# 1

# Fluid–Structure Interaction



## 1.1. Introduction

Recently, several new problems have been formulated in the area of fluid–structure coupling, for example in the automotive industry with the dynamics of airbag inflation and fluid sloshing inside tanks; in aeronautics with the fluttering phenomenon affecting airplane wings, which involves a coupling between the vibrational dynamics of a structure and the flow of a fluid; and in the transportation industry with studies on noise reduction inside vehicles based on vibroacoustic analysis.

Each and every structure in contact with a fluid is subject to phenomena involving mechanical fluid–structure couplings to some extent. This kind of multiphysics coupling often significantly affects the dynamic behavior of mechanical systems. Taking it into account is one of the major challenges in calculating the dimensions of structures, especially when the objective is to ensure that their design meets the necessary safety requirements.

In this chapter, we will examine problems relating to the interaction of a structure with fluids both at rest and in flow. We will give a description of the motion of the fluid based on vibration theory, considering small vibrations in the structure and fluctuations in the pressure of the fluid around a stable equilibrium state, and we will present the relevant equations in the case of flowing fluids and the corresponding numerical methods for calculating couplings with dynamic structures.

## 1.2. Fluid–structure interaction problem

The mechanical coupling between the two media acts in both directions at their surface of contact: deformations in the structure resulting from the forces applied by the fluid flow modify the state of the fluid–structure interface; this affects the flow conditions of the fluid, which induces a change in the forces exerted on the structure at the interface, thus bringing the interaction cycle to a close.

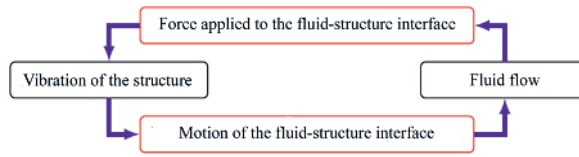


**Figure 1.1.** *Fluid–structure coupling mechanism. For a color version of this figure, see www.iste.co.uk/elhami/interactions.zip*

The fluid–structure interaction is described as the exchange of mechanical energy between a fluid and structure. This definition encompasses a wide range of problems. We can classify these problems using two criteria according to the physics of the problem at hand. The first criterion, proposed by Axisa [AXI 01], is based on the nature of the fluid flow. If the flow is negligible or non-existent, we say that the fluid is stagnant. Otherwise, we say that the fluid is flowing. In the first case, the objective is to describe small movements of the fluid and the structure around an equilibrium rest state. In these conditions, we choose to describe the dynamics of the interaction as a function of frequency; the equations describing the behavior of the structure and the fluid are written in terms of the reference (rest) state and generally lead to linear problems. In the second case, the objective is to establish a description of larger scale motion in the fluid and/or the structure. In these conditions, we choose to describe the dynamics of the interaction as a function of time; the equations describing the behavior of the structure and the fluid are written in terms of the current state of the system and generally lead to nonlinear problems.

The second criterion considers the coupling strength, which may be defined as the magnitude of the interactions or exchanges between the two media.

A coupling is said to be strong if there are high levels of exchange between the two media, i.e. the fluid has a significant impact on the structure, and vice versa. A coupling is said to be weak if the effect of one of the media dominates that of the other (Figure 1.2).

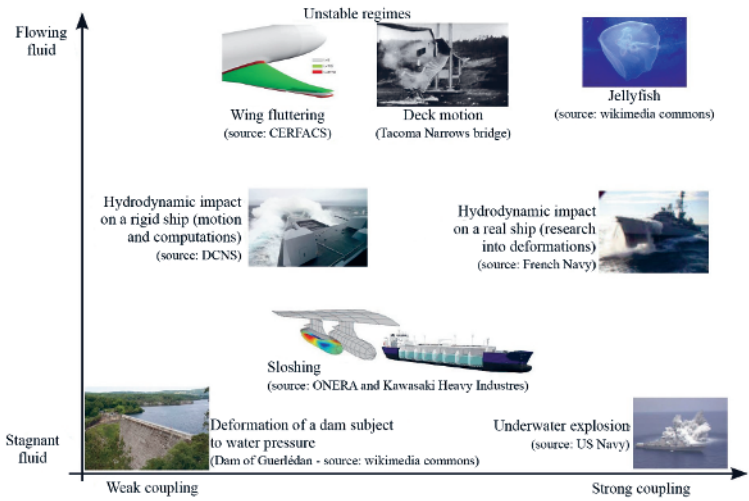


**Figure 1.2.** *Examples of fluid–structure interaction problems [GAU 11]. For a color version of this figure, see www.iste.co.uk/elhami/interactions.zip*

Three dimensionless numbers have been suggested to classify these problems [DEL 01]:

– The *mass number* $M_A$ is defined as the ratio between the density of the fluid $\rho_f$ and that of the structure $\rho_s$:

$$M_A = \frac{\rho_f}{\rho_s} \qquad [1.1]$$

This describes the significance of the inertial effects of the fluid and the structure. If its value is close to one, the inertial effects of the fluid are comparable to those of the structure, and so must be taken into account.

– The *Cauchy number* $C_y$ is the ratio between the dynamic pressure and the elasticity of the structure, which is quantified by Young's modulus $E$.

$$C_y = \frac{\rho_f V^2}{E} \qquad [1.2]$$

This indicates the significance of the deformations induced by the flow. If this number is small, i.e. if the structure is rigid or the fluid velocity is small, structural deformations are negligible.

– The reduced velocity $V_r$ is the ratio between the characteristic flow velocity and the velocity of wave propagation inside the structure:

$$V_r = \frac{V}{\sqrt{\frac{E}{\rho_s}}} = \frac{V}{c_s} \qquad [1.3]$$

If this number is large, the fluid dominates the problem from the perspective of time, and the dynamics of the structure are not important. By contrast, the dynamics of the structure increasingly dominate as this number tends to zero. If the number is close to 1, both dynamics carry similar weight in the problem.

These numbers are highly convenient for checking the importance of each phenomenon within the context of a given problem. However, as is the case for most dimensionless numbers, it is still difficult to define *a priori* threshold values applicable to all problems. In each problem, the large or small terms in the above will correspond to very different numerical values.

Using numerical simulations allows us to understand and predict the dynamic behavior of structures coupled with fluids, which is valuable in a number of industrial sectors. The numerical methods that we will use require us to solve the mathematical equations that model the behavior of the coupled fluid–structure system.

In general, the formulation of a coupled problem is based on the following description:

– the structure problem is formulated in terms of the displacement; the goal is to describe the behavior of the structure as a function of the displacement $u$, strain $\varepsilon(u)$, stress $\sigma(u)$, and to solve the equations of this dynamic to find the $u$, $\varepsilon(u)$ and $\sigma(u)$ fields in the structure domain;

– the fluid problem is formulated in terms of the pressure/velocity; the goal is to describe the behavior of the fluid as a function of the pressure $p$ and the velocity $v$, to solve the equations of conservation of mass and momentum and to find the $p$ and $v$ fields in the fluid domain;

– at the fluid–structure interface, the mechanical exchanges are represented, on the one hand, by considering the force $\varphi$ exerted by the fluid as a boundary condition for the structure problem and, on the other and, by considering the velocity $\frac{\partial u}{\partial t}$ imposed by the structure as a boundary condition for the fluid problem.

The energy exchanges between the fluid and the structure occur simultaneously. This needs to be taken into account by the numerical simulation. Coupled simulations can implement a single computational program to simultaneously solve the equations of the fluid and structure problems or alternatively can have two separate programs, one dedicated to the fluid problem and the other to the structure problem. The degree of complexity of the numerical simulation depends on the problem and methods of spatial and temporal discretization used to solve the equations of the problem.

