

Turbulence, Vibrations, Noise and Fluid Instabilities. Practical Approach.

Dr. Carlos Gavilán Moreno
Cofrentes N.P.P. Iberdrola S.A.
Spain

1. Introduction

Colloquially speaking, turbulence in any language means disorderly, incomprehensible, and of course, unpredictable movement. Consequently, we encounter expressions that employ the word turbulence in social and economic contexts; in aviation whenever there are abnormalities in the air, and even in psychology and the behavioural sciences in reference to turbulent conduct, or a turbulent life, in the sense of a dissolute existence. Thus has the word turbulence become associated with chaos, unpredictability, high energy, uncontrollable movement: dissipation. All of the foregoing concepts have their source in the world of hydrodynamics, or fluid mechanics.

In fluid mechanics, turbulence refers to disturbance in a flow, which under other circumstances would be ordered, and as such would be laminar. These disturbances exert an effect on the flow itself, as well as on the elements it contains, or which are submerged in it. The process that is taking place in the flow in question is also affected. As a result, they possess beneficial properties in some fields, and harmful ones in others. For example, turbulence improves processes in which mixing, heat exchange, etc., are involved. However, it demands greater energy from pumps and fans, reduces turbine efficiency and makes noise in valves and gives rise to vibrations and instabilities in pipes, and other elements.

The study of turbulence and its related effects is a mental process; one that begins with great frustration and goes on to destroy heretofore accepted theories and assumptions, finally ending up in irremediable chaos. "I am an old man now, and when I die and go to heaven, there are two matters on which I hope for enlightenment. One is the quantum electrodynamics, and the other is the turbulent motion of fluids. An about the former I am rather optimistic" (Attributed to Horace Lamb).

Nonetheless, some progress has been made in turbulence knowledge, modelling and prediction. (Kolmogorov, 1941). This chapter will deal briefly with these advances, as well as with the effects of turbulence on practical applications. In this sense, reference will be made to noise effects and modelling, as well as to flow vibration and instabilities provoked by turbulence. (Gavilán 2008, Gavilán 2009).

2. Turbulence.

Of itself, turbulence is a concept that points to unpredictability and chaos. For our purposes, we will deal with this concept as it applies to fluid mechanics. Therefore, we will be dealing

Source: Computational Fluid Dynamics, Book edited by: Hyoung Woo OH,
ISBN 978-953-7619-59-6, pp. 420, January 2010, INTECH, Croatia, downloaded from SCIYO.COM

with turbulent flow. Throughout what follows, the terms *turbulence* and *turbulent flow* will be understood as synonymous. Some texts treat the terms *turbulence* and *vortex* as analogous, however, this seems to be rather simplistic. For the purposes of this work, it seems best to take the concept of turbulence in its broadest sense possible.

Historically, fluid mechanics has been treated in two different ways, namely, in accordance with the Euler approach, or pursuant to the Lagrange approach. The Eulerian method is static, given that upon fixing a point, fluid variations are determined on the effect they have on this point at any given time. On the other hand, the Lagrangian method is dynamic, given that it follows the fluid. In this way, variations in the properties of the fluid in question are observed and/or calculated by following a particle at every single moment over a period of time.

The Eulerian method is the one most employed, above all, in recent times, by means of numerical methods, such as that of finite elements, infinites, finite volumes, etc. Notwithstanding, there is great interest in the Lagrangian method or approach, given that it is one that is compatible with methods that do not use mesh or points. (Oñate, E; et al. 1996) Throughout history, there have been two currents of thought as regards the treatment of turbulence. One is the so-called deterministic approach, which consists of solving the Navier-Stokes equation, with the relevant simplifications, (Euler, Bernoulli) practically exclusively via the use of numbers. The other approach is statistical. The work of Kolmogorov stands out in this field; work which will be dealt with below, given its later influence on numerical methods and the results of same. Apart from the Eulerian or Lagrangian methods, classic turbulent fluid theory will be dealt with in Section 2.1, whereas the statistical or stochastic approach will be dealt with in Section 2.2, in clear reference to Kolmogorov's theory. (Kolmogorov, 1941).

2.1 General theory

This section provides a brief and concise exposition of successive fluid flow approaches designed to respond to the presence of anomalies that were later referred to as *turbulence*, and which gave rise to the concept of turbulent fluid. Furthermore, the equations given enable the visualisation of the turbulence in question and its later development. A Eulerian and deterministic focus will be followed in this section.

Working in reverse to the historical approach, the fluid flow equation formulated by Navier-Stokes in 1820 is given; firstly, because it is the most general one, and secondly, because, of itself, its solution can represent the turbulent flow equation.

$$\frac{\partial u_i}{\partial t} + \sum_{j=1}^n u_j \frac{\partial u_i}{\partial x_j} = \nu \Delta u_i - \frac{\partial p}{\partial x_i} + f_i(x, t) \quad (1)$$

$$\text{div} \vec{u} = \sum_{i=1}^n \frac{\partial u_i}{\partial x_i} = 0 \quad (2)$$

$$u(x, 0) = u^0(x) \quad (3)$$

Where u_i stands for the velocity components at each point and at each moment in time, ν is the viscosity, p is the pressure at each point and at each moment in time, and f_i refers to the external forces at each point and at each moment in time. By annulling the viscosity and its effects, we get Euler's fluid flow equation announced in 1750.

$$\frac{\partial u_i}{\partial t} + \sum_{j=1}^n u_j \frac{\partial u_i}{\partial x_j} = -\frac{\partial p}{\partial x_i} + f_i(x, t) \quad (4)$$

$$\text{div} \vec{u} = \sum_{i=1}^n \frac{\partial u_i}{\partial x_i} = 0 \quad (5)$$

$$u(x, 0) = u^0(x) \quad (6)$$

In addition to this simplification, if we make the fluid stationary, so that:

$$\frac{\partial u}{\partial t} = 0 \quad (7)$$

we get the first Bernoulli theorem announced in 1738.

$$\frac{D}{Dt} \left(p + \rho \cdot \phi + \rho \frac{u^2}{2} \right) = 0 \quad (8)$$

The solving of these equations, given their simplifications and context conditions, provide us with a field of speeds and pressures for a fluid in movement. Indeed, as regards the Navier-Stokes equation, it can be solved and turbulences and instabilities determined by employing powerful numerical solution methods, such as that of the finite element.

