^3$. Nevertheless it is thought that if computers continue developing as fast as they had been doing since now, in some years could be able to solve difficult problems by using the direct numerical simulation.
However, nowadays DNS is useful to obtain very detailed information about the flow and let us understand much better the physics of turbulence. The DNS can be understood as a research tool more than as a design tool. (Ferziger and Peric, 2002)

The following investigations are some of the applications that can be carried out thanks to the DNS:

- Understanding the mechanisms of turbulence production, energy transfer, and dissipations in turbulent flows.
- Simulation of the production of aerodynamics noise.
- Understanding the effects of compressibility on turbulence.
- Understanding the interaction between combustion and turbulence.
- Controlling and reducing drag on a solid surface.

The DNS solutions are also useful in developing turbulence models, such as in LES models.

4.2. Reynolds-averaged Navier-Stokes (RANS)

The idea of this method is to use the Reynolds decomposition to decompose a function into a time-averaged part and a fluctuating part.

Using the Reynolds decomposition, a function \( \tilde{u}(x, t) \) can be expressed as:

\[
\tilde{u}(x, t) = U(x) + u(x, t)
\]  \hspace{1cm} (4.1)

Where \( u(x, t) \) is the fluctuating part, and \( U(x) \) is the mean value independent of time.

Fig. 4.2: Plots of parts of Reynolds decomposition
Considering now the Navier-Stokes equations for incompressible flows \((\rho = \text{cte})\) and without the body-force term:

\[
\nabla \cdot \vec{u} = 0 \tag{4.2}
\]

\[
\rho \left( \frac{\partial \vec{u}}{\partial t} + \vec{u} \cdot \nabla \vec{u} \right) = -\nabla p + \mu \cdot \Delta \vec{u} \tag{4.3}
\]

And applying the Reynolds decomposition it can be obtained:

\[
\rho \left( \frac{\partial}{\partial x_j} (U_i U_j) + \frac{\partial}{\partial x_j} (\bar{u_i} \bar{u_j}) \right) = - \frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_i} \tag{4.4}
\]

Where \(\tau_{ij} = \mu \left( \frac{\partial \bar{u_i}}{\partial x_j} + \frac{\partial \bar{u_j}}{\partial x_i} \right)\) is the mean viscous stress tensor and \(\rho \bar{u_i} \bar{u_j}\) is the Reynolds stress tensor.

The appearance of this tensor is what makes the problem so difficult to solve. The Reynolds stress is a property of the flow so it is dependent on the flow variables themselves. This stress changes from flow to flow and no general relations are available.

The objective of the turbulence models for the RANS equations is to compute the Reynolds stress, which can be done by two main categories of RANS-based turbulence models: the linear eddy viscosity models and the nonlinear eddy viscosity models.

### 4.2.1. Linear eddy viscosity models

In these models the Boussinesq hypothesis, which assumes that the Reynolds stress is proportional to the mean rate of strain, is used:

\[
-\rho \bar{u_i} \bar{u_j} = 2\mu_t S_{ij} - \frac{2}{3} \rho k \delta_{ij} \tag{4.5}
\]

Where the mean strain rate is described as \(S_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) = \frac{1}{3} \frac{\partial u_k}{\partial x_k} \delta_{ij}\)

And \(k\) is the turbulent kinetic energy.

Once this assumption is made, the eddy viscosity, \(\mu_t\), has to be modeled.

The eddy viscosity is derived from turbulent transport equations. Depending on the number of equations solved to compute the eddy viscosity coefficient a classification of the different eddy viscosity models can be done:

- Zero equation models
- One equation models
Turbulence models

- Two equation models

4.2.1.1. Zero equation models

The zero equation models or algebraic models are the ones that do not require the solution of any additional equations. They compute a global value for the turbulence viscosity from the mean velocity and geometric length scale using an empirical formula.

$$\mu_t = \rho f_\mu U_l l_t$$  \hspace{1cm} (4.6)

Where $f_\mu = 0.01$ is a constant and $l_t = \frac{V_D^{\frac{1}{3}}}{7}$ where $V_D$ is the fluid domain volume.

These equations are simple to implement and use and can produce approximate results very quickly. These results are often too simple for use in general situations, but they provide a good initial guess for simulations using more advanced turbulence models.

4.2.1.2. One equation models

One equation turbulence models solve one turbulent transport equation to obtain the value of the eddy viscosity. Prandtl, in 1945, calculated the turbulent viscosity as a function of turbulent kinetic energy $k$. One advantage of this model is that it takes into account the flow history.

The original Prandtl’s model is:

$$\frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_j} = \tau_{ij} \frac{\partial U_i}{\partial x_j} - C_D \frac{k^2}{l} + \frac{\partial}{\partial x_j} \left[ \left( \nu_t + \frac{\nu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right]$$  \hspace{1cm} (4.7)

Where $\nu_t = \frac{\mu_t}{\rho} = k \frac{1}{2} l = C_D \frac{k^2}{\varepsilon}$ is the kinematic eddy viscosity, $C_D$ is a constant with values between 0.7 and 0.9 and $\sigma_k = 1$

4.2.1.3. Two equation models

Two equation models are one of the most common types of turbulence models. They are used in most of the engineering problems. These models include two extra transport equations to represent the turbulent properties of the flow. The turbulent viscosity is modeled as the product of a turbulent velocity and turbulent length scale, which are solved using separate transport equations. The turbulence velocity scale is computed from the turbulent kinetic energy, $k$, and the length scale is obtained from the kinetic energy and the dissipation rate (dissipation, $\varepsilon$, or the specific dissipation, $\omega$).

Using this model only requires introducing the initial and boundary conditions; there is no necessity to have some knowledge about the studied flow in advance.
4.2.1.3.1. \( k - \varepsilon \) models

The transported variables in this case are the kinetic energy \( k \) and the turbulent dissipation \( \varepsilon \).

In this models the eddy viscosity is obtained as:

\[
\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \tag{4.8}
\]

Where \( C_\mu = 0.09 \)

There are different models to obtain the transport variables \( k \) and \( \varepsilon \)

- **Standard \( k - \varepsilon \) model**

The transport equations for this model to obtain the kinetic energy and dissipation are:

\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho U_j k)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + P_k - \rho \varepsilon + P_{kb} \tag{4.9}
\]

\[
\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial (\rho U_j \varepsilon)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + \frac{\varepsilon}{k} \left( C_{\varepsilon 1} P_k - C_{\varepsilon 2} \rho \varepsilon + C_{\varepsilon 1} P_{kb} \right) \tag{4.10}
\]

Where \( C_{\varepsilon 1} = 1.44, C_{\varepsilon 2} = 1.92, \sigma_k = 1 \) and \( \sigma_\varepsilon = 1.3 \)

\[
P_k = \mu_t \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \frac{\partial U_i}{\partial x_j} - \frac{2}{3} \frac{\partial U_k}{\partial x_k} \left( 3 \mu_t \frac{\partial U_k}{\partial x_k} + \rho k \right) \tag{4.11}
\]

\[
P_{kb} = -\frac{\mu_t}{\rho \sigma_b} \frac{\partial \rho}{\partial x_i} \tag{4.12}
\]

\[
P_{eb} = C_3 \cdot \max(0, P_{kb}) \tag{4.13}
\]

\( C_3 = 1 \) and \( \sigma_b = 0.9 \) for Boussinesq buoyancy and \( \sigma_b = 1 \) for full buoyancy model.