Figure 1.3 proposes an overview of the most suitable general methods for simulating fluid–structure interaction problems:

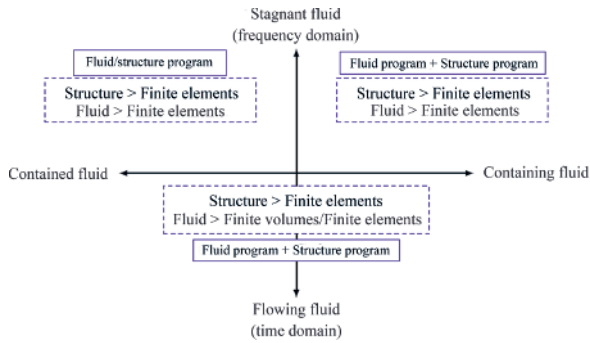


**Figure 1.3.** *General methods for numerically simulating fluid–structure interactions*

### 1.2.1. *Fluid–structure coupling methods*

There are several suitable coupling methods for the kinds of problems that we typically encounter. The following methods are used for stagnant fluids:

– The *decoupled method* finds the load or hydrostatic pressure on the structure, and then uses the results as an input to solve the deformation in the structure problem.

– *Acoustic fluid formulations (in terms of frequency)* allow small displacements around the equilibrium position of a structure to be determined. If the fluid is heavy, the vibrations of the structure and the fluid are strongly coupled. This coupling is reflected in the distinct natural frequencies of the modes, and the shapes of these modes. These methods use formulations that can be either non-symmetric $(u, p)$ or symmetric $(u, p, \varphi)$. They were

proposed by Morand and Ohayon [MOR 95] and illustrated by Sigrist [SIG 11].

Frequency-based formulations of the fluid potential are applicable to problems with stagnant fluids, but can also be used to describe the elevation of a free surface subject to sloshing. The goal is to determine the motion of the free surface in order to find the pressure variations along the walls. These methods are similar to acoustic fluid formulations, which use either symmetric or non-symmetric expressions for the coupling equations, written as $(u, p_0)$ and $(u, h, \varphi)$ [SIG 11].

In this book, we will consider fluids that are flowing. It is important to note that the solutions of flow problems are based on Eulerian formulations. This kind of formulation is particularly well suited to the study of flows and greatly simplifies the process of solving the equations of the fluid problem.

Solving the deformation of a structure more naturally leads to a Lagrangian description. Expressing the interaction problem between a flowing fluid and a structure introduces an additional complication into the choice of formulation for the problem, as this formulation must be compatible with the models of both the fluid and the structure. Existing methods tackle this issue in different ways, which allows different levels of interaction to be taken into account.

– *Monolithic approaches* solve the stated problem as a single block. We can make a distinction between monolithic formulation and monolithic solving. The former category of approach describes and solves the problem with a formulation that is either Lagrangian or Eulerian. This gives the closest solution to the actual physical problem, but this solution is also the most difficult to formulate and solve. We can, for example, refer to the work by Morinishi and Fukui [MOR 12], who used an Eulerian formulation, and the work by Dermidzic and Muzaferija [DER 95], who additionally consider thermal effects. The latter type of approach, monolithic solving, describes the problem using both Eulerian and Lagrangian formulations simultaneously. It then merges and solves these two formulations in a single system. The methods commonly used for this are based on virtual domains with Lagrange multipliers or penalty-based methods [DIN 07, AQU 04], and immersed boundary methods [ZHA 07]. Hübner *et al*. [HÜB 04] implemented monolithic solving by computing the advancement of time with a finite element method that allows the fluid and structure equations to be solved simultaneously.

Both approaches are particularly suitable for strongly coupled problems. However, they are complex to implement, and currently they have been neither implemented nor validated by any widely available programs.

– *Partitioned approaches* separate the problem into fluid and structure subproblems, each of which is solved using a dedicated program. These two programs communicate with each other to exchange parameter values (pressure on the structure, displacement of the fluid–structure interface) required to perform the calculations. To improve the accuracy of the calculation, internal loops are iteratively executed at each time step to allow the parameter values exchanged by the fluid and structure solvers to converge. Using two distinct programs allows us to choose the most suitable solving method for each of the fluid and structure problems, and enables us to fully utilize all of the tools available to us. The difficulty with these methods lies in ensuring the quality of the coupling between the solvers and guaranteeing that all solvers converge properly.

– *Chained approaches* are a simplified form of partitioned methods. The same methods are generally used, but without internal iterations at each time step. These methods are suitable for weak couplings in which the deformations experienced by the structure are limited.

– *Rigid body methods* are used in cases where the transfer of mechanical energy causes the structure to move, but with negligible deformation. Since the motion is driven either by the flow or by gravity, there exists a coupling between the motion of the solid and that of the fluid. These types of problem are solved by finding the forces exerted by the fluid on the structure and using them to solve an equation of motion with six degrees of freedom (DOF; three translations and three rotations). There are two major methods for doing this. The first, known as 6 DOF, is commonly available in general purpose Navier–Stokes solvers. The second is the method of immersed boundaries, which is more complex and less widely employed.

The quality of the coupling depends not only on the quality of the solution of each system (fluid and structure), but also on the quality of the coupling algorithm [PIP 95]. Consequently, to achieve higher order couplings, the fluid and structure systems must have order greater than or equal to that of the coupling algorithm in time and space, which shows the importance of the choice of algorithm. These computation times are a consequence of the complexity of the system of fluid mechanics equations that must be solved. This system requires smaller time steps and a finer mesh than is usually necessary for the fluid component, which determines the number of unknowns of the problem. The computation time required by the dynamic mesh depends strongly on the amplitude of the motion of the structure and the algorithms used to calculate the displacement of the mesh nodes and recalculate the mesh if necessary.

To implement a partitioned coupling, we need coupling algorithms to connect the fluid and structure programs together, as well as the actual solvers themselves. These algorithms must allow the solvers to synchronize and exchange data. Synchronization consists of sending the right data at the right times, and executing iterative loops or establishing useful predictions. The data exchange protocol needs to guarantee that the information sent from one solver to the other is interpolated in an acceptable manner between the meshes of the two solvers. These two families of algorithms are further subdivided into the categories of temporal and spatial coupling.

### 1.2.2. *Temporal coupling*

Partitioned algorithms have the advantage of requiring less computation time than other methods of temporal coupling. However, they introduce an error into the information exchange between the fluid and structure components. This error is reduced when using models with predictors [FAR 98, PIP 95] and can be almost completely eliminated by using implicit algorithms.

#### *Explicit synchronous method*

The coupling algorithm is illustrated schematically in Figure 1.4. In this algorithm, the fluid and structure components are solved at the same time step using the prediction of the displacement of the mesh at time $t^{n+1}$ as a function of the variables calculated at times $t^n$ and $t^{n-1}$ [SOU 10]. There are several possible forms of predictor; the choice of parameters affects the precision and stability of the models [SIG 10].

The forces calculated by the fluid program are not directly transmitted to the structure program; first, they are averaged: this allows the errors arising from the prediction of the displacement to be reduced. The averaging methods used for this are presented in [BEN 07].

#### *Explicit asynchronous or shifted method*

In this algorithm, the fluid iterations are shifted relative to those of the structure. The prediction is established for an intermediate time using a method similar to the one described above. The algorithm is presented in Figure 1.5.

#### *Implicit method*

This algorithm introduces an iterative loop to ensure that the predicted and calculated displacements converge (see Figure 1.6). This greatly reduces the

errors arising from the prediction. Convergence criteria with higher levels of precision require longer computation times. This solver is more stable than explicit schemes.