Thus, the equations that govern fluid movement. By means of these two equations, particularly the last two (those of Euler and Bernoulli), it was observed that, under certain conditions, the results did not correspond to reality, on account of a certain problem of disorder developing in the fluid and its flow. Only the accurate and numerical solving of the Navier-Stokes equation can exactly reproduce these phenomena. To this end, resort must be had to potent computational fluid dynamic (CFD) software. Notwithstanding, in 1883, Osborne Reynolds discovered a parameter that predicted or anticipated the chaotic and turbulent of the fluid: the Reynolds number.

$$Re = \frac{\rho \cdot v_s \cdot D}{\mu} = \frac{v_s \cdot D}{\nu} \quad (9)$$

Thus was it established that the flow is stationary, and therefore laminar, for $Re < 2000$ values, a fact which meant that the solutions given by Bernoulli and Euler were very accurate. For values of $2000 < Re < 4000$ the system was deemed to be in transition, and therefore, not stationary. The functions that work best are those of Euler and Navier-Stokes. This turbulence undergoes three phases or states of development.

- A. Growth of coherent bidimensional vortices¹.
- B. Joining of Vortices
- C. Separation of vortices and turbulent state in 3D

Finally, if completely turbulent, the fluid is non-stationary and tri-dimensional for values of $Re > 4000$. These solutions are only possible by means of using the Navier-Stokes equations.

In conclusion, only the numerical solution of the Navier-Stokes equation provides a solution that considers turbulence. Nevertheless, there are other processes and approaches that will be developed in Section 2.3. Turbulence is, therefore, produced by the interaction of the fluid with geometry, by the loss of energy due to viscosity, by density variations caused by temperature, or other factors, such as changes in speed, or all of these at once. Consequently,

¹ These are referred to as coherent because the vorticity is concentrated and the fluid flows around as if it were a solid obstacle. It keeps its shape for longer than a single rotation.

the turbulent flow is unpredictable and chaotic in the sense that it depends on a host of small variations in the initial conditions and these disturbances are amplified in such a way that it becomes possible to predict them in space and time. Another of its features is its great capacity for mixing and, lastly, that it affects at various scales and wavelengths. It could be said that together, fluid, structure and context conditions, constitute a non-linear, non-stationary dynamic system. Its most noteworthy characteristic is its sensitivity to the initial conditions and its self-similarity, which will serve as a starting point to develop Kolmogorov's theory.

In the 19th and 20th centuries, several researchers devoted great efforts to studying turbulence under certain, extremely particular conditions. Examples of such turbulence or instabilities are to those of Von Karman, Kevin-Helmoltz, Raleygh-Bernard, and so on.

2.2 Kolmogorov's theory

Kolmogorov's theory cannot be dealt with without first mentioning the spectral analysis of turbulence, or the application of Fourier's analysis to the study of turbulence. Fourier's theory decomposes the fluctuations into sinusoidal components and studies the distribution of the turbulent energy along several wavelengths. In this way it becomes possible to get several scales of turbulence and their evolution in time. This technique works and produces acceptable results when turbulence is homogenous. Under this condition, accurate equations can be determined for the speed spectrum and for the transferring of energy between difference scales of turbulence, as well as the dissipation of turbulent energy due to viscosity. The simplest development assumes that there is no average speed gradient, in such a way that the turbulence interacts with itself, with the energy dropping by itself. Neither energy sources nor sinks are taken into account.

To start, we assume that the speed of a particle in the fluid can be decomposed into an average speed plus a fluctuation component.

$$U_i = \bar{U}_i + u_i \quad (10)$$

R_{ij} is defined as the speed correlation function by the expression:

$$R_{ij}(x, x', t) = \overline{u(x, t)_i u(x', t)_j} \quad (11)$$

Given that we assume homogenous turbulence, R_{ij} is only a function of the distance r , which is the distance between x and x' :

$$R_{ij}(r, t) = \overline{u(x, t)_i u(x', t)_j} \quad (12)$$

Therefore, we assume that the value of the correlation function tends to zero when the radius tends to the infinite. We now define a spectral function Φ_{ij} as the Fourier transform of the correlation function in 3D:

$$\Phi_{ij}(k, t) = \frac{1}{(2\pi)^3} \int R_{ij}(r, t) \cdot e^{-ikr} \cdot d^3k \quad (13)$$

This spectral function, which will form the spectral matrix, depends on time and on the wave number k , which is a vector. The turbulent kinetic energy can be expressed as:

$$\frac{1}{2} \overline{u_i u_i} = \frac{1}{2} R_{ij}(0, t) = \frac{1}{2} \int \Phi_{ii}(k, t) d^3k \quad (14)$$

The magnitude of the vector of the wave number $k=[k]$ is the inverse of the length, thus the length is k^{-1} . Therefore, if L is a characteristic scale of turbulence, the L^{-1} value of the wave number represents the higher scales of turbulence, where all the turbulent kinetic energy is contained. The spectral function, Φ_{ij} , describes the energy distribution in relation to wave numbers. It has a maximum value of $[k]=L^{-1}$, which drops to zero when $[k]$ tends to the infinite. The evolution of kinetic energy is defined as:

$$E = 2\pi k^2 \Phi_{ii} \tag{15}$$

As far as the evolution of the k energy value is concerned, this can be seen in the following figure (Figure 1).