- **RNG \( k - \varepsilon \) model**

The RNG \( k - \varepsilon \) model takes into account the different scales of motion by changing the constants of the Standard \( k - \varepsilon \) model, but using the same equations for \( k \) and \( \varepsilon \).
The constant $C_{e1}$ is changed to a function $C_{e1 RNG}$ and the constant $C_{e2}$ to the value $C_{e2 RNG}=1.68$

$$C_{e1 RNG} = 1.42 - f_\eta$$

With $f_\eta = \frac{\eta(1 - \frac{\eta}{4.38})}{(1 + \beta RNG \eta^2)}$, $\eta = \sqrt{\frac{p_k}{\rho C_{\mu RNG} \varepsilon}}$ and $C_{\mu RNG} = 0.085$

In general, turbulence models based on the $\varepsilon$-equation predicts the onset of separation too late and under predicts the amount of separation later on. These models are not suitable for:

- Flows with boundary layer separation
- Flows with sudden changes in the mean strain rate
- Flows in rotating fluids
- Flows over curved surfaces

### 4.2.1.3.2. $k - \omega$ models

The transported variables in this case are the kinetic energy $k$ and the specific dissipation $\omega$. One of the main problems in turbulence modeling is the accurate prediction of flow separation from a smooth surface. The $k - \omega$ models try to avoid the problems of the $k - \varepsilon$ models. The advantage of the $k - \omega$ formulation is the near wall treatment for low-Reynolds number.

Using these models the eddy viscosity is obtained as:

$$\mu_t = \rho \frac{k}{\omega} \quad (4.14)$$

- Wilcox $k - \omega$ model

The transport equations used in this model to obtain the turbulence kinetic energy and the specific dissipation rate are:

$$\frac{\partial (\rho k)}{\partial t} + \frac{\partial}{\partial x_j} (\rho U_j k) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \mu_t \sigma_k \right) \frac{\partial k}{\partial x_j} \right] + P_k - \beta' \rho k \omega + P_{kb} \quad (4.15)$$

$$\frac{\partial (\rho \omega)}{\partial t} + \frac{\partial}{\partial x_j} (\rho U_j \omega) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \mu_t \sigma_\omega \right) \frac{\partial \omega}{\partial x_j} \right] + \alpha \frac{\omega}{k} P_k - \beta \rho \omega^2 + P_{\omega b} \quad (4.16)$$

Where $\beta' = 0.09$, $\alpha = 5/9$, $\beta = 0.075$, $\sigma_k = 2$, $\sigma_\omega = 2$ and $P_{\omega b} = \frac{\omega}{k} ((\alpha + 1)C_3 \max(P_{kb}, 0) - P_{kb})$
• The Baseline $k - \omega$ model (BSL $k - \omega$)

The main problem with the Wilson $k - \omega$ models is its strong sensitivity to freestream conditions. The BSL $k - \omega$ models solve this problem blending the $k - \omega$ models and $k - \varepsilon$ models.

The resultant transport equations are:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial}{\partial x_j}(\rho U_j k) = \frac{\partial}{\partial x_j}\left[(\mu + \frac{\mu_t}{\sigma_{k3}}) \frac{\partial k}{\partial x_j}\right] + P_k - \beta' \rho k \omega + P_{kb} \tag{4.17}$$

$$\frac{\partial(\rho \omega)}{\partial t} + \frac{\partial}{\partial x_j}(\rho U_j \omega) = \frac{\partial}{\partial x_j}\left[(\mu + \frac{\mu_t}{\sigma_{\omega3}}) \frac{\partial \omega}{\partial x_j}\right] + (1 - F_t)2\rho \frac{1}{\sigma_{\omega2}} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} + \frac{\alpha_3}{k} P_k - \beta_3 \rho \omega^2 + P_{\omega b} \tag{4.18}$$

• The Shear stress transport $k - \omega$ model (SST $k - \omega$)

Taking into account the different regions of a boundary layer, which will be explained in chapter 5, a comparison between the $k - \omega$ models and the $k - \varepsilon$ models can be done as follows:

<table>
<thead>
<tr>
<th>Model</th>
<th>$k - \omega$</th>
<th>$k - \varepsilon$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Sublayer</td>
<td>Robust</td>
<td>Stiff</td>
</tr>
<tr>
<td></td>
<td>Simple</td>
<td>Less accurate</td>
</tr>
<tr>
<td></td>
<td>Accurate</td>
<td>Complex</td>
</tr>
<tr>
<td>Log- Layer</td>
<td>Accurate</td>
<td>Large length scales</td>
</tr>
<tr>
<td>Wake Region</td>
<td>Missing transport effects</td>
<td>Missing transport effects</td>
</tr>
<tr>
<td>Boundary layer edge</td>
<td>Free-stream sensitive</td>
<td>Well defined</td>
</tr>
</tbody>
</table>

It can be seen, that the $k - \omega$ models have a higher quality in the inner part of the boundary layer and they give worst results as we move away from it. On the other hand the $k - \varepsilon$ models have a higher quality in the outer part of the boundary layer and they reduce their accuracy as they get closer to the inner part of the boundary layer.
The SST model uses this knowledge to obtain high quality results combining the use of both models. It uses a $k - \omega$ formulation in the inner parts of the boundary layer and switches to a $k - \varepsilon$ behavior in the freestream and thereby avoids the common $k - \omega$ problem that the model is too sensitive to the inlet freestream turbulence properties. This model gives a high accurate prediction of the onset.

The eddy viscosity is computed as:

$$\mu_t = \frac{a_1 k \rho}{\max(a_1 \omega, SF_2)}$$  \hspace{1cm} (4.19)

$F_2$ is a blending function which restricts the limiter to the wall boundary layer:

$$F_2 = \tanh(\max(\frac{2\sqrt{k}}{\beta' \omega y}, \frac{500 \nu}{y^2})^2)$$  \hspace{1cm} (4.20)

$$S = \sqrt{2S_{ij} S_{ij}}$$  \hspace{1cm} (4.21)

$$S_{ij} = \frac{1}{2} \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right)$$  \hspace{1cm} (4.22)

$$a_1 = 0.31$$  \hspace{1cm} (4.23)

And the resultant transport equations are:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial}{\partial x_j}(\rho U_j k) = \frac{\partial}{\partial x_j} \left[ (\mu + \sigma' \mu_t) \frac{\partial k}{\partial x_j} \right] + P_k - \beta' \rho k \omega$$  \hspace{1cm} (4.24)

$$\frac{\partial(\rho \omega)}{\partial t} + \frac{\partial}{\partial x_j}(\rho U_j \omega) = \frac{\partial}{\partial x_j} \left[ (\mu + \sigma_\omega \mu_t) \frac{\partial \omega}{\partial x_j} \right] + (1 - F_1) 2 \sigma_\omega \frac{1}{\omega} \frac{\partial k}{\partial x_i} \frac{\partial \omega}{\partial x_j} + \alpha S^2 - \beta \omega^2$$  \hspace{1cm} (4.25)

Where $\sigma' = 0.85$, $P_k = \tau_{ij} \frac{\partial U_i}{\partial x_j}$, $\sigma_\omega = 0.856$

$$F_1 = \tanh \left( \min \left( \max \left( \frac{\sqrt{k}}{\beta' \omega y}, \frac{500 \nu}{\gamma^2 \omega}, \frac{4 \sigma_\omega}{CD_k \omega y^2} \right) \right) \right)^4$$  \hspace{1cm} (4.26)

$$CD_{k\omega} = \max(2 \rho \sigma_\omega \frac{1}{\omega} \frac{\partial k}{\partial x_i} \frac{\partial \omega}{\partial x_j}, 10^{-10})$$  \hspace{1cm} (4.27)
This model was developed to overcome deficiencies in the other models, that is why it is recommended to use the SST model over the $k-\omega$ and the BSL $k-\omega$ model.

### 4.2.2. Nonlinear eddy viscosity models or Reynolds Stress models (RSM)

Linear eddy viscosity models fail in some flow situations. In flows where the turbulent transport or non-equilibrium effects are important, the eddy-viscosity assumption is no longer valid. The RSM are nonlinear eddy viscosity models that offer better predictions than linear eddy viscosity models by increasing moderately the computing resources.