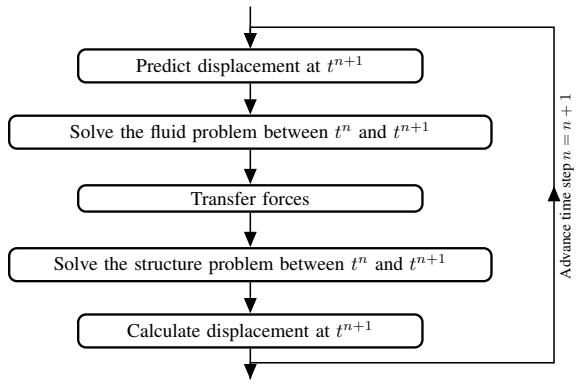


**Figure 1.4.** *Diagram of the explicit synchronous temporal coupling algorithm*

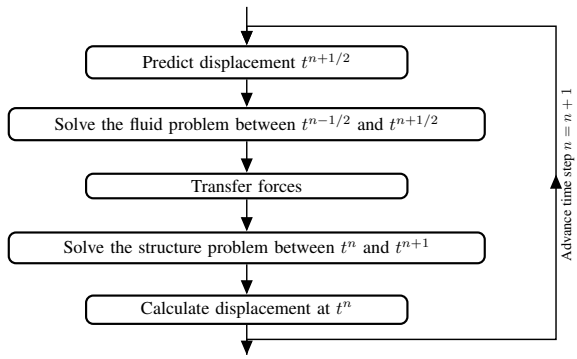


**Figure 1.5.** *Diagram of the explicit temporal asynchronous or shifted coupling algorithm*

Instability arises when the effects of added mass are non-negligible and the dimensions of the fluid domain are too large relative to the structure [CAU 04]. Hybrid methods have been developed to improve the stability of explicit methods [FER 05, FER 06, GER 03]. These methods determine the coupling forces implicitly and other coupling terms explicitly.

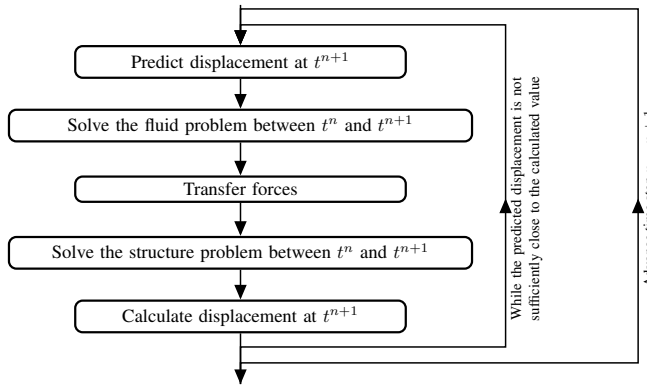


**Figure 1.6.** *Diagram of the implicit temporal coupling algorithm*

To evaluate the accuracy of the results, a comparative study of different models was performed on a system with two masses connected by a spring [BEN 07]. It was shown that the monolithic approach gives the best results, but that the results of this approach are similar to those obtained from the implicit method and same order as those obtained by explicit methods when the system includes damping. It was also shown that the correction step is not necessarily required for the forces in synchronous implicit and explicit methods, although it remains useful for asynchronous explicit methods. Piperno [PIP 97] showed that schemes with predictors are superior to schemes without predictors. On the other hand, an explicit scheme with a predictor does not perfectly conserve continuity in the displacements between the fluid and the structure.

Other studies [FAR 00] have shown that serial algorithms with predictors that are chosen to conserve continuity conditions for the velocity at the interface produce better results than other methods both with and without predictors. They also show that parallel models are less stable than the equivalent serial models, which further demonstrates that the choice of predictor significantly affects the quality of the coupling. Piperno and Farhat [PIP 00] show the benefit of predictors in both synchronous and asynchronous explicit methods, allowing higher order coupling algorithms that better conserve the energy balance of the coupled system to be found. Piperno *et al.* [PIP 05] and Piperno and Farhat [PIP 01] present a comparison criterion for temporal coupling models based on the energy exchanged at the interface.

### 1.2.3. *Spatial coupling*

The forces exerted by the fluid on the structure and the displacements induced by the structure on the fluid are exchanged through meshes of nodes. The difficulty with spatial coupling is that information must be transmitted losslessly between these meshes without introducing an error [FAR 98, MAM 95]. In most cases, the structure and fluid meshes do not coincide. Spatial coupling can therefore be thought of as two separate phases: first, the nodes of one mesh are projected onto the other, and then the parameter values are determined at these nodes.

#### *Projecting nodes*

The simplest method is to project the nodes of one mesh onto the other along the normal of one of the two meshes. The choice of normal and projected mesh is significant, as it will partially determine the quality of the spatial coupling [MAM 95].

Figure 1.7 shows an example in which the nodes of a structure mesh have been projected onto a fluid mesh. Note that projection is not possible for the structure node $S_5$. The value at this node is reassigned to the closest fluid node, which is $F_8$. There are two possible projections, $d_1$ and $d_2$, for the structure node $S_2$. We choose the smaller of the two, which in this case is $d_2$. The fluid element bounded by the nodes $F_2$ and $F_3$ does not contain any information originating directly from the structure.

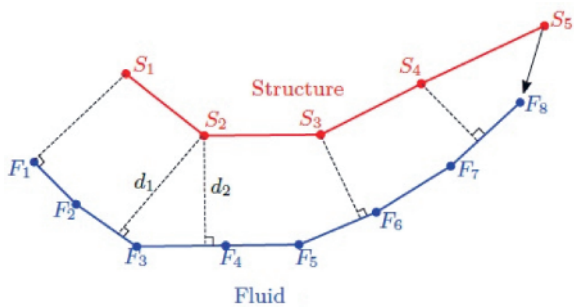


**Figure 1.7.** *Example of projection between two meshes. For a color version of this figure, see www.iste.co.uk/elhami/interactions.zip*

In Figure 1.7, we chose to project the structure nodes onto the fluid mesh. This solution results in fewer projections, since the structure mesh is usually less dense; however, the reverse solution is also possible. Projecting the nodes

of both meshes onto an intermediate coupling element has also been suggested [FAR 98].

To improve the implementation of this method, we can subdivide the interface into areas known as *buckets*. Each area may then be processed independently of the others, with special algorithms to handle cases where only one of the two interfaces is present in a given area. Figure 1.8 shows the concept of buckets for the previous example.

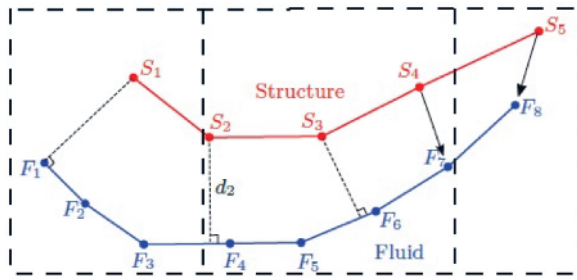


**Figure 1.8.** *Example of a projection with a buckets method. For a color version of this figure, see www.iste.co.uk/elhami/interactions.zip*

### *Determining the parameter values at each node*

This presents the methods used by Ansys MFX to determine the parameter values at each node. The first method, conservation of profile, linearly interpolates parameter values at the projected nodes using an idea similar to that proposed by Piperno [PIP 97]. This method conserves the integral of the exchanged parameter value $I$ (see Figure 1.9). The second method, global conservation, determines the parameter values at the projected nodes using a shape function. It conserves the sum of the parameter values taken over the interface (see Figure 1.10).

As shown by Figures 1.9 and 1.10, the method of determining the parameter values at the nodes strongly affects the error introduced into the calculation by spatial discretization. The global method is usually better for discrete values that do not need to be conserved exactly. However, for forces and fluxes, keeping the integrals equal is important to ensure that the energy exchanged by the fluid and the structure is conserved. Therefore, in these cases, the method of conservation of profile is better.