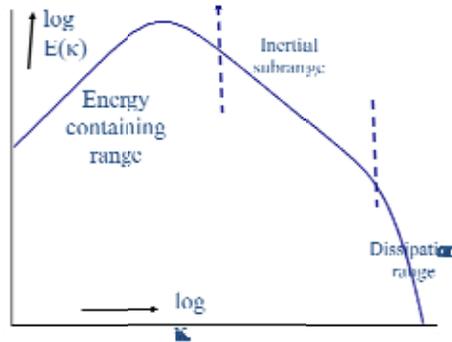


Fig. 1. Energy ranges.

This is where the term, or value, η comes in, referring to Kolmogorov’s scale, which determines the scale point at which energy dissipation by viscosity is most significant. We will use the following equation for Kolmogorov’s theory:

$$\frac{\partial E}{\partial t} = T - 2 \cdot \nu \cdot k^2 E \tag{16}$$

That is to say, the energy and its evolution in time will be a function of that which is transferred and that which is dissipated. Integrating it from $k=0$ to a greater, or finite, as well as limited, value gives us:

$$\frac{d}{dt} \int_0^k E(k', t) dk' = -\mathfrak{N}(k, t) - 2\nu \int_0^k k'^2 E(k', t) dk' \tag{17}$$

Where:

$$\mathfrak{N}(k, t) = - \int_0^k T(k', t) dk' = \int_k^\infty T(k', t) dk' \tag{18}$$

This can be interpreted as follows:

“Energy changes in wave numbers below k on account of the interchange with larger wavelengths through the energy spectral fluid and the dissipation caused by viscosity forces on its own wavelength”

Kolmogorov holds that turbulence properties in the inertial range, above η , only depend on the energy spectral fluid, in such a way that:

$$\kappa = \bar{\epsilon} \tag{19}$$

The idea behind this is that the transfer cascade evolves towards lower scales, leaving the spectral fluid as a fixed parameter. Consequently, the energy equation now stands at:

$$E(k, t) = C \cdot \epsilon^{-\frac{2}{3}} \cdot k^{-\frac{5}{3}} \tag{20}$$

where C is a Kolmogorov constant, considered as a universal, the value of which is 1.5 when $L^{-1} \ll k \ll \eta^{-1}$ and the Re (Reynolds number) is high. Thus, beyond the inertial scale wave numbers, we have the dissipation range numbers, as shown in Figure 2.

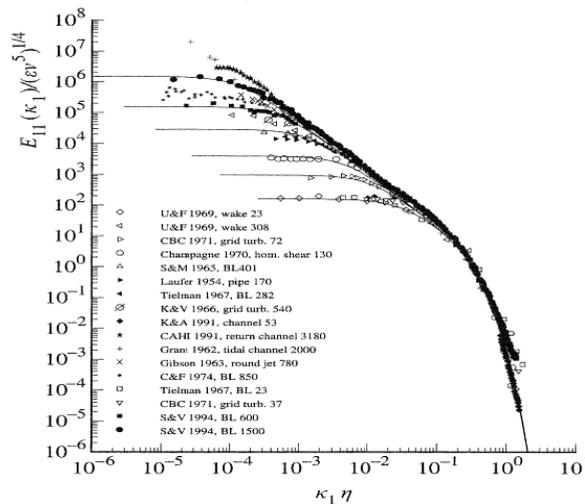


Fig. 2. Kolmogorov scales and data from several tests.

This range is only determined by the η parameter, which only has a dependence parameter, as shown by the equation (21).

$$\eta = \left(\frac{\nu}{\epsilon}\right)^{\frac{1}{4}} \tag{21}$$

This Kolmogorov derived value gives the size of the smallest turbulence dissipation scale. The dissipation range energy expression now becomes:

$$E(k, t) = \epsilon^{\frac{1}{4}} \nu^{\frac{5}{4}} \cdot F(k\eta) \tag{22}$$

where $F(\cdot)$ is also a universal function.

We will define the Kolmogorov time and speed scale factor with the ϵ y ν parameters, which determine the behaviour of the flow in the dissipation range:

$$t_\eta = \left(\frac{\nu}{\epsilon}\right)^{\frac{1}{2}} \tag{23}$$

$$u_\eta = (\nu \cdot \epsilon)^{\frac{1}{4}} \tag{24}$$

Thus, the Kolmogorov theory completes the spectral analysis. The theory postulates high Reynolds number values. The small turbulence scales are assumed that serve to balance and be controlled by the average energy flow, which is generated in the inertial scale and which equals the dissipation rate. Furthermore, Kolmogorov's theory universally predicts speed properties and their differences for small separations, as well as their correlations and spectre, only depending on the ν y ϵ parameters. Kolmogorov also marked the boundary between the transferred or contained energy range (inertial range) and the dissipative structures by way of the following expression:

$$\frac{L}{\eta} = f(Re^{3/4}) \quad (25)$$

Therefore, given a Reynolds number, the lower scales are not sensitive to the turbulent flow in which they find themselves. Nevertheless, the lower scales are intimately related to the flow, with their properties varying substantially depending on the specific flow. These concepts will form the basis for future methods of modelling and solving fluid movement problems, such as k - ϵ and Large Eddy Simulation (LES) models.

2.3 Simulation of turbulent fluids.

As is well known, Navier, L.H.H. and Stokes, G.G derived fluid movement equations over 150 years ago. This equation, along with that of continuity, provides an answer to any fluid movement problem.

The solution and determination of the speed field is, therefore, nothing more than discretising the domain and the differential equations and applying context conditions in order to repeatedly solve the system formed until achieving convergence. This is the so-called Direct Numerical Solution (DNS). Nonetheless, this simple method is only useful in simple geometries, given that otherwise, the time calculation would be so big as to make any simulation unfeasible. This defect arises noticeably when we design fluid-structure (FIS) models. Consequently, other models need to be found, which are less costly computationally speaking.