In these models the Reynolds stresses are directly computed. The Reynolds stress model involves calculation of the individual Reynolds stresses using differential transport equations. The individual Reynolds stresses are then used to obtain closure of the Reynolds-averaged momentum equation.

\[
\frac{\partial \rho U_i}{\partial t} + \frac{\partial}{\partial x_j} (\rho U_i U_j) - \frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \right] = - \frac{\partial}{\partial x_i} \frac{\partial \rho v^\prime v^\prime}{\partial x_j} \frac{\rho U_i U_j}{\partial x_j} 
\] (4.28)

Where

\[
p^{\prime\prime} = p + \frac{2}{3} \mu \frac{\partial U_k}{\partial x_k}
\] (4.29)

The differential equation Reynolds stress transport is:

\[
\frac{\partial \rho \bar{u}_i \bar{u}_j}{\partial t} + \frac{\partial}{\partial x_k} (U_k \rho \bar{u}_i \bar{u}_j) - \frac{\partial}{\partial x_k} \left( \delta_{ki} \mu + \rho C_s \frac{k}{\epsilon} \bar{u}_i \bar{u}_j \right) \frac{\partial \bar{u}_i \bar{u}_j}{\partial x_k} 
\]  
\[= P_{ij} - \frac{2}{3} \delta_{ij} \rho \epsilon + \phi_{ij} + P_{ij,b} \] (4.30)

Where $P_{ij}$ and $P_{ij,b}$ are the shear and buoyancy turbulence terms of the Reynolds stresses, $\phi_{ij}$ is the pressure-strain tensor, and $C_s = 0.22$.

These models can be classified into:

- The Reynolds stress model, which is based on the $\varepsilon$-equation.*
- The Omega-based Reynolds stress model, which is based on the $\omega$-equation. The advantage of this model is that it avoids the $\varepsilon$-equation problems, as said before, the turbulence models based on the $\varepsilon$-equation predicts the onset of separation too late and under predicts the amount of separation.*
- The explicit algebraic Reynolds stress model (EARS), which is derived from the Reynolds stress transport equation and give a nonlinear relation between the Reynolds stresses and the mean strain-rate and vorticity tensors.*
Turbulence models

(*)For more information about the equations to use in these models see (Ansys CFX Solver theory guide, chapter 2)

These models are appropriated for:
- Free shear flows with strong anisotropy, like a strong swirl component. This includes flows in rotating fluids.
- Flows with sudden changes in the mean strain rate
- Flows where the strain fields are complex, and reproduce the anisotropic nature of turbulence itself
- Flows with strong streamline curvature
- Secondary flow
- Buoyant flow

4.3. Large Eddy Simulation (LES)

This method is based on the consideration that the large eddies are flow-dependent and the small scales are more universal and independent from what is happening on the larger scales. Making this assumption the large eddies are computed accurately and the small ones are modeled. DNS gives the best results, but as will be explained, it requires high computational costs, that is why doing a separation of the eddies and using a LES model can reduce the computational costs without highly decreasing the quality in the results. (Menter, 2010)

Fig. 4.3: Representation of turbulent motion.

In the large eddy simulation a separation between large and small scales is done.

\[ u(x, t) = \bar{U}(x, t) + u'(x, t) \]  

(4.31)
Turbulence models

Where $\tilde{U}$ is the large-scale part of the solution and $u'$ is the small-scale, or subgrid-scale part.

To obtain the governing LES equations it is necessary to filter the Navier-Stokes equations. The filter separates the small eddies from the large ones.

$$\tilde{U}(x, t) = \int_D u(\xi, t) G(x; \xi) d\xi$$  \hspace{1cm} (4.32)

Where D is the fluid domain and G is the filter function that determines the scale of the resolved eddies.

Once the Navier-Stokes equations are filtered we obtain:

$$\frac{\partial \tilde{U}_i}{\partial t} + \frac{\partial}{\partial x_j} (\tilde{U}_i U_j) = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} [\nu \left( \frac{\partial \tilde{U}_i}{\partial x_j} + \frac{\partial \tilde{U}_j}{\partial x_i} \right)] - \frac{\partial \tau_{ij}}{\partial x_j}$$  \hspace{1cm} (4.33)

Where $\tau_{ij}$ is the subgrid-scale stress and is defined by:

$$\tau_{ij} = \tilde{U}_i \tilde{U}_j - \tilde{\mu}_{ij}$$  \hspace{1cm} (4.34)

The large scale turbulent flow is solved directly and the small scales are modeled with an appropriate subgrid-scale (SGS) model. The relation between the subgrid-scale stresses $\tau_{ij}$ and the large-scale strain rate tensor $\bar{S}_{ij}$ is:

$$- \left( \tau_{ij} - \frac{\delta_{ij}}{3} \tau_{kk} \right) = 2 \nu_{sgs} \bar{S}_{ij}$$  \hspace{1cm} (4.35)

$$\bar{S}_{ij} = \frac{1}{2} \left( \frac{\partial \tilde{U}_i}{\partial x_j} + \frac{\partial \tilde{U}_j}{\partial x_i} \right)$$  \hspace{1cm} (4.36)

Where $\nu_{sgs}$ is the subgrid-scale viscosity which needs to be modeled.

These models are suitable in cases where:

- The flow is likely to be unstable, with large scale flapping of a shear layer or vortex shedding.
- The flow is likely to be unsteady with coherent structures.
- The flow is buoyant, with large unstable regions created by heating from below, or by lighter fluid below heavier fluid.
- Conventional RANS approach are known to fail.
- A good representation of the turbulent structure is required for small-scale processes such as micromixing or chemical reaction.
- The noise from the flow is to be calculated, and especially when the broadband contribution is significant.
- Other fluctuating information is required.

There are different models to obtain the subgrid-scale viscosity, \( v_{sgs} \).

### 4.3.1. Smagorinsky model

The Smagorinsky model was the earliest and is now the most commonly used model to obtain the subgrid-scale viscosity. The subgrid-scale viscosity is calculated with the equation:

\[
    v_{sgs} = (C_s \Delta)^2 |\vec{S}|
\]

Where \( \Delta = (\text{Vol})^{\frac{1}{3}} \), \( |\vec{S}| = \sqrt{2S_{ij}S_{ij}} \) and \( C_s = 0.18 \)

### 4.3.2. WALE model

The wall-adapted local eddy-viscosity model uses the following equation:

\[
    v_{sgs} = (C_w \Delta)^2 \frac{(S_{d}^d)^{\frac{3}{2}}}{(S_{ij}^dS_{ij})^{\frac{3}{2}} + (S_{ij}^dS_{ij})^{\frac{3}{2}}} \]

Where \( S_{ij}^d = \frac{1}{2}(\bar{S}_{ij}^2 + \bar{S}_{ji}^2) - \frac{1}{3} \delta_{ij} \bar{S}_{kk}^2 \) and where \( \bar{S}_{ij}^2 = \bar{g}_{ik}\bar{g}_{kj} \bar{g}_{ij} = \frac{\partial \bar{U}_i}{\partial S_j} \)

### 4.3.3. Dynamic Smagorinsky-Lilly model

A modification of the Smagorinsky model was introduced by Germano et al., 1991, and Lilly, 1992, which let the before constant \( C_s \) vary in time and space, and it is now called \( C_d \). The value of \( C_d \) is calculated in each time step based upon two filtering of the flow variables. So we have:

\[
    L_{ij} = T_{ij} - \{\tau_{ij}\}
\]