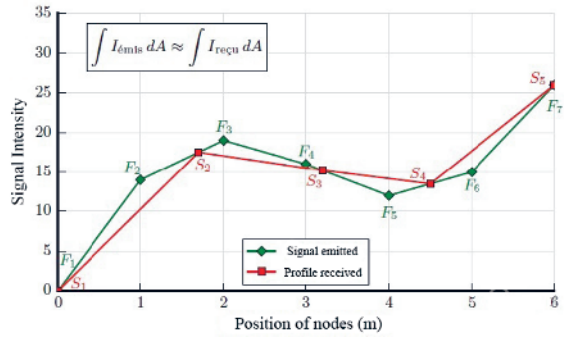


**Figure 1.9.** *Method of profile's conservation. For a color version of this figure, see www.iste.co.uk/elhami/interactions.zip*

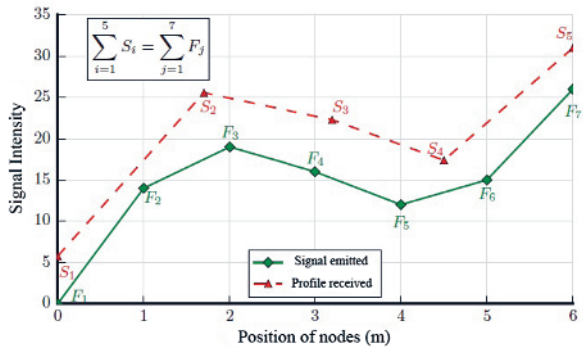


**Figure 1.10.** *Method of global conservation. For a color version of this figure, see www.iste.co.uk/elhami/interactions.zip*

### *General grid interface method*

One variant of the global method is the *general grid interface* (GGI) method. This method incorporates the steps of projection and interpolation. Figure 1.11 shows how this method works in a simple example.

First, the faces of each element of the interface are divided by the number of nodes on the face. These faces, which are described as IP, are converted into polygons composed of rows and columns of pixels with a resolution of $100 \times 100$. The polygons thus created on the emitting face (which in this case is the fluid mesh) are intersected with those on the receiving face. This constructs

the control surfaces. These control surfaces may then be used to exchange parameter values.

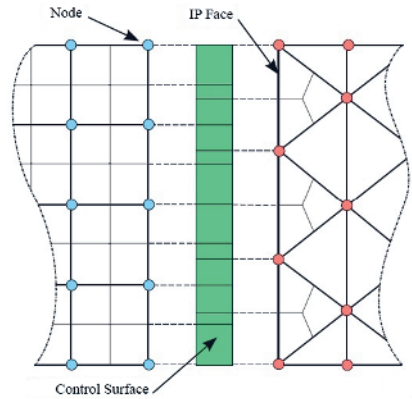


**Figure 1.11.** *General grid interface (GGI) exchange method. For a color version of this figure, see www.iste.co.uk/elhami/interactions.zip*

Other methods are possible, such as the closest neighbor method, which assigns the value of the projected node to the closest node in the receiving mesh. Other interpolation methods can also be used. Using a consistent method (i.e. the truncation error tends to zero as the grid spacing decreases) significantly improves the robustness and accuracy of the results [FAR 98].

## 1.3. Vibroacoustics

In many situations, sound is created by the interaction between the vibration of a structure and its acoustic radiation. This field of study, known by the name of vibroacoustics, has an extremely wide and diverse range of applications, from musical instruments to ultrasound scanning and active vibration control to the transmission of vibrations and sound between media with distinct propagation properties, or in the form of acoustic radiation. Studying the history of a field allows us to understand how both teaching and research have progressed within it. The classical approach to teaching vibrations focused on the mechanical response of a structure subjected to an idealized stimulus, neglecting to consider the surrounding environment. Later, the properties of fluids satisfying the assumption of incompressibility (sound waves do not propagate) were taken into account. Finally, the study of coupling phenomena and fluid–structure interactions led to vibroacoustics as we know it today.

Several different reasons motivated these shifts in perspective:

– difficulties encountered in the fields of nuclear, spatial and aeronautical research, and later also in land-based forms of transport, inspired a large amount of research into improving the reliability of mechanical systems;

– the development of increasingly powerful software tools in the 1970s made it possible to apply a number of very old conceptual tools (ranging from Helmholtz's radiation integral from 1865 to Courant finite elements from 1940). These tools also enabled the development of increasingly refined models and led to highly efficient experimental methods for signal processing;

– the introduction of a number of national and international regulations created the need to reduce noise levels in both the industrial and construction sectors;

– competition prompted manufacturers to research, innovate and adopt new technologies, even in the absence of regulations (for example in the area of internal noise levels in cars, trains and planes).

The concepts of acoustic quality and sound design currently represent the culmination of these developments. By the very nature of the couplings on which it is based, the field of vibroacoustics requires a cross-disciplinary approach. Whether to study a musical instrument, or to estimate the level of internal noise in an airplane, the researcher must take into account each of the various sources and how they correlate with each other, the propagation pathways of the vibrational and acoustic energy, the properties of materials and systems assumed to be decoupled, and the type of coupling involved in order to ultimately formulate a description of the vibrational response of these structures, the pressure, the acoustic intensity, and the radiated power. When modeling the propagation through real environments, we can distinguish between two different cases. The first is the case of an interface between two propagating media. The reader would therefore be correct in suspecting that the second is that of an interface between one propagating and one non-propagating medium. In fact, the distinction is slightly more subtle, as in the first case we are only interested in propagating media of same type (two fluids), whereas in most cases physical problems involve propagation between two propagating media of different types, typically a fluid and an elastic solid. The complexity of vibrational coupling phenomena involving waves with different types of behavior makes them highly difficult to understand.

### 1.3.1. *Vibrations of three-dimensional solids*

Wave propagation in elastic media differs significantly from wave propagation in fluid media. For example, in a fluid, there is only one type of

wave, whereas in an infinite homogeneous and isotropic elastic solid there can be both longitudinal waves, in which the particles move in the direction of propagation, and independent transverse waves, in which the particles move orthogonally to the direction of propagation. These two types of wave are coupled through phenomena that occur near discontinuities (geometrical or mechanical). Most internal source mechanisms generate both types of wave simultaneously.

### *Mechanical wave propagation equation*

Consider a homogeneous and isotropic solid with density $\rho$, Young's modulus $(E)$, Poisson's ratio $(\nu)$ and Lamé coefficients $(\lambda, \mu)$, satisfying:

$$\lambda = \frac{E\nu}{(1-2\nu)(1+\nu)} \; ; \quad \mu = \frac{E}{2(1+\nu)}$$

Similarly to the fluid case, we are interested in how a perturbation propagates throughout the solid. In most cases, this hypothesis enables us to neglect all but the linear part of the motion, and we can assume that the displacements and strains that occur within the solid are sufficiently small. Given these assumptions, we know that the relation between the strain experienced by each point in the solid and the stress applied to these points is given by Hooke's law, which in the homogeneous and isotropic case may be written as:

$$\sigma = \lambda \mathrm{Tr}(D) + 2\mu D \qquad [1.4]$$

Recall the equation of conservation of momentum (with Einstein's summation convention for repeated indices) in terms of Euler variables:

$$\rho \frac{\partial^2 u_i}{\partial t^2} + \frac{\partial(\rho u_i u_j - \sigma_{ij})}{\partial x_j} = F_i \qquad [1.5]$$

where $u_i$ are the components of the displacement field of the solid, $\sigma_{ij}$ are the components of the stress tensor and $F_i$ are the components of the applied force.

Assuming that the strain is small, we can linearize this equation, which amounts to neglecting the $u_i u_j$ term. From Hooke's law, we deduce that:

$$\begin{aligned} \frac{\partial \sigma_{ij}}{\partial x_j} &= \frac{\partial \lambda d_{ll} \delta_{ij} + 2\mu d_{ij}}{\partial x_j} \\ &= \lambda \frac{\partial d_{ll}}{\partial x_i} + 2\mu \frac{\partial d_{ij}}{\partial x_j} \end{aligned}$$

where

$$d_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) \qquad [1.6]$$

Let:

$$\frac{\partial \sigma_{ij}}{\partial x_j} = \lambda \frac{\partial^2 u_j}{\partial x_i \partial x_j} + \mu\left(\frac{\partial^2 u_j}{\partial x_j^2} + \frac{\partial^2 u_j}{\partial x_i \partial x_j}\right)$$
$$= (\lambda + \mu)\frac{\partial^2 u_j}{\partial x_i \partial x_j} + \mu \frac{\partial^2 u_2}{\partial x_j^2}$$

We substitute this expression into the law of conservation of momentum. We obtain

$$\rho \frac{\partial^2 u_i}{\partial t^2} - (\lambda + \mu)\frac{\partial^2 u_j}{\partial x_i \partial x_j} - \mu \frac{\partial^2 u_2}{\partial x_j^2} = F_i \qquad [1.7]$$

This is the linearized equation of elastic wave propagation.