The most common solution to the high number of elements required by the former simulation method, DNS, is to use weighted, or weighting, techniques. In this way, modelling is done on a small scale expecting that the solution will respond to the flow as a whole. This is the idea that underlies the so-called Reynolds Averaged Navier-Stokes (RANS). It must be assumed that the speed of a turbulent flow can be described as follows:

$$\vec{u} = \bar{\vec{u}} + \vec{u}' \quad (26)$$

That is to say, as the sum of an average speed plus a fluctuating component on this average speed. The following is an example of the application of this assumption. It refers to the 2D modelling of a turbulent flow on a flat plate, the Navier Stokes (NS) equation for which is given below:

$$\rho \left(\frac{\partial u_x}{\partial t} + u_x \frac{\partial u_x}{\partial x} + u_x \frac{\partial u_x}{\partial y} \right) = \mu \frac{\partial^2 u_x}{\partial y^2} \quad (27)$$

replacing

$$\rho \left(\overline{u_x \frac{\partial u_x}{\partial x}} + \overline{u_x \frac{\partial u_x}{\partial y}} \right) = \mu \frac{\partial^2 \overline{u_x}}{\partial y^2} - \rho \frac{\partial}{\partial y} \overline{u'_x u'_y} \quad (28)$$

The difficulty of this expression and of all those assuming equation (28) is that an analytical expression of the term is required:

$$\rho \frac{\partial}{\partial y} \overline{u'_x u'_y} \quad (29)$$

known as Reynold Stress. In order to avoid this difficulty, the kinetic energy (k) and the dissipation rate (ϵ) are modelled. This gives rise to a well known model: the k - ϵ model. The simplification consists of the following:

$$\rho \cdot \overline{u'_x u'_y} = \mu_{turb} \frac{\partial \overline{u_x}}{\partial y} \quad (30)$$

where, moreover,

$$\mu_{turb} = \frac{C_k \cdot k^2}{\epsilon} \quad (31)$$

by replacing all of this, in the proposed movement equation, the result is as follows:

$$\left(\overline{u_x} \frac{\partial \overline{u_x}}{\partial x} + \overline{u_y} \frac{\partial \overline{u_x}}{\partial y} \right) = \nu \frac{\partial^2 \overline{u_x}}{\partial y^2} - \frac{C_k \cdot k^2}{\epsilon} \frac{\partial \overline{u_x}}{\partial y} \quad (32)$$

This k - ϵ model has a big advantage over the DNS model in that it takes less time to compute. This model provides acceptable solutions when flow fluctuations are not very important. When the flow motor, or the problem, to be studied has to do with pressure fluctuation, that is to say, external flow Flow Induced Vibrations (FIV), the RANS model is unviable, given that it does not provide quality solutions. There are several RANS model variations, the main features of which are shown in Table 1.

Method	Strengths	Weaknesses
Spalart-Allmaras	A one-equation model, which provides less computational effort than most other models. Produced for external flow over airfoils but is increasing in popularity for turbo machinery applications. Performs well for attached wall-bounded flows with weakly complex boundary layers.	Weak for adverse pressure gradients that produce boundary layer separation. Since it is relatively new, it has a lack of submodels available.
RNG k- ϵ	Possesses many of the same characteristics as the standard k- ϵ , but uses mathematical group theory to determine the previously empirical constants. It performs better for moderately complex flows like jet impingement.	Subject to limitations due to isotropic eddy viscosity assumptions.
Reynolds Stress Model (RSM)	Highly rooted in the physics by solving a transport equation for each Reynolds stress.	Requires much more computational effort than any other technique

Table 1. RANS submodels, strengths and weaknesses.

Lastly, there is a model that lies halfway between the RANS and DNS. The DNS model has been seen to represent the real circumstance and disturbances very well. On the other hand, though the RANS model is extremely comfortable computationally speaking, it does not represent turbulence or disturbances very well. The so-called Large Eddy Simulation (LES) model provides instantaneous solutions, as does the DNS model, while at the same time containing models and simplifications such as those in the RANS model. The LES model solves the Navier-Stokes equation as does the DNS model, but the equations are spatially filtered, or refined. This filtering or refining of the equations means that the flow is determined at a characteristic scale, and is modelled afterwards at lower scales. It is the application of Kolmogorov's theory to non-stochastic numerical models.

This succession of modelling and refining is due to the fact that big eddies behave anisotropically and, therefore, must be calculated, whereas smaller eddies behave isotropically and can, as such, be treated statistically. Thus, the mesh is such that the majority of energy is contained in big eddies and calculated directly, the rest is assigned in a weighted manner to flow as a whole.

The bigger the Reynolds number, the bigger the cost in this LES model. Mathematically, the model is based on the Φ flow being formed by two superimposed flows:

$$\phi = \bar{\phi} + \phi' \quad (33)$$

the mesh flow, or the grand scale flow, $\bar{\phi}$

the sub-mesh flow, or the small scale flow ϕ'

The grand scale (GS) flow is calculated as follows:

$$\overline{\phi(x)} = \int \phi(x') G(x, x') dx' \quad (34)$$

where

$$G(x, x') = \begin{cases} \frac{1}{v} \sqrt{x \epsilon \Delta v} \\ 0 \text{ resto} \end{cases} \quad (35)$$

then replacing

$$\overline{\phi(x)} = \frac{1}{v} \int \phi(x') dx' \quad (36)$$

By applying these assumptions to the flow movement equation

$$\frac{\partial \bar{u}_i}{\partial t} + \frac{\partial}{\partial x} (\bar{u}_i \bar{u}_j) = -\frac{1}{\rho} \frac{\partial}{\partial x} \bar{p} + \nu \frac{\partial^2 \bar{u}_i}{\partial x_j^2} + \tau_{ij} \quad (37)$$

$$\tau_{ij} = \overline{u_i u_j} - \bar{u}_i \bar{u}_j \quad (38)$$

the latter term is modelled on the sub-mesh, or on the SGS sub-model and the accuracy of the model rests on the idea that lower speeds than the mesh are homogenous and, therefore, can be modelled with great accuracy. The aim of the LES method is to solve the majority of the flow and to model only a small part of it. Thus, the LES method strikes a balance between mesh size and accuracy. Figure 3, in which the solution for a turbulent flow in a pipe compares the RANS method with the LES one, provides an explanatory example.