Where \( \tau_{ij} \) represents the subgrid-scale stress at scale \( \Delta \) and \( T_{ij} \) represents the SGS stress at scale \( \{\Delta\} \)

\[
    \tau_{ij} = \bar{U}_i\bar{U}_j - \bar{U}_i\bar{U}_j
\]

\[
    T_{ij} = \{\bar{U}_i\bar{U}_j\} - \{\bar{U}_i\}\{\bar{U}_j\}
\]

And \( \{\ldots\} \) denotes secondary filtering of a quantity.
\[ C_d = \frac{L_{ij} M_{ij}}{M_{ij} M_{ij}} \]  \hspace{1cm} (4.42)

\[ \tau_{ij} = \frac{d_{ij}}{3} \tau_{kk} = C_d (-2 \Delta^2 |\vec{S}_{ij}|) = C_d m_{ij}^{\text{sgs}} \]  \hspace{1cm} (4.43)

\[ T_{ij} = \frac{\delta_{ij}}{3} T_{kk} = C_d (-2 \|\vec{S}_{ij}\| |\vec{S}_{ij}|) = C_d m_{ij}^{\text{test}} \]  \hspace{1cm} (4.44)

\[ L_{ij} = L_{ij} - \frac{\delta_{ij}}{3} L_{kk} = C_d m_{ij}^{\text{test}} - \{C_d m_{ij}^{\text{sgs}}\} \]  \hspace{1cm} (4.45)

\[ M_{ij} = m_{ij}^{\text{test}} - \{m_{ij}^{\text{sgs}}\} \]  \hspace{1cm} (4.46)

Using the new coefficient \( C_d \), the eddy viscosity is obtained by:

\[ \nu_{sgs} = C_d \Delta^2 |\vec{S}_{ij}| \]  \hspace{1cm} (4.47)

### 4.4. Detached eddy simulation model (DES)

This model is a hybrid which combines RANS and LES methods. It is based on the idea of covering the boundary layer by a RANS model and using a LES model in detached regions as the use of LES in boundary layer flows at high \( \text{RE} \) numbers is expensive. The main problem of this model is to get to determine where the match should occur and how to match the solutions. The time resolution for this model imposes high CPU demands.

This model is appropriate in the following cases:

- Flow around non-aerodynamic obstacles (building, bridges,…).
- Flow around ground transport vehicles with massively separated regions (cars, trains, trucks,…).
- Flow around noise generating obstacles (car side-mirror,…).
- Massively separated flow around stalled wings.

### 4.5. SST-DES formulation

This model switch from the SST-RANS model to a LES model in regions where the turbulent length scale predicted by the RANS model is larger than the local grid spacing, \( \Delta \).
4.6. Summary

After having presented the different turbulence models that are used nowadays some specific recommendations when choosing a model can be done:

- For free shear flows or for boundary layer in equilibrium the differences between the models are fairly small.
- For flows which need a high accuracy in boundary layers a SST model should better be used.
- For flows with strong swirl a Reynolds Stress model should be used.

As shown in the Figure 4.1 above, DNS and LES models resolve shorter length scales than RANS models. That is why better results can be obtained by using these models over the models that use RANS equations. Nevertheless, as it is expected, the more detailed solutions we want to obtain, the higher computational cost we need. It is then necessary to do an evaluation between the degree of the detail we want to obtain and the computational effort that we are willing to assume.

For most of the CFD simulations used as a design tool, there is no necessity to get to the degree of detail that DNS and LES offer. Generally, the results obtained by using RANS models are detailed enough. That is why RANS models are nowadays the most widely used models, they offer a significant degree of detail without demanding higher computational costs.
5. Boundary layer

The boundary layer is a thin region on the surface of a body in which viscous effects are important. The effect of friction near the wall causes the fluid immediately adjacent to the surface to stick to it, this effect takes place only in a thin region near the surface, the boundary layer. The velocity of the fluid at the surface is zero and it increases enormously within a thin layer until it reaches the free stream value of the velocity away from the surface. This region of very large velocity gradients is the boundary layer. (Anderson, 2005), (Schlichting, 1955).

There are laminar boundary layers and turbulent boundary layers depending on the Reynolds number. For lower Reynolds numbers, the boundary layer is laminar and the velocity changes uniformly as one moves away from the wall. For higher Reynolds numbers, the boundary layer is turbulent and the velocity is characterized by unsteady swirling flows inside the boundary layer.

The laminar boundary layer can change to a turbulent boundary layer when the external velocity is sufficiently large. The transition from laminar to turbulent flow in the boundary layer is clearly seen by a sudden increase in the boundary layer thickness.

![Fig. 5.1: Comparison of laminar and turbulent boundary layers](image)

5.1. Boundary layer equations

In the boundary layer occurs that:

- The diffusive transport of momentum in the principal flow direction is much smaller than convection, and so, it can be neglected.
- The velocity component in the main flow direction is much larger than the components in other directions.
- The pressure gradient across the flow is much smaller than in the principal flow direction.

Therefore, the Navier-Stokes equations can be simplified for a steady two dimensional, laminar and incompressible flow as follows:
\[ u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = -\frac{\partial p}{\partial x} + \frac{1}{Re} \frac{\partial^2 u}{\partial y^2} \]  

(5.1)

With the continuity equation being:

\[ \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \]  

(5.2)

The boundary conditions to be imposed are:

On the surface:

\[ y = 0, \quad u = 0, \quad v = 0 \]

On the outer edge of the boundary layer:

\[ y = \delta, \quad u = U(x) \]

5.2. **Boundary layer thickness**

The boundary layer thickness, \( \delta \), is defined as the distance, starting from the surface, that the flow needs to nearly achieve the free stream velocity, \( u_0 \). It is said that the boundary layer ends when the velocity of the flow reaches 99% of the free stream velocity

\[ u(y) = 0.99u_0 \]  

(5.3)

The boundary layer thickness increases along the body in downstream direction, as seen in figure (5.2).

![Fig. 5.2: Boundary layer thickness and velocity profile](image)

There are no general equations for the boundary layer thickness. There are only expressions for some specific cases of boundary layer.

For laminar plate boundary layers the boundary layer thickness can be easily estimated. In the boundary layer there is equilibrium between the inertial forces and the friction forces.
\[ \rho u \frac{\partial u}{\partial x} = \frac{\partial \tau}{\partial y} \quad (5.4) \]

Taking into account that for a plate \( \frac{\partial u}{\partial x} \) is proportional to \( \frac{u_0}{x} \), the inertial forces in a plate are proportional to \( \rho \frac{u_0^2}{x} \). On the other hand the friction forces in a laminar flow are \( \frac{\partial \tau}{\partial y} = \mu \frac{\partial^2 u}{\partial y^2} \) and as \( \frac{\partial u}{\partial y} \) is proportional to \( \frac{u_0}{\delta} \) the friction forces in this case are proportional to \( \mu \frac{u_0^2}{\delta^2} \).

Arranging equation (5.4) it can be obtained that the laminar boundary layer thickness is:

\[ \delta \propto \sqrt[\mu]{ux} = \frac{x}{\sqrt{Re_x}} \quad (5.5) \]

In the case of the laminar boundary layer over a flat plate the boundary layer thickness is:

\[ \delta = \frac{5.2x}{Re_x^{1/2}} \quad (5.6) \]

Where \( x \) is the distance downstream from the start of the boundary layer, and

\[ Re_x = \frac{\rho_ux}{\mu} \]

With equation (5.5) it is easy to see that the boundary layer thickness increases proportionally to \( \sqrt{x} \) and it decreases proportionally to \( \sqrt{Re} \), so in cases with large Reynolds number the boundary layer thickness vanishes.