### 1.3.2. *Acoustics of fluids*

In order to ensure that the mathematical problem that models the propagation of sound is well posed when the domain in which propagation occurs contains obstacles, we need to introduce boundary conditions at the edges in the wave equation. If the domain is infinite, we impose "outgoing wave" type boundary conditions to satisfy the principle of conservation of energy. This essentially says that energy cannot come from infinity. In particular, when we calculate the Green's functions, we will choose the elementary solution that satisfies this condition.

The vibrational behavior of a fluid (small fluctuations around an equilibrium state) assumed to be homogeneous, perfect and initially at rest may be described by a scalar variable that characterizes the state of the fluid. The problem equations are obtained by linearizing the general equations for the fluid flow [AXI 01, GIB 86, MOR 95] (Navier–Stokes equations). These equations express the principles of conservation of mass and momentum, and may be written in the following forms:

– conservation of mass

$$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho v_j)}{\partial x_j} = 0 \qquad [1.8]$$

– conservation of momentum

$$\frac{\partial (\rho v_i)}{\partial t} + \frac{\partial (\rho v_i v_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \mu \frac{\partial^2 v_j}{\partial x_j^2} \qquad [1.9]$$

where $p$ and $v = (v_i)$ are the pressure and velocity fields of the fluid.

These equations are supplemented by the equation of state of the fluid, which may, for example, be written as:

$$p = \psi(\rho)$$

Linear vibrations of the fluid may be described using formulations based on the displacement [BER 04, HAM 78], the velocity potential [SIG 11], the displacement potential [BOU 87, SAN 88] or the pressure [MAR 78, SIG 07]. In the last of these cases, we examine changes in the pressure $p$ and density $\rho$ of a non-viscous fluid (i.e. such that $\mu = 0$) starting from some initial state. These values can be written using the following decomposition:

$$p(x,t) = p_0(x) + p'(x,t) \qquad \rho(x,t) = \rho_0(x) + \rho'(x,t)$$

in which we separate the fluctuating component $(p', \rho')$ from the steady component $(p_0, \rho_0)$, which characterizes the initial state of the fluid.

### 1.3.3. *Numerical methods for calculating a structure coupled with a stagnant fluid*

#### *Modeling assumptions*

In this section, we will discuss the numerical methods that may be used to solve a coupled fluid/structure problem in the case where the fluid is stagnant. The analysis framework is based on the vibrations in the structure and the fluid; in the fluid, we consider the propagation of waves arising from gravitational effects (at low frequencies) or compressibility (at high frequencies).

The state of the coupled systems is described by the value of the displacement field in the structure domain and by the value of the pressure

field in the fluid domain. The method of finite elements is useful for describing vibrations in elastic structures; it is equally suitable for describing vibrations in a fluid (gravitational or compressional waves) for continuous fluids (bounded domains) contained inside a structure; in this case, finite element (structure)/finite element (fluid) coupling can be used to solve the coupled problem. This type of method can be implemented using a finite elements approach: a coupling operator gives a rigorous description of the interaction in terms of action/reaction.

In the case of a fluid that contains a structure (non-bounded domains), the method of finite elements ceases to be applicable: the boundary element method may instead be used to take into account effects arising from the fact that the fluid domain is infinite. Finite element (structure)/boundary element (fluid) coupling can be used to solve the coupled problem: this category of method generally requires two distinct computational programs to be coupled together (one program for the finite elements and another for the boundary elements).

### *Vibrational analysis of elastic structures*

In linear problems involving small movements, the vibrations of the structure are expressed in terms of frequency; the local equilibrium equation, which is obtained from the fundamental principle of dynamics without external forces, can be written as follows in a Cartesian coordinate system:

$$\omega^2 \rho_S u_i + \frac{\partial \sigma_{ij}(u)}{\partial x_j} = 0 \quad \text{in} \quad \Omega_S \qquad [1.10]$$

Initially, the structure is considered on its own, i.e. without the fluid, so that $\Gamma = \emptyset$. The boundary conditions on the constrained boundary and the stress-free boundary may be written as:

$$u_i = 0 \quad \text{on} \quad \Gamma_{S_0} \qquad [1.11]$$

$$\sigma_{ij}(u) n_j^S = 0 \quad \text{on} \quad \Gamma_{S_\sigma} \qquad [1.12]$$

Grouping together equations [3.155]–[3.157], we deduce an equation for the displacement field only (Navier equation):

$$\rho_S \omega^2 u_i + (\lambda + \mu) \frac{\partial}{\partial x} \left( \frac{\partial u_j}{\partial x_j} \right) + \mu \frac{\partial^2 u_i}{\partial x_j \partial x_j} = 0 \qquad [1.13]$$

In the general case, it is not possible to find an analytical solution to equation [3.160] with boundary conditions [3.158] and [3.159]; an approximate solution can be obtained by discretizing a formulation based on weighted integrals that is equivalent to the initial problem. This formulation is derived using the method of test functions by considering an arbitrary displacement field $\delta u$, known as the virtual displacement field, that satisfies the boundary condition [3.158]. Multiplying [3.157] by $\delta u$ and integrating over the domain, we can write:

$$-\omega^2 \int_\Omega \rho u_i \delta u_i d\Omega - \int_\Omega \frac{\partial \sigma_{ij}(u)}{\partial x_j} \delta u_i d\Omega = 0 \qquad [1.14]$$

Using generalized integration by parts and taking into account the boundary conditions [3.158] and [3.159], we deduce the following integral formulation satisfied by any field $\delta u$:

$$-\omega^2 \int_{\Omega_S} \rho_S u_i \delta u_i d\Omega_S + \int_{\Omega_S} \sigma_{ij}(u) \varepsilon_{ij}(\delta u) d\Omega_S = 0 \qquad [1.15]$$

This integral formulation can then be discretized using numerical methods. While many approaches are possible (finite differences, boundary elements, distinct elements, etc.), the finite element method [BAT 82, ZIE 89, BEL 00] is currently the most widely employed in computational programs.

### *Vibrational analysis of stagnant fluids*

The Helmholtz equation describes the propagation of sound waves as a function of frequency:

$$-\frac{\omega^2}{c^2} p - \frac{\partial^2 p}{\partial x_i \partial x_i} = 0 \quad \text{in} \quad \Omega_F \qquad [1.16]$$

This equation comes with boundary conditions of the form:

$$\frac{\partial p}{\partial x_j} n_j^F = 0 \quad \text{on} \quad \Gamma_{F_\pi} \qquad [1.17]$$

$$p = 0 \quad \text{on} \quad \Gamma_{F_0} \qquad [1.18]$$

The boundary condition [1.17] models the presence of a fixed wall bounding the fluid domain: it says that the fluid flux at the wall is zero in the normal direction, but that fluid motion is possible in the tangential direction and independently from the motion of the wall (perfect fluid). This condition is also used to formulate a symmetry condition for the fluid problem. The boundary condition [1.18] models the existence of an acoustic free surface: it says that on this free surface, the (absolute) pressure is fixed at a given value by external factors, and so the pressure fluctuations are equal to zero. This condition is also used to formulate an antisymmetry condition for the fluid problem. Analytical solutions can sometimes be found for problems determined by the above equations, usually for elementary geometries. In the general case, approximate solutions can be found using numerical methods. The finite element method is particularly well suited to the problem as stated here; it requires us to pass via the weighted integral formulation, which is equivalent to [1.16]–[1.18]. This formulation is derived using the method of test functions by considering an arbitrary pressure field $\delta p$ that satisfies the boundary condition [1.18]. By multiplying [1.16] by $\delta p$ and integrating over the domain using generalized integration by parts and taking into account the boundary conditions [1.17] and [1.18], we deduce the following integral formulation satisfied by any field $\delta p$:

$$-\omega^2 \int_{\Omega_F} \frac{p\delta p}{c^2} d\Omega_F + \int_{\Omega_F} \frac{\partial p}{\partial x_i} \frac{\partial \delta p}{\partial x_i} d\Omega_F = 0 \qquad [1.19]$$

## 1.4. Aerodynamics

In general, aeroelasticity is the area of applied mechanics that deals with the motion of deformable bodies in gaseous flows (or hydroelasticity for fluid flows). Although aeroelasticity was originally studied in connection with aeronautics, because the types of problem considered by aeroelasticity proved to be critical in the early days of propulsion-based flight, aeroelastic phenomena have also played a very important role in other fields of applied science. For instance, civil engineering has a history of undertaking projects with ever bolder designs and ever higher flexibility (buildings, bridges, towers, power lines, etc.). Similarly, when designing turbomachines, hydroelectric projects, land or sea vehicles, etc., aeroelastic and hydroelastic problems are becoming ever more relevant. Hence, the field of aerolasticity remains highly significant in scientific and industrial research to this day.