These are practically the general models that are used in flow simulation and fluid movement. Notwithstanding, there are variations of these basic methods depending on the application in question.

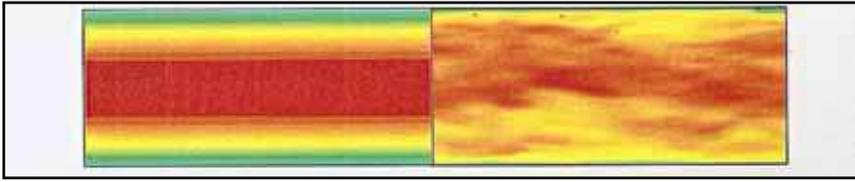


Fig. 3. Solutions of the same problem with a RANS model (on the left) and LES model (on the right).

Quite clearly, discretisations can be Eulerian, or Lagrangian, in such a way as to respond to both forms of tackling problems concerning fluid dynamics.

3. General effects of the turbulence. (FIC)

Having introduced turbulence, the turbulent flow and its equations and simulations, the question arises as to what can turbulence do? What are its possible effects? To answer these questions, first off it is necessary to decide whether the effects in question are general or local, that is to say, whether they are provoked by the turbulent flow or, on the contrary, are caused by turbulence or instabilities. The latter will be dealt with in Sections 4, 5 and 6, while the general effects are considered below.

Numerous studies have been carried out highlighting the benefits of turbulence with respect to miscibility, diffusion and heat exchange, though it can also have harmful effects. Here we are going to deal with the phenomenon of spontaneous cavitation, and its effects, in turbulent flow. Specifically, our aim is to study the effect of the change of speed caused by vorticity and pressure due to load loss on account of turbulence in high energy turbulent fluids near saturation point, such as, for example, feed water in power stations.

Firstly, we are going to define a non-dimensional parameter, which is called the cavitation number.

$$Ca = \frac{Pa - Pv}{\frac{1}{2}\rho v^2} \quad (39)$$

where:

P_a is the local or system pressure;

P_v is the vaporization pressure at system temperature;

ρ is the fluid density at system temperature;

V is the fluid speed.

Cavitation will occur when the parameter value is low. There is a limit value that corresponds to a determined speed, for each temperature and pressure, below which cavitation is ensured.

Speed can be defined in a turbulent flow as follows:

$$u_T(t) = u_m + \delta u(t) \quad (40)$$

That is to say, an average value and a fluctuation or oscillation component. Moreover, turbulence intensity is defined as:

$$u'^2(t) = \frac{1}{T} \int_0^T u_T(t)^2 dt \quad (41)$$

replacing the equation 40 in the equation 39 gives:

$$Ca = \frac{Pa - Pv}{\frac{1}{2}\rho(u_m^2(t) + 2 \cdot u_m(t) \cdot \delta u(t) + \delta u^2(t))} \quad (42)$$

If we assume that $\delta u(t)$ is proportional to the standard deviation of the measured values in the speed system, and that in turn, this standard deviation is a function of the average speed, (Gavilán, C.J. 2008) we find that:

- $\delta u(t)$ is zero for low speeds, therefore, the Y expression is accurate and can be applied to all fluid points. Moreover, the Bernoulli and Euler theorem is applicable, therefore, there will be no spontaneous cavitation, unless the average speed changes.
- For high speeds, $\delta u(t) \gg 0$, there may be some speed values at which Ca is lower than the average value, and therefore, spontaneous cavitation may occur.

This effect will be called Fluid Induced Cavitation (FIC), given that, although cavitation is associated with turbulence, in turbo-machines or rotary pumps, little attention is paid to spontaneous cavitation due to the effect of turbulence on fluid systems. This effect is not widely known, though it is most definitely of great importance when dealing with fluids that are working very near the vaporization limit, or to put it more clearly, when the system pressure is very near the fluid vapour pressure at working temperature. This is particularly important for thermal power plants, or energy producing stations. In such facilities, the liquid is heated before entering the vaporization element; if there are elements that make the pressure fall, such as elbows, T's, filters, etc. in the conditions prior to vaporization, the numerator of equation 39 drops, reducing the Ca value, thus representing a high risk situation. In the same way, if we have elements in which the speed rises, such as jet pumps, venturi tubes, and other restrictions, the denominator increases, in such a way that the Ca value falls, thus increasing the risk of cavitation.

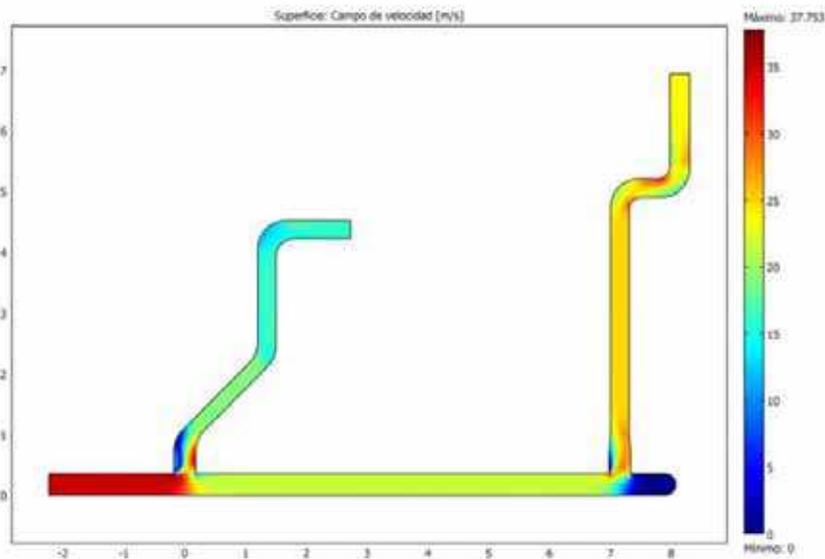


Fig. 4. Flow in pipe system.