In the case of a turbulent boundary layer over a flat plate the boundary layer thickness is:

\[ \delta = \frac{0.382x}{Re_x^{1/3}} \quad (5.7) \]
5.3. **Boundary sub-layers**

The boundary layer can be divided into four sub-layers (McDonough, 2004):

![Figure 5.3: Representation of the sub-layers in a turbulent boundary layer](image)

1. **Linear or viscous sub-layer:**

   At the solid surface the fluid is stationary. As there are no turbulent Reynolds shear stress effects, the viscous stresses dominate the flow adjacent to the surface and so friction must be taken into account. This layer is extremely thin ($y^+ < 5$) and it is often assumed that the shear stress is approximately constant and equal to the wall shear stress. It can finally be demonstrated that $u^+ = y^+$.

   Where $y^+$ is the dimensionless distance from the wall $y^+ = y \frac{u_x}{v}$, $u^+ = \frac{u(y)}{u_x}$, $u_x = \sqrt{\frac{\tau_w}{\rho}}$, $\tau_w$ is the wall shear stress and $y$ the distance from the wall.

2. **Buffer layer:**

   This layer connects the viscous sub-layer to the inertial sub-layer. In this layer, inertial and dissipation effects are nearly balanced. This layer goes from $y^+ = 5$ to $y^+ = 30$.

3. **Inertial or log-law layer:**

   In this layer the turbulent Reynolds stresses dominate the flow. The velocity follows the log-law:

   $$u^+ = \frac{1}{k} ln(y^+) + B \quad (5.8)$$

   Where $k$ is the von Kármán constant, $k = 0.41$, and $B$ is an empirical constant with a value of $B = 5.5$ in the case of smooth walls, this value decrease in the case of wall roughness.

   This layer goes from $y^+ = 30$ to $y^+ = 500$
4. Outer or defect layer:

The log–law is valid in the region \(0.02 < \frac{y}{\delta} < 0.2\). For larger values of \(y\), the wake contribution needs to be taken into account, so the mean velocity profile over the whole boundary layer is well predicted by the sum of the law of the wall and the law of the wake. The correct law in this region is the velocity-defect law:

\[
u^+ = \frac{1}{k} \ln \left( \frac{y}{\delta} \right) + \frac{\Pi}{k} \left( W \left( \frac{y}{\delta} \right) - 2 \right)
\]

(5.9)

Where \(\Pi\) is the wake strength parameter and is flow dependent and \(W \left( \frac{y}{\delta} \right)\) is the wake function which is supposed to be universal and a convenient approximation for it is \(W \left( \frac{y}{\delta} \right) = 2 \sin^2 \left( \frac{\pi y}{2 \delta} \right)\).

In this region, the viscosity can be neglected, and the flow corresponds to the inviscid limiting solution.

5.4. Separation of the boundary layer

When the fluid particles in the boundary layer are sufficiently decelerated by the inertial forces boundary layer separation occurs. The flow near the surface reverses its direction and flows upstream and streamlines meet and then leave the surface. The presence of adverse pressure gradients in the flow produces these deceleration effects.

This phenomenon is associated with the generation of vortices, which are swirl into the wake flow behind the bluff body, and large energy losses.

The position of separation is given by the condition that the velocity gradient perpendicular to the wall vanishes at the wall, that means, the wall shear stress vanishes:
Fig. 5.5: Boundary layer separation

5.5. Example of a fluid flow over a cylinder.

In order to understand the phenomenon of separation the case of a fluid flow over a cylinder will be explained. (Schlichting, 1955), (Srinivasa, 2010), (Tutty, Price and Parsons, 2002).

Using ANSYS CFD the case of fluid flow over a cylinder, with a diameter of 0.5m was studied. Doing that, the results could then be compared to the theoretical and experimental results of this commonly studied case.

Setting different values of the velocity of the fluid flow at the inlet, a wide range of Reynolds numbers can be obtained, and so, the streamline in different cases can be studied.

In this case, for low Reynolds numbers the laminar model has been chosen. There the fluid is set as water at 25°C which has a density of \( \rho = 997 \text{kg/m}^3 \) and a viscosity of \( \nu = 8.9 \times 10^{-4} \text{kg/(m \cdot s)} \). And in the case of higher Reynolds numbers a \( k-\omega \) turbulent model has been chosen. The fluid flow is set as air at 25°C which has a density of \( \rho = 1.185 \text{kg/m}^3 \) and a viscosity of \( \nu = 1.83 \times 10^{-5} \text{kg/(m \cdot s)} \).

In figure (5.6) the mesh with inflation layers in the near wall region and the defined boundary conditions can be seen:
For very low Reynolds number, $Re \sim 1$, the streamlines show no unexpected properties. The flow is symmetrical upstream and downstream.

**Fig. 5.6: Mesh for fluid flow over a cylinder**

**Fig. 5.7: Streamlines for $Re=0.112$**
As Reynolds number increases, the symmetry disappears. Outside the boundary layer as the particles go from D to E there is a transformation of pressure into kinetic energy and a transformation of kinetic energy into pressure occurs as the particles go from E to F. The particles arrive at F with the same velocity as they had at D.

In the boundary layer, the particles have the influence of the same pressure field as that existing outside. But, due to the large friction forces in the boundary layer, the particles consumes more kinetic energy as they go from E to F, that cannot move far in this region. The external pressure causes it then to move in the opposite direction.

There is a circulation behind the sphere. And as a result, eddies are generated behind the cylinder. These eddies get bigger as Reynolds number increases.

For Re>20 small eddies can be seen behind the cylinder.

\[ \text{Fig. 5.8: Streamlines for } Re=22.4 \]

For Re>40 the flow becomes unsteady and there is a sudden change in the character of motion. At a large distance from the body it is possible to identify a regular pattern of vortices which move alternately clockwise and counterclockwise. This is known as the Kármán vortex street. Here, one of the vortices behind the cylinder gets so long that it breaks off and travels downstream with the fluid. Then the fluid curls around behind the cylinder and makes a new vortex.
When Re>100 the eddies are periodically shed from the cylinder to form the vortices of the Kármán vortex street (Figure 5.11). While the eddy on one side sheds from the cylinder the one of the other side is re-forming.

Under Reynolds number from 200 the vortex street continues downstream. Above this value, the vortex street breaks down and produce a turbulent wake. As the vortices travel downstream irregularities in the vortices increase in amplitude. These irregularities become dominant and the wake is turbulent.

At Re~400 the turbulence spreads into the regions between the vortices and disrupts the regular periodicity, and it finally gets to a fully turbulent wake. This continue happening until Reynolds number of about 3x10^5.

**Fig. 5.9: Streamlines for Re=50,41**

**Fig. 5.10: Flow past a cylinder at Re=2000 (Photograph by Werle and Gallon, ONERA)**
Fig. 5.11: Flow past a cylinder at Re=10000 (Photograph by Thomas Corke and Hasan Najib, Illinois Institute of Technology, Chicago)
Table 5.1: Flow regimes at a circular cylinder (incompressible flow)

<table>
<thead>
<tr>
<th>Flow regime</th>
<th>Strouhal number Sr</th>
<th>Flow characteristic</th>
<th>Drag coefficient CD</th>
<th>Separation angle θs</th>
</tr>
</thead>
<tbody>
<tr>
<td>Creeping flow</td>
<td>−</td>
<td>Steady, no wake</td>
<td>−</td>
<td>−</td>
</tr>
<tr>
<td>Vortex pairs in wake</td>
<td>Sr &gt; 0</td>
<td>Steady, symmetric separation</td>
<td>−</td>
<td>−</td>
</tr>
<tr>
<td>Karman vortex street</td>
<td>0.14 &lt; Sr &lt; 0.21</td>
<td>Laminae, unstable wake</td>
<td>−</td>
<td>−</td>
</tr>
<tr>
<td>Pure Karman vortex street</td>
<td>Sr &gt; 0.21</td>
<td>Karman vortex street</td>
<td>0.14 &lt; Sr &lt; 0.21</td>
<td>−</td>
</tr>
<tr>
<td>Subcritical regime</td>
<td>−</td>
<td>Laminae, with vortex street separation</td>
<td>−</td>
<td>−</td>
</tr>
<tr>
<td>Critical regime</td>
<td>−</td>
<td>Laminae separation</td>
<td>−</td>
<td>−</td>
</tr>
<tr>
<td>Supercritical regime (transcritical)</td>
<td>−</td>
<td>Turbulent separation</td>
<td>−</td>
<td>−</td>
</tr>
</tbody>
</table>

Fig. 5.12: Flow regimes at a circular cylinder
6. Meshing

In order to analyze fluid flows, flow domains are split into smaller subdomains. Then, the governing equations are solved in each of these subdomains. These subdomains are the cells that form the mesh. The choice of a mesh has a significant impact on the solution accuracy, the rate of convergence and the time needed to obtain the solution.