Nevertheless, aerospace manufacturing is the area in which aeroelastic phenomena are most prominent. Consequently, since the Second World War,

great emphasis has been placed on aeroelastic phenomena when building high-speed (transonic and supersonic) and large-scale aircraft (gliders, etc.). In particular, for high-speed aircraft, it has been necessary to modify the geometry of the wings and introduce servocontrols, which has inspired new problems. The emergence of new materials (such as composite materials) has also prompted an increase in scale (and flexibility) in all categories of aircraft, which increases the significance of aeroelastic effects.

Aeroelasticity belongs to the family of phenomena with interactions between a flow and a structure. One characteristic feature of this kind of phenomenon is that, in some cases, the fluid can supply an indefinite quantity of energy to the structure, which leads to system instability. There is another way to characterize these phenomena. We can distinguish phenomena in which:

– the fluid and the structure have comparable densities. This is, for example, the case with immersed structures and physiological flows;

– the fluid and the structure have very different densities: usually, the density of the fluid is lower than that of the structure.

The aeroelastic phenomena that we will consider here belong to the second category, as the fluid is significantly "lighter" than the structure. One final property can be used to characterize aeroelasticity problems: the distinct spatial separation between the fluid and the structure. It is important to emphasize this feature, as it plays an important role in the choice of numerical method when computing these phenomena. Aeroelasticity may therefore be defined as the study of the elastic behavior of structures whose motion within the flow generates induced stress. This topic combines three disciplines:

– aerodynamics, to predict the forces experienced by the structure;

– elasticity, to determine alterations to the structure (displacements and deformations);

– structural dynamics, to determine the inertia matrices and modal properties (modes, natural frequencies) and in some cases the inertial forces (for motion involving non-uniform acceleration).

We will study two major phenomena:

– static phenomena: the structure experiences strain as a result of the aerodynamic forces that it applies to itself.

– dynamic phenomena: the fluid supplies energy to the structure, which may either amplify oscillatory motion or cause the system to break up if the maximal tolerances are exceeded. This phenomenon is called fluttering.

### 1.4.1. *Aeroelastic problems*

Aeroelastic problems arise from the interaction of different types of force:

– elastic forces, which are structural in origin;

– inertial forces, which are also structural;

– aerodynamic forces, which are induced by strain (steady or oscillatory) in the structure, and are the result of external perturbations.

Aeroelastic problems only arise because the aircraft structure is not perfectly rigid when an air flow is applied to it. Aircraft structures are always flexible to a greater or lesser extent, and this flexibility is the underlying cause of each observable aeroelastic phenomenon.

Inertial effects can also play a very important role, and so we will differentiate between *dynamic aeroelasticity*, which involves all three of the forces listed above, and *static aeroelasticity*, which only involves elastic and aerodynamic forces.

Classically, aeroelastic phenomena are classified using Collar's triangle of forces (Figure 1.12). The three types of force arising from motion (elastic, aerodynamic and inertial) are represented by the three vertices of the triangle. Each aeroelastic phenomenon can be situated on this diagram according to how it relates to each vertex. For instance, phenomena relating to dynamic aeroelasticity are located at the center of the triangle, whereas effects relating to static aeroelasticity are located on the left-hand side. The right-hand side groups together phenomena that only involve aerodynamic and inertial forces, such as the dynamics of rigid aircraft studied in *flight mechanics*. The base of the triangle corresponds to vibrational problems from *structural dynamics*.

The most important phenomenon of static aeroelasticity is *divergence*: above a certain speed, the equilibrium between aerodynamic forces and elastic spring forces becomes unstable, which leads to a sudden collapse of the structure.

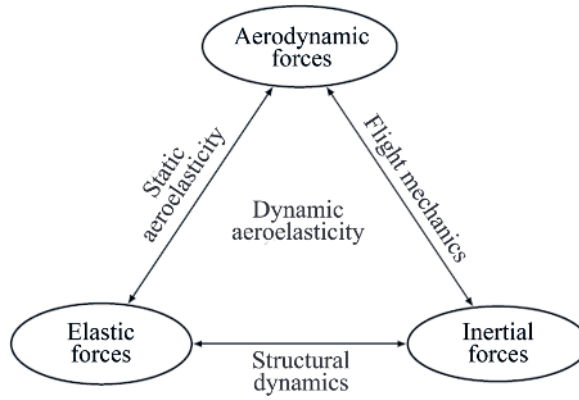


**Figure 1.12.** *Aeroelasticity triangle*

Dynamic aeroelasticity includes, in particular, the following phenomena:

– *fluttering*: this is an example of dynamic instability, coupling an unsteady flow with the vibratory motion of the structure, which either dampens or amplifies this vibration.

– *dynamic response*: the flexibility of the aircraft can significantly alter its response to atmospheric perturbations (gusts, turbulence) and fast maneuvers.

From a mathematical perspective, the study of instability is complementary to that of dynamic response. Indeed, instability conditions are generally determined by the presence of non-trivial solutions in a system of homogeneous equations, whereas the dynamic response is obtained by solving these same equations after adding a source term.

In air flows around a structure in motion, three (or more) dimensionless quantities can be used to classify the nature of the mechanisms that can be expected to occur:

– *The Reynolds number* is the ratio between the convected kinetic energy and the energy that is emitted and dissipated by friction in the fluid, defined as:

$$Re = \frac{VL}{\nu}$$

where $L$ is the wing length (in m), $V$ is the wind speed (in m/s) and $\nu$ is the kinematic viscosity (in m/s).

– *The Strouhal numberl* is related to the instability of boundary layers, and represents the ratio between the characteristic length and the distance travelled by a fluid particle at the characteristic speed over one characteristic period of the flow:

$$St = \frac{L}{V_{ref} T_{ref}}$$

– *The reduced frequency* $f_r$ determines which mode of operation is most appropriate for studying fluid–structure interaction phenomena in the presence of a flow. It is defined as the ratio between two characteristic periods of the system, the first of which is the time taken by a fluid particle to travel across the structure ($T_f$), and the second of which is the natural period of vibration of the structure ($T_s = \frac{1}{f_s}$). We therefore have that:

$$f_r = \frac{T_f}{T_s} = \frac{\frac{L}{V}}{T_s} = \frac{L f_s}{V}$$

The inverse of the reduced frequency is known as the reduced speed. $f_r$ is sometimes also defined in terms of the angular frequency instead of the natural frequency. From the perspective of aeroelastics, three different regimes can be distinguished according to their reduced frequency:

– if $f_r \gg 1$, physically this means that the motion of the fluid is slow relative to the motion of the structure. This domain is studied by the field of acoustics, or aeroacoustics if the fluid velocity is taken into consideration;

– if $f_r \simeq 1$, we are in the regime of strong interactions. In this regime, there exists some form of resonance, since each medium exerts stress upon the other with similar frequencies. This is the most complex regime and by far the most difficult to solve;

– when $f_r \ll 1$, the motion of the structure is slow relative to that of the fluid, which allows us to estimate the stresses applied to the structure by means of a permanent flow. This greatly simplifies the analysis and the formulations involved. This is the domain of quasi-steady aeroelasticity, which is extremely common in industrial applications, and for which a great number of reliable strategies have been developed and perfected.