Cavitation, which elsewhere and in other fora is referred to as *flashing*, is a vibration and noise - FIV and FIN - and may even manage to change the flow pattern, and therefore, cause FII flow instabilities. Other effects also need to be taken into account, such as the presence of erosion-corrosion, in what is, in principle, a single-phase fluid. Furthermore, if there are stable cavitation conditions, and a real and permanent void ratio is established, pump cavitation is ensured, even when this should not occur according to the calculations. There are other reasons for possible cavitation, such as variations in the density and vapour pressure parameters on account of changes in the pressure and temperature conditions. Let us have a look at a real example. Figure 4 shows the case of feed water speed in a pipe to an electricity-generating installation boiler. If we calculate the Ca parameter between two points, A and B, already having the average, or tendency, speed and the real one, we see that their Ca values are different. After the Ca , v and p values have been calculated in several situations, if we extrapolate them, there is such a turbulent speed value that it provokes spontaneous cavitation (Figure 5). Therefore, there is a speed limit for each flow at which spontaneous cavitation occurs, which coincides with the asymptotic value.

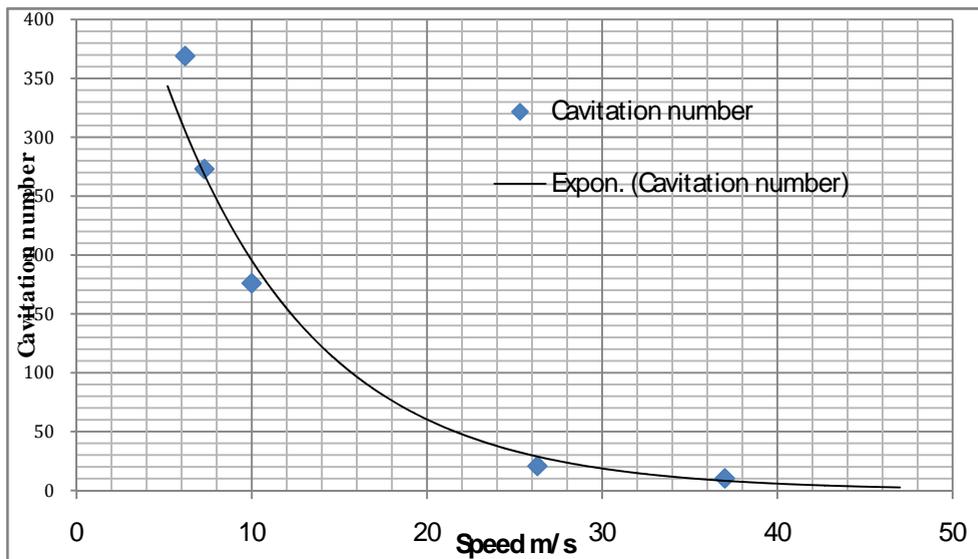


Fig. 5. Cavitation number versus flow speed.

The cavitation effect is not completely harmful, given that occasionally it is provoked by means of the speed of the fluid around a vehicle in such a way that movement resistance is very low and energy is saved in the movement, the fluid entry length is increased, or the energy is increased on knocking against another object in the fluid. This is the case of the supercavitating torpedoes used by the Russian navy.

4. Fluid Induced Vibration. (FIV)

There are lots of books and articles that have dealt with this subject. Strictly speaking, the term was only coined in the 1980's. In truth, this case study is a particular case of fluid-

structure interaction. Interest in this subject lies in the fact that the source of vibration is dissipated energy caused by turbulence, or in other cases, by eddies that produce oscillating lift forces that impregnate objects immersed in the fluid with a vibratory movement.

There are two basic FIV mechanisms:

- a self-induced vibrating mechanism
- a forced vibration mechanism.

Mathematically speaking, this is true. Nevertheless, the subject is somewhat wider than this might suggest, as can be seen from the following classification, which provides us with a more complete view. (Figure 6).

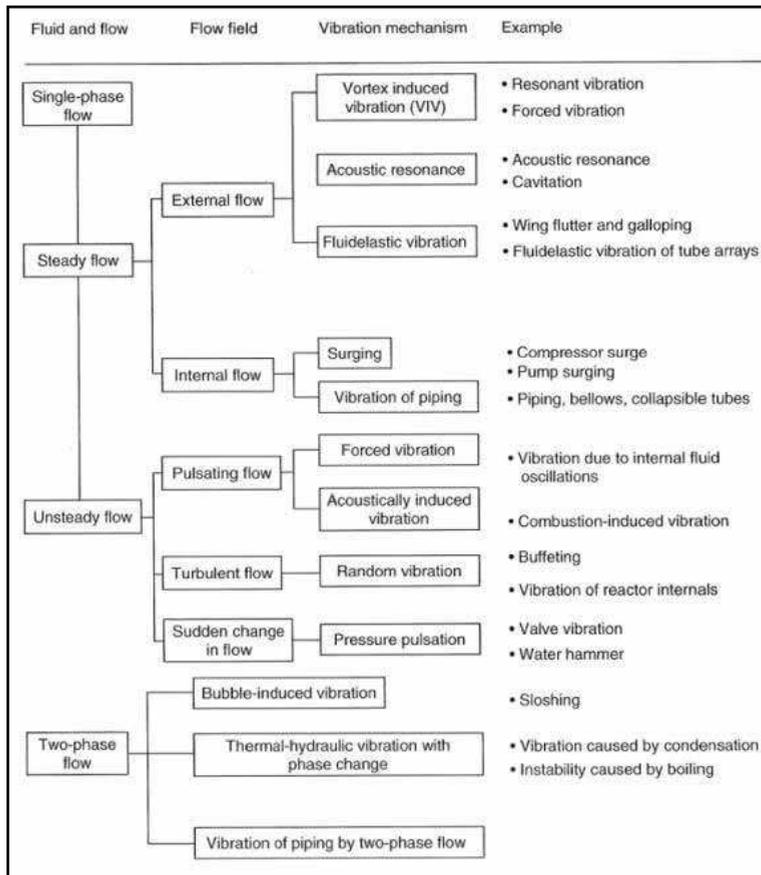


Fig. 6. Vibration mechanism.