There are different types of meshing depending on the elements they are formed of. The most common elements used in a mesh are triangles and squares for a 2-D mesh and hexahedrons, tetrahedrons, pyramids and prisms in a 3-D mesh.

Depending on the problem and the solver capabilities, one or another option can be used.

For simple geometries a quadrilateral/hexahedron mesh gives high-quality solutions using fewer cells than a triangle/tetrahedron one, while for complex geometries is better to use a triangle/tetrahedron mesh, because it needs fewer meshing effort and a quadrilateral/hexahedron mesh shows no advantages in this case.

6.1. Meshing types

The meshes can be classified in structured, unstructured or hybrid.

1. Structured Meshes

A structured mesh has a regular connectivity, which restricts the elements to be hexahedral. In structured meshes, the positions of the nodes of a face can be stored in two-dimensional arrays, so that it demands less computational memory than an unstructured mesh.
2. **Unstructured Meshes**

An unstructured mesh is characterized by irregular connectivity. There is no relationship between the indices of one node and the indices of the one next to it. A separate list containing the connectivity information of the nodes has to be stored, that is why the storage requirements for this type of meshes are larger than in structured meshes. In spite of that, most commercial CFD software (like ANSYS CFD) uses only the unstructured storage type, also for structured meshes.

3. **Hybrid Meshes**

This kind of meshes contains structured portions and unstructured portions. It uses the most appropriate cell type in each region of the geometry.
6.2. Mesh quality

There are some aspects that need to be taken into account to get to a high mesh quality.

- Skewness:
  Skewness is defined as the ratio of the difference between the size of one cell and an optimal equilateral cell of equivalent volume and the optimal cell size.

\[
\frac{\text{optimal cell size} - \text{cell size}}{\text{optimal cell size}}
\]

The quality of the cell is better when the value of skewness is closer to zero, and worst as it approaches to one.

- Aspect ratio:
  The aspect ratio measures the stretching of the cell. It is defined as the ratio between the longest edge length and the shortest edge length of a cell. The ideal is to have an aspect ratio of one, but values until five are also accepted.

- Smoothness:
  The change in size from one element to the following one should be gradual. Sudden changes in cell size need to be avoid. Ideally, the maximum change in grid spacing should not be greater than the 20 %.

- Mesh density:
  The mesh density must be high enough to capture all the relevant flow features. If the mesh density is not high enough, some important aspects of the flow could be overlooked.

If the mesh fulfills all the mesh quality requirements a good and reliable solution can be obtained.

One of the most used methods when creating a mesh in most of the CFD software is the sweeping. In sweeping, two faces topologically on the opposite sides of the body are chosen. One of these will be set up as the source and the other one as the target. First of all, the source face will be meshed up with quadrilateral and/or triangular elements and then this mesh will be copied onto the target face. Finally hexahedral or prism elements will be created connecting the two faces and generating the mesh in the whole volume of the geometry.

Whenever it is possible, this method is preferred over the other ones. The sweeping method can be used in all the sweepable bodies. A body cannot be swept if there is more than one set of continuously connected faces in the body, if there is a completely contained internal void in the body or if no two faces can be identified as
source and target. Otherwise, the body is sweepable and choosing this method to create the mesh is the best option.

In near-wall regions, boundary layer effects must be taken into account. The velocity gradient varies significantly, and this must be studied in detail. There, it is important to have a more detailed mesh in order to obtain a result with all the relevant flow features. Computationally efficient meshes in these regions require that the elements have high aspect ratios. One option to do that is to create an inflated region near the wall. The inflation mesh region is a region where the mesh density is higher than the normal one and it is formed by a group of layers parallel to the boundary. (Lecheler, 2009).

Fig. 6.5: Example of inflation
7. Boundary conditions

The equations given in chapter 1 are the governing equations for all the flows, no matter the geometry or case to be studied. Although the governing equations are the same, the flow fields are different for every case. To obtain the exact solution in every situation, boundary conditions need to be applied. The boundary conditions are quite different for each case, and they dictate the particular solutions. (Ansys CFX Solver theory guide chapter 1.8 and Ansys CFX Solver modeling guide chapter 2).

The main boundaries in a fluid problem are:

1- Inlet
2- Outlet
3- Opening
4- Wall
5- Symmetry plane

Fig. 7.1: Types of boundary conditions
7.1. **Inlet**

In the inlet boundary the fluid predominantly flows into the domain. Depending on the Mach number we can choose between a subsonic flow and a supersonic flow. The Mach number is defined as the fluid flow velocity, $V$, divided by the speed of sound in the medium, $a$.

$$Ma = \frac{V}{a} \quad (7.1)$$

The subsonic flow is the one which has a Mach number smaller than the unity. On the other hand, the supersonic flow has a Mach number greater than one.

**Subsonic flow:**

In the inlet boundary condition the velocity or pressure must be specified. The different options that the program ANSYS offers are:

- Normal speed: the magnitude of the resultant normal velocity at the boundary is specified. The value specified is transferred from the fluid domain normal to each element face on that boundary.

- Cartesian velocity components: specify the Cartesian components of velocity on the inlet boundary.

- Cylindrical velocity components: specify the r, theta and z components of the velocity on the inlet boundary in cylindrical coordinates.

- Mass flow rate: the total mass flow rate into the domain at the boundary is specified.

- Total pressure: the relative total pressure and a flow direction are specified.

- Stationary frame total pressure: this is the same as the total pressure condition in a stationary domain.

- Static pressure: the relative static pressure and a flow direction are specified.

- Fluid velocity: this option is only available for an inhomogeneous multiphase simulation.

We must also define the level of incoming turbulence. If this level is known it can be specified with an appropriate length scale. The turbulence level, also known as turbulence intensity is defined as:
\[ I = \frac{u'}{U} \quad (7.2) \]

Where \( u' \) is the root-mean-square of the turbulent velocity fluctuations, and \( U \) is the mean velocity.

If the turbulent energy \( k \) is known, \( u' \) can be calculated as:

\[ u' = \sqrt{\frac{2}{3}} k \quad (7.3) \]

**Supersonic flow:**

This case can only be imposed if the total energy model is employed. The fluid should be an ideal gas, a real fluid, or a general fluid whose density is a function of pressure.

### 7.2. Outlet

In the outlet boundary the fluid predominantly flows out of the domain. The different options that the program Ansys offers to define the outlet are:

- Static pressure: The relative pressure is maintained at a fixed specified value over the outlet boundary.

- Normal speed: specify the magnitude of the flow velocity at the outlet.

- Cartesian velocity components: the boundary velocity components are specified.

- Cylindrical velocity components: the components and axis are specified in the same way as for an inlet.

- Average static pressure: with this option, the static pressure is allowed to locally vary on the outlet boundary such that the average pressure is constrained in a specified manner.

- Mass flow rate: the mass flow rate through the outlet boundary is specified.

- Degassing condition: this option is used to model a free surface from which dispersed bubbles are permitted to escape, but the liquid phase is not.