When the reduced frequency takes very low values, we can justifiably assume that the oscillations do not produce any effect on the flow around the structure, and that changes in the motion of the solid are instantaneously applicable to the flow and the stresses exerted by it. Quasi-steady theory uses

static aerodynamic coefficients, i.e. coefficients evaluated on a stationary structure, to calculate changes in the fluttering.

### 1.4.2. *Aerodynamic loads*

Aerodynamic loads can be evaluated locally on a structure using the pressure distribution at the wall, or alternatively using a stress torsor.

#### *Action of the pressure and pressure coefficients*

The pressure acts according to Cauchy's principle along the normal to the wall and is proportional to the static pressure $p$. The elementary force $dF$ exerted by a surface element with unit normal vector $N$ may simply be written as:

$$dF = -pNds$$

The minus sign is included by convention and depends on the choice of direction for the vector $N$.

However, the action of the static pressure is not the only force applied locally. Viscous fluid flow creates a boundary layer characterized by its velocity gradient at the wall. Given the assumption of a Newtonian fluid, this gradient is linear and so results in shear stress. This adds an additional friction term to the pressure force that is ultimately due to the viscosity of the fluid.

In practice, for non-profiled structures, this friction term is negligible relative to the action of the pressure. But for structures with specific profiles such as the wings of an aircraft, the friction term contributes in equal measure to the drag force, and cannot be neglected.

In practice, the pressure is written in dimensionless form. Since it depends on the flow velocity, it is often expressed as a pressure coefficient $C_p$ given by:

$$C_p = \frac{p - p_{ref}}{q_{ref}}$$

where $q_{ref} = \frac{1}{2}\rho V_{ref}^2$.

The characteristic values are averages that are usually determined prior to modeling, for example using a Prandtl tube. The reference pressure that is chosen in practice is often the static pressure as determined in advance. Hence,

the pressure coefficient will never have a value greater than 1, as it follows from Bernouilli's theorem for steady flows that the maximum possible value of $p$ is the stagnation pressure $p_0$, which indeed has a maximum possible value of 1.

At the rear end of non-profiled structures, the pressure coefficient is negative, and does not have a theoretical lower bound. The drag of this type of structure is primarily caused by a low-pressure area in a phenomenon known as the base flow effect.

The pressure coefficient is a parameter that is generally independent of the flow rate, and depends solely on the geometry of the structure. It can depend on the Reynolds number, in particular due to variations in the positions of the points at which the boundary layer detaches and reattaches on non-profiled structures.

### *Aerodynamic forces and moments*

If the pressure distribution along the wall is known, it is relatively straightforward to integrate it to find the global forces. But in some situations we might also wish to measure these forces directly using aerodynamic moments. In this case, it is essential to first define the reference point with respect to which the torque is defined. Researchers in aerodynamics traditionally express the forces in terms of wind coordinates (by convention the Eiffel coordinate system), but when computing the structure it is often more convenient to work in structure coordinates (known as the Lilienthal coordinate system) [DES 08]. Using the notation in Figure 1.13, the wind coordinate system with axes $(O, D)$ and $(O, L)$ is obtained by rotating through angle $\alpha$ from the structure coordinate system with axes $(O, x)$ and $(O, z)$. In the three-dimensional case, we can add the $(O, y)$ axis, which is perpendicular to the plane shown in the figure. All of these are examples of right-handed coordinate systems. The angle $\alpha$ is the angle of incidence.

The three-dimensional stress torsor in structure coordinates is composed of three forces:

– $F_x$: drag force;

– $F_y$: cross-wind force;

– $F_z$: lift force;

and three moments:

– $M_x$: rolling moment;

– $M_y$: pitching moment;

– $M_z$: yawing moment.

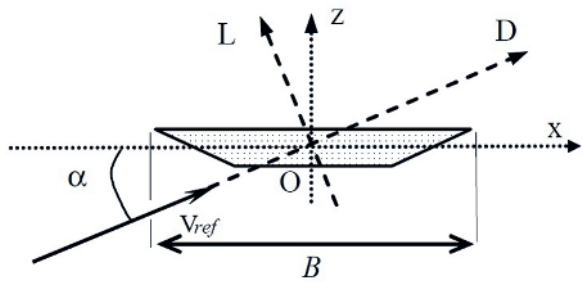


**Figure 1.13.** *Notation of two-dimensional coordinate systems*

In the two-dimensional case, if we consider the vertical plane, only the drag and lift forces and the pitching moment are significant. The aerodynamic forces are usually given in the form of dimensionless coefficients, which makes it easy to transpose forces measured on models over to physical structures. We thus define the force coefficients as:

$$C_x = \frac{F_x}{\frac{1}{2}\rho V_{ref}^2 S} \quad [1.20]$$

$$C_y = \frac{F_y}{\frac{1}{2}\rho V_{ref}^2 S} \quad [1.21]$$

$$C_z = \frac{F_z}{\frac{1}{2}\rho V_{ref}^2 S} \quad [1.22]$$

where $S$ is a reference surface, for example the deck surface of a bridge.

When the forces are given in two dimensions, the forces are given per unit length and the reference surface $S$ is usually replaced with the width $B$ of the structure. The drag and lift coefficients in wind coordinates are written as $C_D$ and $C_L$, respectively.

The moment coefficients are given by:

$$C_L = \frac{M_x}{\frac{1}{2}\rho V_{ref}^2 SB} \quad [1.23]$$

$$C_M = \frac{M_y}{\frac{1}{2}\rho V_{ref}^2 SB} \quad [1.24]$$

$$C_N = \frac{M_z}{\frac{1}{2}\rho V_{ref}^2 SB} \qquad [1.25]$$

In two dimensions, only the pitching coefficient $C_M$ is useful; it characterizes the torque per unit length. In general, we choose the axis that passes through the center of the structure to define the pitching coefficient.

### 1.4.3. *Problem equations*

To model aeroelastic phenomena, we have to simultaneously solve the fluid mechanics equations and the structural mechanics equations.

#### *Flow equations*

The description of fluid flows is generally based on an Eulerian formulation that allows us to examine the flow properties in each of the different regions of the fluid domain. We will restrict ourselves to the case of linear viscous fluids, commonly known as Newtonian fluids, which are by far the most important in practical applications. Newtonian fluids are characterized by their behavioral equations, such as the fact that the relation between the viscous stress $\sigma$ and the strain is linear:

$$\sigma_{ij} = \mu\left(\frac{\partial v_i}{\partial x_j} + \frac{\partial v_j}{\partial x_i} - \frac{2}{3}\frac{\partial v_k}{\partial x_k}\delta_{ij}\right) - p\delta_{ij} \qquad [1.26]$$

where $v_i$ is the velocity vector with Cartesian coordinates $x_i$, $p$ is the pressure, $\mu$ is the dynamic viscosity and $\delta_{ij}$ is the Kronecker symbol.

The laws of conservation of mass, motion and energy may therefore be written as:

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho v_i)}{\partial x_i} = 0 \qquad [1.27]$$

$$\frac{\partial(\rho v_i)}{\partial t} + \frac{\partial(\rho v_i v_j)}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial v_i}{\partial x_j} + \frac{\partial v_j}{\partial x_i} - \frac{2}{3}\frac{\partial v_k}{\partial x_k}\delta_{ij}\right)\right] - \frac{\partial p}{\partial x_i} + \rho f_i \qquad [1.28]$$

$$\frac{\partial(\rho e)}{\partial t} + \frac{\partial(\rho v_i e)}{\partial x_i} = \mu\left[\frac{\partial v_i}{\partial x_j}\left(\frac{\partial v_i}{\partial x_j} + \frac{\partial v_j}{\partial x_i}\right) - \frac{2}{3}\left(\frac{\partial v_i}{\partial x_i}\right)^2\right] - p\frac{\partial v_i}{\partial x_i} + \frac{\partial}{\partial x_i}\left(\kappa\frac{\partial T}{\partial x_i}\right) + \rho q \qquad [1.29]$$

where $\rho$ is the density of the fluid, $e$ is the specific internal energy, the $f_i$ are the external forces and $q$ represents the heat sources.