As far as we are concerned, two of the most important effects, or examples, as regards nuclear plants of the Boiling Water Reactor (BWR) type, are those that reflect the influence of FIV on reactor internals. That is to say, on single-phase fluids in a non-stationary operating system with turbulent flow, and in the case of a bi-phase flow, those that are subject to FIV on account of the vibrations caused by the phase change, which may even go as far as to cause thermo-hydraulic instabilities.

Therefore, we will analyse two examples, fuel element vibrations in a flow with phase change and vibration induced by the leak flow in a BWR's jet pumps.

In the latter case, the jet pump FIV can be modelled as a vibration induced by the external axial flow. This is so because the leak flow through the Slip-Joint is deemed to be external to the mixer, as can be seen from Figure 7.

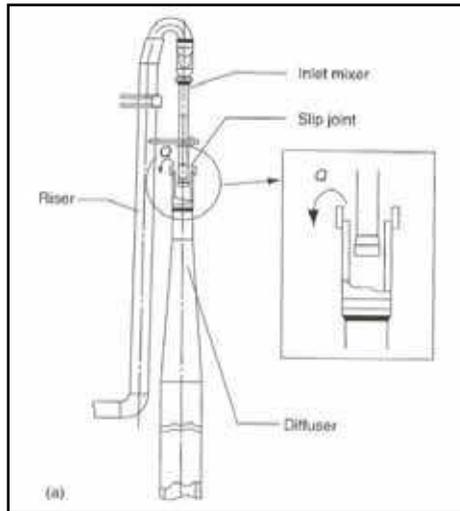


Fig. 7. JetPump and Slipjoint leak.

Thus, the model will be an axial flow cylinder, as described by Chen and Wambsgans (1970). The turbulent flow in the exterior, axial fluid gives rise to uneven pressure distribution on the outer wall of the pipe. Thus, on lacking balance it possesses resulting transverse forces. Moreover, as it is neither a permanent nor a stationary situation, the continuous change in pressure distribution over time, in space and in axial length, produces vibrations. The situation described is that which is shown in Figure 8.

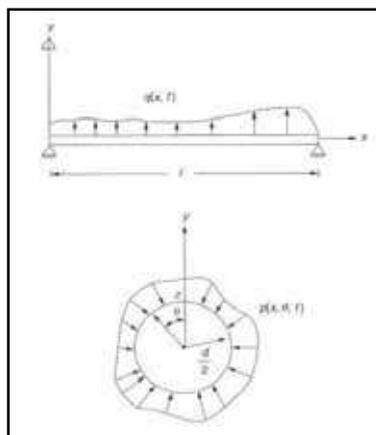


Fig. 8. Pressure distribution in an external flow pattern.

The Corcos (1963) model is used to analyse this type of situation. The model is expressed by:

$$\psi_{pp}(w, z_1, z_2, \theta_1, \theta_2) = \phi_{pp}(w) \cdot A\left(\frac{w|z_2 - z_1|}{V_c}\right) \cdot B\left(\frac{w \cdot D \cdot |\theta_2 - \theta_1|}{2 \cdot V_c}\right) \cdot e^{iw \frac{w|z_2 - z_1|}{V_c}} \quad (43)$$

where ψ_{pp} is the cross-spectral density of the pressure field, ϕ_{pp} is the pressure power spectrum at the point, A and B are spatial functions and V_c is the convection speed. ϕ_{pp} value is given by:

$$\phi_{pp}(fr) = \begin{cases} 0.272 \cdot 10^{-5} / fr^{0.25} & \text{si } fr < 5 \\ 22.75 \cdot 10^{-5} / fr^3 & \text{si } fr > 5 \end{cases} \quad (44)$$

where f_r is the reduced frequency, which is given by the following, where Dh is the hydraulic diameter:

$$fr = \frac{f \cdot Dh}{V} = \frac{w \cdot Dh}{2 \cdot \pi \cdot V} \quad (45)$$

The convection speed of the limit layer is given by:

$$\frac{V_c}{V} = 0.6 + 0.4e^{(-2.2 \cdot \frac{w \cdot \delta}{V})} \quad (46)$$

where

$$\delta = \frac{Dh}{2(n+1)} \quad (47)$$

$$n = 0.125 \cdot m^3 - 0.181 \cdot m^2 + 0.625 \cdot m + 5.851 \quad (48)$$

$$m = \log(Re) - 3 \quad (49)$$

Lastly, the vibration amplitude is given by the approximations:

$$y_{rms} = \begin{cases} V^{1.5} & \text{si } fr < 0.2 \\ V^2 & \text{si } 0.2 < fr < 3.5 \\ V^3 & \text{si } 3.5 < fr \end{cases} \quad (50)$$

These equations result in the generation of transversal forces, which move the mixer parts, the elbows and the riser. Moreover, as a result of its geometrical configuration, mode 3 vibration causes damage as a result of the stress at the first fixed point of the system, which is the joint between the riser and its support. Indeed, it is this stress that provokes breakages. A detail of the turbulence associated with the simulation of slipjoint leak can be seen in Figure 9.

In general, the Païdousis (1973) model is the equation used for industrial settings.

As far as FIV caused by external flow with phase change is concerned, its study and development can basically be put down to the nuclear industry, after the development of boiling water reactors. At such plants, boiling occurs on the external part of fuel rods in the upflow. This study was later extended to the steam generator pipes of pressurised water reactors. Figure 10.