- Fluid velocity: this option is only available for an inhomogeneous multiphase simulation.
- Supercritical: Supercritical free surface flow means that the liquid velocity exceeds the local wave velocity, and nothing needs to be set at the outlet.

There are recommended configurations of boundary conditions due to the robustness of the problem:

<table>
<thead>
<tr>
<th></th>
<th>Inlet</th>
<th>Outlet</th>
</tr>
</thead>
<tbody>
<tr>
<td>Most robust</td>
<td>Velocity</td>
<td>Static pressure</td>
</tr>
<tr>
<td>Robust</td>
<td>Total pressure</td>
<td>Velocity</td>
</tr>
<tr>
<td>Sensitive to initial guess</td>
<td>Total pressure</td>
<td>Static pressure</td>
</tr>
<tr>
<td>Very unreliable</td>
<td>Static pressure</td>
<td>Static pressure</td>
</tr>
<tr>
<td>Not possible</td>
<td></td>
<td>Total pressure</td>
</tr>
</tbody>
</table>

7.3. Opening

When having an opening boundary condition the flow is into and/or out of the domain. The fluid flows in both directions across the boundary.

The different methods of defining an opening are:

- Cartesian velocity components
- Cylindrical velocity components
- Opening pressure and direction
- Static pressure and direction
- Entrainment: this option is useful for situations in which the main flow tends to pull fluid through the boundary where the flow direction is unknown
- Fluid velocity

7.4. Wall

A wall is an impenetrable boundary to fluid flow. Walls allow the permeation of heat and additional variables into and out of the domain through the setting of flux and fixed value conditions at wall boundaries.
The different types of defining a wall are:

- No slip wall: In the case of viscous fluids there is no relative velocity between the surface and the fluid immediately at the surface. The fluid next to the wall assumes the velocity of the wall.

- Free slip wall: For an inviscid fluid, the flow slips over the surface as there is no friction. The shear stress at the wall is zero, and the velocity of the fluid near the wall is not retarded by wall friction effects.

- Finite slip wall: The fluid slips at the wall when the wall shear stress is greater than a critical stress. (Typical use to simulate the flow of a non-Newtonian fluid)

- Counter-rotating wall: The wall boundary is assumed to be stationary.

- Rotating wall: Enables the wall to rotate with a specified angular velocity.

7.5. **Symmetry plane**

We use the symmetry plane to define a symmetric problem about a plane when the flow on one side of the plane is a mirror image of flow on the opposite side.
8. Validation: experiment on solar modules

8.1. Motivation

The flow mechanism around solar modules has become in past years a topic of greater importance, since the use of solar energy has been increasingly expanding so that nowadays many roofs of industrial buildings are covered with these elements.

The fundamental design parameter of these elements is the wind loading. More exactly the uplift load acting on them is the decisive safety criteria, since these elements are very often not connected to the roof, with self-weight acting as only stabilizing force.

Classical wind tunnels experiments fail to truthfully represent the flow mechanisms due to scaling reasons, therefore, CFD modeling appears to be an interesting alternative for understanding the flow behavior around this elements.

For validating the results of CFD modeling, 1:1 scale solar modules were tested at the wind tunnel facilities of the Institute.

A photo of the solar module can be seen in the figure below. (More detailed drawings of the solar module are in Annex A).

![Fig. 8.1: Solar module](image-url)
8.2. Experiment

8.2.1. Experiment set-up

At the exit of the wind tunnel a platform was designed to support the solar module and measure the forces at each of the four supports during the tests.

The force sensors were of the type K3D120 from the company ME-Messysteme. They were able to measure forces in each of the three spatial directions with a measuring range of ±500 N.

The velocity at the wind tunnel was measured at the reference height of 130 cm, with a hot-wire anemometer.

![Sensor K3D120](image1)

**Fig. 8.2: Sensor K3D120**

![Base of the solar module in the wind tunnel](image2)

**Fig. 8.3: Base of the solar module in the wind tunnel**
In the figure 8.3 the base where the solar module was set can be seen. The four points represent the places where the sensors were as well as the four supports of the solar module.

Additionally the wind profile (mean wind speed and turbulence) at the exit of the wind tunnel was measured and later given as a boundary condition in the inputs of the simulation (see chapter 8.3)

8.2.2. Experimental results

The forces at the supports were measured for different wind tunnel speeds between 0 and 24 m/s. The global horizontal reaction forces in flow direction are:

The vertical forces for each support:
Validation: experiment on solar modules

Fig. 8.6: Forces in the z direction for each support measured by the wind tunnel

This can be translated into a global system resulting force:

Fig. 8.7: Forces in the z direction measured by the wind tunnel
8.3. CFD modeling

To get to a CFD solution different steps have to be followed:

8.3.1. Creation of the geometry

First of all, the geometry has to be defined. In order to do that, a model of the solar module has been created using the Autodesk Inventor program, which will then be imported in the ANSYS workbench.

![Fig. 8.8: Solar module geometry](image)

To get to study the fluid flow around the solar module it is necessary to define a fluid flow volume. In this case, a rectangular prism around the object has been created defining the fluid flow domain.

8.3.2. Meshing

Once the geometry is done it is time to create an appropriate mesh. Different meshes have been created to compare the results obtained.

As explained in chapter 6 there are different parameters to define a good mesh. It is to be expected that for meshes with higher number of elements and nodes, better results will be obtained, but, in the same way, to obtain better result more computational time should be expected.

- Mesh option A

The first option is to let ANSYS program create an automatic mesh. Doing that we obtain a tetrahedral mesh with 295738 elements and 55521 nodes.
Fig. 8.9: Automatic mesh cut

- Mesh option B

Trying to obtain a better result, the volume has been cut into different pieces. In each piece the automatic method has generated a mesh. As a result, a mesh with smaller size elements has been created. In the pieces which do not contain part of the solar panel a structural hexahedral mesh has been generated as it is a basic geometry. On the other hand, in the other pieces which contain part of the object a tetrahedral and pyramidal unstructured mesh is generated. This mesh consists of 387098 nodes and 1502440 elements.

Fig. 8.10: Automatic mesh cut in sections with smaller element size

- Mesh option C

As it has been told on chapter 6 in the near-wall regions, boundary layer effects must be taken into account, so a region with smaller elements in the near-wall of the object is created. Doing that, ANSYS generates a more specified mesh, and then a high
number of elements and nodes are to be expected. We obtain a mesh with 4153655 elements and 744492 nodes.

![Automatic mesh with smaller element size in the near-wall regions](image1)

*Fig. 8.11: Automatic mesh with smaller element size in the near-wall regions*

![Detail of automatic mesh with smaller element size in the near-wall regions](image2)

*Fig. 8.12: Detail of automatic mesh with smaller element size in the near-wall regions*

- Mesh option D

Due to the high CPU memory and time this mesh demands, it could not be possible to solve the problem with a mesh with such a high number of elements. Then, the mesh relevance has been reduced to obtain a mesh with also a small element size in the near wall zones but with a minor number of elements and nodes. In this case the mesh generated has 2209575 elements and 416543 nodes. Now, it is possible to execute the program.
- Mesh option E

Trying to find the best option taking into account computational cost and accurate results, the fluid domain has been cut in different sections, where different mesh options can be applied. Due to the requirements of the sweep mesh option, it cannot be used in the entire volume. The volume is then divided in the parts where sweep can be used and the parts where an automatic method has to be chosen. Finally we obtain a mesh with 1457347 elements and 832381 nodes.

**8.3.3. Boundary conditions**

Now that the geometry has been created and a suitable mesh has been generated it is time to define the appropriate boundary conditions.
In the ANSYS program the fluid of study has been defined as air at 25°C, with a density of $\rho = 1,185 \text{ kg/m}^3$ and a viscosity of $\nu = 1,831 \cdot 10^{-5} \text{ kg/m/s}$.