For the heat flux $h_i$ in the scalar equation [1.29], we used Fourier's law:

$$h_i = -\kappa_f \frac{\partial T}{\partial x_i} \qquad [1.30]$$

where $\kappa_f$ is the thermal conductivity, the specific heat capacity is assumed to be constant, and the work performed by the pressure and friction forces is neglected. System [1.27]–[1.29] needs to be completed by two equations of state of the form:

$$p = p(\rho, T) \quad \text{and} \quad e = e(\rho, T) \qquad [1.31]$$

These relations determine the thermodynamic properties of the fluid. They are known as the thermal and caloric equations of state.

Usually, it is not necessary to solve the system of equations in its general form, and additional assumptions can be made to simplify the system. From a practical standpoint, the most useful assumptions are those of incompressibility and non-viscosity [SIG 11].

We will consider the Navier–Stokes equations describing the flow of incompressible Newtonian viscous fluids. This equations can be expressed in the fluid-filled domain $\Omega^f$ using the equations of conservation of momentum and the incompressibility condition:

$$\rho_f(\frac{\partial v}{\partial t} + v \cdot \nabla v) = -\nabla p + \mu_f \Delta v + \rho_f b_f \qquad [1.32]$$

$$\nabla \cdot v = 0 \qquad [1.33]$$

Equations [3.80] and [3.81] can be further expressed in dimensionless form by setting:

$$v_a = \frac{v}{V}, x_a = \frac{x}{L}, p_a = \frac{p - p_0}{\rho_0 V^2}, b_{fa} = \frac{b_f L}{V^2} \text{ and } t_a = \frac{tV}{L}$$

where $x$ are the spatial coordinates, $p_0$ is a reference pressure, $L$ and $V$ are the characteristic length and velocity, respectively. Hence, by applying these relations to [3.80] and [3.81] and omitting the $a$ index, we obtain the following equations:

$$\frac{\partial v}{\partial t} + v \cdot \nabla v = -\nabla p + \frac{1}{Re}\Delta v + b_f \qquad [1.34]$$

$$\nabla \cdot v = 0 \qquad [1.35]$$

We write $\Gamma_D$ (or $\Gamma_N$) the boundary of $\Omega^f$ to which the Dirichlet boundary conditions (or surface stress conditions) are applied.

On $\Gamma_D$, writing $\bar{v}|_{\Gamma_D}$ for the Dirichlet data functions, we have

$$v = \bar{v}|_{\Gamma_D} \qquad [1.36]$$

By definition, the function $t_f$ that describes the density of the contact forces on the fluid may be written as:

$$t_f = \sigma_f n_f \qquad [1.37]$$

where $n_f$ is the outward unit normal vector on $\Gamma_N$.

On the edge $\Gamma_N$, we take boundary conditions [PIR 89, DEV 02] such that:

$$t_f = -pI_n + 2\mu_f d(v)n = \bar{t}$$

where $n = n_f$ and $\bar{t}$ is an imposed stress on $\Gamma_N$.

### *Equations describing the dynamics of elastic solids*

In structural mechanics, in general, the goal is to determine the strain in a solid body induced by the action of various forces and to deduce the corresponding stresses, which are very important in a number of applications. There are many different laws that describe the various material properties. Combined with the equations describing the dynamics, these laws produce complex systems that enable us to find the strain (or the displacement).

### *Fundamental equations*

In the field of structural dynamics, a distinction is usually made between linear and nonlinear models, which can either be geometric or physical. From a geometric perspective, linear problems are characterized by the linear relation between the stress and the strain, such that the following expression holds for the strain tensor $\varepsilon_{ij}$:

$$\varepsilon_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) \qquad [1.38]$$

where $u_i$ is the displacement vector. Physically, linear problems are based on a material law involving a linear relation between the strain and the stress. We will restrict ourselves to the case of linear elasticity, which is the most relevant in many industrial applications.

The theory of linear elasticity is geometrically and physically linear. As stated above, there is no need to distinguish between Eulerian and Lagrangian descriptions in the context of a geometrically linear theory.

The equations of linear elasticity theory are derived from linear stress–strain relations [1.38], which are conservation laws formulated in terms of the displacement [3.165] (known as equations of motion),

$$\rho\frac{D^2u_i}{Dt^2} = \frac{\partial\sigma_{s_{ij}}}{\partial x_j} + \rho_s f_i \qquad [1.39]$$

and the assumption of linear elasticity in the material behavior, which is expressed by a constitutive equation of the form:

$$\sigma_{S_{ij}} = \lambda\varepsilon_{kk}\delta_{ij} + 2\mu\varepsilon_{ij} \qquad [1.40]$$

Finally, by eliminating $\varepsilon_{ij}$ and $T_{ij}$, equations [1.38], [3.85] and [3.87] may be rewritten in the form of the following system of differential equations based on the displacement $u_i$:

$$\rho_S\frac{D^2u_i}{Dt^2} = (\lambda+\mu)\frac{\partial^2u_j}{\partial x_i\partial x_j} + \mu\frac{\partial^2u_i}{\partial x_j\partial x_i} + \rho_s f_i \qquad [1.41]$$

These (unsteady) equations are known as the Navier–Cauchy equations for linear elasticity. In steady problems, we have that:

$$(\lambda+\mu)\frac{\partial^2 u_j}{\partial x_i \partial x_j}+\mu\frac{\partial^2 u_i}{\partial x_j \partial x_i}+\rho_S f_i=0 \qquad [1.42]$$

In the solid, we denote $\Gamma_D^s$ the edge with the Dirichlet boundary condition and $\Gamma_N^s$ the section of the edge with surface stress conditions. Thus, on $\Gamma_D^s$, we can write that:

$$u=\bar{u}|_{\Gamma_D^s} \qquad [1.43]$$

Depending on the type of geometric model that will be considered later, we can fix internal conditions that complement the Dirichlet condition on the surface of the structure. Similarly, we denote $t_s$ the density of the contact forces on $\Gamma_N^s$ such that:

$$t_s=\sigma_s n_s \qquad [1.44]$$

where $n_s$ is the outward unit normal vector from the solid.

### *Boundary conditions at the interface*

The fluid–structure interaction describes the mechanical problem of the contact between an elastic solid body and a viscous fluid in motion. This essential step in our problem requires us to specify additional boundary conditions to describe the dialog between the fluid and the solid. Consider two domains, one fluid domain $\Omega^f$ and another solid domain $\Omega^s$, in contact with each other along the interface $\Gamma_I$. We will use two conditions at the interface (Figure 1.14). On the one hand, we assume that the velocities are continuous with respect to time at the interface. This implies that

$$v=\dot{u} \quad \text{on} \quad \Gamma_I(t) \qquad [1.45]$$

On the other hand, we assume that the interface is in mechanical equilibrium

$$t_f+t_s=0 \quad \text{on} \quad \Gamma_I(t) \qquad [1.46]$$

This second equation expresses the principle of the reciprocal action of forces at the interface.

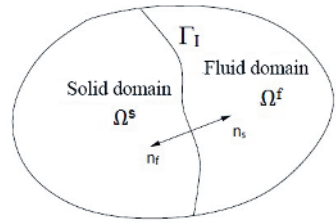


**Figure 1.14.** *Fluid–structure interface*

At the interface, the normal vectors are linked by the relation:

$$\mathrm{n} = \mathrm{n}_f = -\mathrm{n}_s \qquad [1.47]$$

From [1.37], [1.44], [1.46] and [1.47], we can deduce that:

$$(\sigma_f - \sigma_s)n = 0 \quad \text{on} \quad \Gamma_I(t). \qquad [1.48]$$