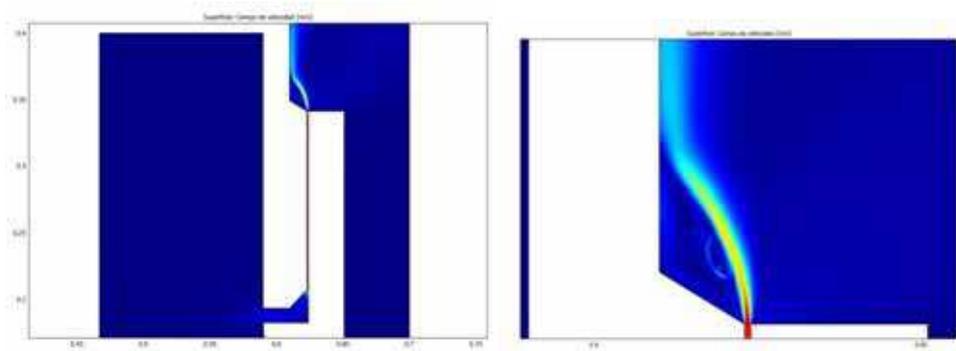


Fig. 9. Simulated Slipjoint leak and vortex induced.

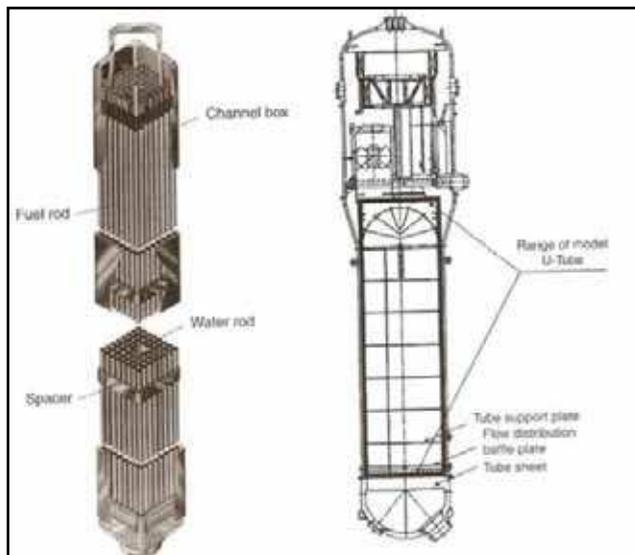


Fig. 10. Fuel Bundle and Steam Generator.

Both studies reached the same conclusion, namely that the vibration amplitude is proportional to the mass upflow, but decreases with pressure. As far as fluid quality, or the void ratio, is concerned, two peaks can be seen, 0.1-0.25 and 0.4-0.5, which suggests that there is great dependence between vibration amplitude and the void ratio, or fluid quality.

On a BWR (Figure x) type fuel bar, it was established that the pressure power spectrum of the excitation force is proportional to $V^{1.56-2.7}$, while the amplitude is proportional to $V^{0.78-1.35}$. These values were determined in a rod test carried out by Pettigrew and Taylor [13] under the following conditions:

Pressure: 2.8-9 MPa

Mass flow: 0-4600 Kg/m²s

Power: 1-1000 Kw

Quality: 0-0.25.

Saito (2002) came to the same conclusions in another test under different conditions.

5. Fluid Induced Noise (FIN)

In this section, we will take a look at the noise generated by a turbulent flow. Throughout this chapter and the following sections, fluid dynamics is based on movement equations, with speed as the unknown, essential analysis value.

For this particular case of noise induced by turbulence, a change of mindframe is called for in the sense of assessing the fluid in terms of pressure, and even density. Original FIN theory and expressions are given by the Lighthill aeroacoustic model (1952/1954), as indicated below:

$$\nabla p^2 - \frac{1}{c_0^2} \frac{\partial^2 p}{\partial t^2} = - \frac{\partial^2 T_{ij}}{\partial x_i \partial x_j} \quad (51)$$

$$T_{ij} = \rho v_i v_j \quad (52)$$

where T_{ij} is the turbulent stress tensor. Whenever the fluid can be compressed, pressure variation is accompanied by density variation, the expression of which is:

$$\frac{\partial^2 \rho}{\partial t^2} - c_0 \frac{\partial^2 \rho}{\partial x_i \partial x_j} = \rho_0 \frac{\partial^2 v_i v_j}{\partial x_i \partial x_j} \quad (53)$$

Thus, turbulence collaterally generates pressure and density variation in the fluid. By means of the turbulent stress tensor the turbulence produces variations in pressure and density. The former cause the noise, and as such, are deemed to be sound sources.

There are many practical applications of the analysis of turbulence generated noise. Two particularly curious, albeit useful ones, have solved some serious problems. The two situations in question are:

- Determination of leaks through the seat of safety relief valves from the outside by means of non-intrusive techniques.
- Element breakage due to resonance frequencies.

The first of the above situations has been used to detect safety relief valve leaks in BWR nuclear plants. The theoretical principle employed is that the seat leak flow produces turbulence which in turn generates a characteristic sound. The conceptual model is similar to the one used by Van Herpe and Creghton (1994), in which they model the fluid flow through a conduit with a restriction inside. These researchers came to the conclusion that the acoustic power, dB, is proportional to the speed in the conduit and to the passage diameter determined by the restricting element. Consequently, an SRV leak can be detected by means of sound and its register, but also by its evolution. The latter claim is based on the tests done by EPRI on this particular study NP-2444-SY. In this study, the frequency band is established at which the leak is best detected, 40-55 KHz, as well as the complete detection interval, 30-60KHz. Likewise, the author holds that regular monitoring can establish drift and trend patterns, the development mathematical law of which is that determined by Van Herpe y Creghton (1994).

The other case refers to the catastrophic breakages of steam dryers at BWR plants due to an acoustically sourced resonance. (Figure 11).

After the failure of the Quad cities dryer, a study was carried out on the loads to which it was subject. In none of the cases did the operating loads justify the breakage, or degradation, of the component. Consequently, a study was made of the vibratory or pulsatory phenomena, by measuring the vibrations at the component and the passage of

Thank You for previewing this eBook

You can read the full version of this eBook in different formats:

- HTML (Free /Available to everyone)
- PDF / TXT (Available to V.I.P. members. Free Standard members can access up to 5 PDF/TXT eBooks per month each month)
- Epub & Mobipocket (Exclusive to V.I.P. members)

To download this full book, simply select the format you desire below