The inlet is set at the entrance of the fluid domain and the outlet at the end of it.

![Fig. 8.15: Boundary conditions](image)

Trying to generate the same conditions as in the wind tunnel the velocity profile at the inlet has been defined as:

$$V(z) = 0,664 \cdot \nu_{ref} \cdot \log(z) + 1,347 \cdot \nu_{ref}$$  \hspace{1cm} (8.1)

![Fig. 8.16: Inlet velocity profile](image)

Where $\nu_{ref}$ is the velocity of reference which will adopt values from 2 m/s to 24 m/s to study different cases.
As there is no transition of flow through the base and the solar module they have been set as no slip wall. Finally, the other three walls that form the fluid domain have been defined as entrainments. There the fluid flows in both directions across the boundary.

Here it is also necessary to define the turbulence model that will be used as well as the intensity of turbulence.

Having studied the different turbulence models, the SST turbulence model has been chosen. Nowadays it is one of the most widely used methods because it combines the $k-\omega$ and the $k-\varepsilon$ model avoiding the problems of using only a $k-\varepsilon$ method or only a $k-\omega$ method.

The intensity of turbulence has been set to 16%, as it is the one measured in the wind tunnel experiments.

### 8.3.4. Results and comparison

First of all a comparison between the different meshes will be done. As it is well known, meshes with higher quality provide better results.

Ansys offers a variety of tools to check the quality of our mesh. It calculates a list of variables and then they can be compared to the optimal values.

- **Maximum face angle**
  This calculates the largest face angle for all faces that touch a node. For each face, the angle between the two edges of the face that touch the node is calculated and the largest angle from all face is returned for each node. Therefore, there is one maximum values for each node. It is considered to be a measure of skewness.

- **Minimum face angle**
  This calculates the smallest face angle for all faces that touch a node in the same way as it is calculated for the largest face angle.

- **Edge length ratio**
  This is a ratio of the longest edge of a face divided by the shortest edge of the face.

- **Connectivity number**
  It is the number of elements that touch a node.

- **Element volume ratio**
  It is defined as the ratio of the maximum volume of an element that touches a node, to the minimum volume of an element that touches a node. Accuracy decreases as the element volume ratio increases.
The following table gives some guidelines for checking the mesh quality.

![Guidelines for checking mesh quality](image)

**Fig. 8.17: Guidelines for checking mesh quality**

In the following table the mesh quality variables for the different meshes have been calculated. And so it is easy to compare them in terms of quality. It is also important to remark the differences of time demands of each mesh. The computational time in the table shows the time required to solve each velocity case.

<table>
<thead>
<tr>
<th>Mesh Option</th>
<th>Elements</th>
<th>Nodes</th>
<th>Maximum face angle</th>
<th>Minimum face angle</th>
<th>Edge length ratio</th>
<th>Connectivity number</th>
<th>Element volume ratio</th>
<th>Computational time</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>295738</td>
<td>55521</td>
<td>167,655</td>
<td>0.468972</td>
<td>122,217</td>
<td>(3-58)</td>
<td>3533,52</td>
<td>30min</td>
</tr>
<tr>
<td>B</td>
<td>1502440</td>
<td>387098</td>
<td>177,066</td>
<td>0.468972</td>
<td>122,217</td>
<td>(1-82)</td>
<td>1544,37</td>
<td>2h</td>
</tr>
<tr>
<td>C</td>
<td>4153655</td>
<td>744492</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>(3-44)</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>D</td>
<td>2209575</td>
<td>416543</td>
<td>151,261</td>
<td>0.839534</td>
<td>67,9816</td>
<td>(1-78)</td>
<td>437,941</td>
<td>2h 15min</td>
</tr>
<tr>
<td>E</td>
<td>1457347</td>
<td>832381</td>
<td>176,25</td>
<td>0.746</td>
<td>82,72</td>
<td></td>
<td>3995,15</td>
<td>4h 20 min</td>
</tr>
</tbody>
</table>

**Fig. 8.18: Mesh quality parameters**

The mesh option B and D are the ones chosen to continue with the calculations since they are the ones that offer a higher mesh quality with an affordable computational time. Nevertheless, calculations with some specific values of the velocity have been run.
up with the other types of meshing just to compare the accuracy of their results with the other ones.

Once ANSYS finish with the calculations it is time to observe the results.

Some pictures of the fluid flow around the solar module have been taken, as well as a plot of the pressure over the solar module.

The results can be seen in the following pictures. They show the streamlines over the solar module (Figure 8.19 and 8.20) as well as the velocity of the fluid around it (Figure 8.21 and 8.22) and the pressure over it (Figure 8.23).

*Fig. 8.19: Streamlines around the solar module for $v_{ref} = 4m/s$*
Fig. 8.20: Streamlines around the solar module for $v_{\text{ref}} = 4 \text{m/s}$

Fig. 8.21: Velocity of the fluid around the solar module for $v_{\text{ref}} = 4 \text{m/s}$
Fig. 8.22: Velocity of the fluid around the solar module for $v_{ref} = 4m/s$

Fig. 8.23: Pressure over the solar module for $v_{ref} = 4m/s$
Now is time to compare the results obtained for the forces. The forces studied are the ones acting to the solar module in the x (horizontal) and the z (vertical) direction.

The graphics shown below (figures 8.17 and 8.18) represent the forces calculated in each case for different velocities:

**Fig. 8.24: Forces in the x direction**

**Fig. 8.25: Forces in the z direction**

Finally the figures below (Fig. 8.22 and Fig 8.23) contain the graphics of all the different types of meshing with the wind tunnel results, so it is easy to compare them.
We can see that in the case of the forces measured in the x direction, all the models created with different types of meshing generate similar results, even though they differ a little bit from the experimental results in the wind tunnel.

In the other hand, taking a look to the forces measured in the z direction with the different models created it is easy to see, that, as predicted, the mesh options B and D are the ones that approximate the most to the real values obtained with the wind tunnel. Nevertheless, the other mesh options do also show a high accuracy in their results, as they do not differ so much from the experimental values.
Conclusions

The main goals of the project were achieved within the expected time line.

In the first stage of the project, an extensive investigation of the key aspects of the simulation of fluid dynamics was performed.

Goal of this part of the project was to create a simple and comprehensive knowledge database, which would allow the inexperienced user to understand some of the backgrounds of the simulation as well as give him some advice for the proper modeling of his problem. An special focus was given on the turbulence treatment as it is still one of the main difficulties that CFD calculations must deal with.

In the second part of the project, a commercial software package, Ansys CFX, was used to simulate the flow around solar modules. The Institute of Steel Construction of the RWTH Aachen had performed extensive wind tunnel tests on these elements, allowing for a validation of results of the CFD calculation.

All aspects investigated during the first part of the project could successfully be applied in a careful modeling of the geometry and flow conditions around these elements.

The results show good agreement to the studied values, the force in the x direction and in the z direction.

As a parameter, different mesh types were investigated. Meshes with different grade of detail were generated and finally conclude that, as expected, meshes with a more detailed element size give a better result. Nevertheless, all the different types of meshes have given reliable results.

It could be proven that the simulation could reliably predict the uplift forces (which are the most important parameter for this type of elements) within a very reasonable amount of time compared to a classical wind tunnel investigation.

CFD calculations show an increasing potential for application in many fields of wind engineering as it has been shown in this study. Nevertheless a careful study of the backgrounds of fluid mechanics as well as CFD modeling is strongly advised in order to make the right choices regarding turbulence, meshing and solver type for a given problem.
Bibliography


[27] www.kxcad.net/ansys

[28] www.cfd-online.com

[29] Ansys tutorials

Annex A

This annex contains detailed drawings of the solar module studied.
Solar module:
Solar panel: