Basic phenomena

Moving free surface

The movement of the free surface should be considered whenever it is important to account for:

- transients (for example: pool-filling or draining);
- dispersion of scalar tracers/components by the free surface motion;
- hydrodynamic effects on immersed structures;
- formation of free-surface vortices due to the variation in space or in time of the water level (to the knowledge of the authors, no predictive formula exists);
- swell and wave-breaking; let H_h stands for the crest-to-trough wave height, L_h for the wave length, H for the water depth: the limit of wave-breaking for beaches with a gentle slope is $H_h/L_h < 0.14$ in deep water and $H_h/H < 0.8$ in shallow water; an approximate generalization of these two criteria is $H_h/L_h < 0.14 \tanh(2\pi H/L_h)$, see (Aelbrecht 2000).

The key non-dimensional numbers to define each case are:

- **Froude** number $F = U/(g\ H)^{1/2}$ with $F >> 1$ for supercritical flows and $F << 1$ for subcritical flows (the Froude number plays a similar role for free surface flows as the Mach number does in aerodynamics)
- **Reynolds** number $Re=UH/\nu$ and **Weber** number $We=\rho U^2H/\sigma$ (if $Re > 30\ 000$ and $We > 120$, the surface tension has no influence on the formation of free-surface vortices)

In *Code_Saturne*:

The Euler approach with a moving mesh (a.k.a. ALE) is generally well suited for large deformations of the free surface ("large" with respect to the cell size of mesh; for example: one may consider that a deformation is large if there are more than 20 cells for one wave length). Wave-breaking cannot be captured with this approach.

The displacement of the free-surface may be computed or specified by the user. If it is computed, the mass conservation may not be perfectly satisfied; this drawback is associated with the algorithm that is implemented to move the vertices and the faces of the mesh (and there is no Volume Of Fluid "VOF" method implemented in *Code_Saturne* for the time being).

Particle laden flows

The particles feature should be used if the following phenomena are to be studied:

- transport, deposition, re-entrainment
- chemistry (wall interaction)

In particular, it is important to decide if:

- the particles follow the fluid or if they should be tracked separately because of their own inertia or fluctuating motion (this is especially important to capture deposition phenomena)
- the influence of the particles on the flow must be accounted for

A non-dimensional number is important for this configuration:

- **The non-dimensional relaxation time-scale of the particles** $\tau_p^+ = (\rho_p/\rho_f)d_p^2u^{*2}/(18\nu_f^2)$ represents the time that is necessary for the velocity of a particle to adapt to that of the ambient flow (ρ_p and ρ_f represent the densities of the particles and of the fluid respectively, d_p the diameter of the particles, u* the wall friction velocity (roughly estimated as 5 or 10% of the mean bulk velocity) and ν_f the kinematic molecular viscosity of the fluid). One may consider that the particles follow the fluid if $\tau_p^+ << 1$. Moreover, if $\tau_p^+ < 0.1$ (generally for particles with a

diameter smaller that one micron) the deposition is affected by Brownian motion. For more details, one may refer to Peirano 2006.

In *Code_Saturne*:

If it is necessary to account for particles, at least two types of approaches may be considered:

- Eulerian: represent the particles as a passive scalar tracer, with a diffusivity to be defined, under the hypothesis that they follow the fluid and that they do not have any influence on the flow itself (that is the choice that has been made for the coal combustion module).
- Lagrangian: represent the particles with a statistical approach using a Lagrangian method. This method, more general, does not require the hypothesis that the particles follow the fluid. If it is necessary, the influence of the particles on the flow may be accounted for. The method is useful to study deposition and re-entrainment; however, it is important to underline that the physical phenomena are complex: for particles whose relaxation timescale τ_p^+ is lower than 1 or 0.1, the standard modelling available in *Code_Saturne* is not appropriate and more advanced and more specific models are required.

Incompressible, dilatable and compressible flows

By definition, a flow is “compressible” if the fluid density is variable; if not it is defined as “incompressible”[1]. For compressible flows, the pressure waves travel at a finite velocity and the Mach number (M=U/c) shows the relative importance of the fluid velocity to the compression wave velocity. In reality, the two types of flow are not clearly distinct since compressible flows start to behave like incompressible flows when the Mach number becomes small.

For Mach numbers M=U/c > 0.3, phenomena travelling at the speed of the sound begin to be important[2] and it becomes necessary to take into account the variations of the density due to pressure, temperature, species… The Boussinesq approximation (that takes into account the density variations only through the presence of the gravity force in the momentum equation) is not sufficient any more: the mass equation must contain the unsteady term (i.e.: $\partial\rho/\partial t + \mathrm{div}(\rho u) = 0$) so that acoustic waves are accounted for.

Only weakly compressible flows at M=U/c < 0.3 will be considered here and it will be assumed that the effects of the phenomena travelling at the speed of the sound are negligible (otherwise, *Code_Saturne* compressible flow module should be used). However, the flow may still be compressible, since the density is not necessarily uniform and constant. Several cases must be considered, depending on the amplitude of the relative variations of the density.

Before the different cases are presented, the Boussinesq approximation must be introduced. This linearized approach accounts for the density variations, when they are small enough, only through the gravity force that appears in the momentum equation. With this approach, the mass conservation reduces to a steady constraint on the velocity[3]: $\mathrm{div}(u) = 0$.

---

[1] « Dilatable » usually refers to a compressible flow for which the variations of the density are assumed to be due to variations of the composition or of the temperature (and not to variations of the pressure or of the velocity), and for which the possible phenomena associated with pressure waves (that are assumed to be infinitely fast) can be neglected.

[2] The characteristic value M = 0.3 is the usual limit. It comes from the following analysis, and ensures that the density variations remain “small enough” (see for example (Viollet 1997) or (Wilcox 1997)). One considers a fluid accelerating out of a pressurized reservoir: the total enthalpy conservation on a streamline (without volume force and for steady-state conditions) leads to defining the total enthalpy, from which the “total” temperature (or “reservoir temperature”) is deduced. For a perfect gas, the total temperature reads: $T_t = T\,(1+(\gamma-1)/2\ M^2)$. Under the hypothesis that the entropy does not change ($P/P_t = (T/T_t)^{\gamma/(\gamma-1)} = (\rho/\rho_t)^{\gamma}$), one obtains the “reservoir” pressure $P_t = P\,(1+(\gamma-1)/2\ M^2)^{\gamma/(\gamma-1)}$ and the corresponding density $\rho_t = \rho\,(1+(\gamma-1)/2\ M^2)^{1/(\gamma-1)}$. With these formulae, one can see that the variation of ρ remains lower than 2% for M < 0.3.

[3] This simplified mass conservation equation comes from dimensional analysis considerations, starting from the full mass conservation equation $\partial\rho/\partial t + \mathrm{div}(\rho u) = 0$, with ρ standing for the density and u for the fluid velocity. The equation can also be written as $(1/\rho)\,\partial\rho/\partial t + u\ \mathrm{grad}(\rho)/\rho + \mathrm{div}(u) = 0$. The characteristic length-scale L and time-scale δt depends on the physical configuration under consideration. With this notation, the magnitude of the three

It is usually considered that the Boussinesq approximation generally applies if[4] $\Delta\rho/\rho < 0.1$ (see for example: (LeQuéré 1992)). Indeed, (Paillère 2000) presents a configuration with $\Delta\rho/\rho = 0.01$ for which the approximation is valid and a configuration with $\Delta\rho/\rho = 0.6$ for which the approximation is not valid any more. Of course, this limit ($\Delta\rho/\rho < 0.1$) is not absolute: it merely indicates that the Boussinesq approximation is valid for "sufficiently small" density variations.

The cases that must be considered are the following, depending on the density variation magnitude:

- for small variations of the density, i.e. $\Delta\rho/\rho << 1$ (usually $\Delta\rho/\rho < 0.1$), the full mass conservation equation ($\partial\rho/\partial t + \mathrm{div}(\rho u) = 0$) may be used, but one may retain an approximation that is valid under some hypotheses:
    - for steady flows ($\partial\rho/\partial t=0$), the Boussinesq approximation may be used;
    - if the flow is not steady, one still may rely on the Boussinesq approximation (and use $\mathrm{div}(u) = 0$ as the mass equation) provided the time-scale associated with the variations of the density be the convective scale (U/L), which is precisely what is assumed here with $M < 0.3$. Since $\Delta\rho/\rho << 1$, one may also use the approximation $\mathrm{div}(\rho u) = 0$.
- For significant variations of the density (usually $\Delta\rho/\rho \geq 0.1$), the Boussinesq approximation is not valid any more:
    - for steady flows, ($\partial\rho/\partial t=0$), one may retain $\mathrm{div}(\rho u) = 0$ as the mass conservation equation;
    - if the flow is not steady, it is necessary to use the full mass conservation equation $\partial\rho/\partial t + \mathrm{div}(\rho u) = 0$.

As a conclusion, for $M < 0.3$, and under the hypothesis that the effects of the phenomena travelling at the speed of the sound are negligible (pressure waves, faster than the material waves), the mass conservation equation that may be used is as follows:

M < 0.3	Steady	Unsteady
$\Delta\rho/\rho < 0.1$	$\partial\rho/\partial t + \mathrm{div}(\rho u) = 0$ or $\mathrm{div}(\rho u) = 0$ or $\mathrm{div}(u) = 0$	$\partial\rho/\partial t + \mathrm{div}(\rho u) = 0$ or $\mathrm{div}(\rho u) = 0$ or $\mathrm{div}(u) = 0$
$\Delta\rho/\rho \geq 0.1$	$\partial\rho/\partial t + \mathrm{div}(\rho u) = 0$ or $\mathrm{div}(\rho u) = 0$	$\partial\rho/\partial t + \mathrm{div}(\rho u) = 0$

Table 1: mass conservation equation for M < 0.3

For more detail, (Gray 1976) provides an example for the derivation of the mass, momentum and temperature equations where Taylor expansions are used to write the physical properties (and in particular the density) as functions of temperature and pressure variations. The hypotheses required for the Boussinesq approximation to be valid, appear clearly in this systematic approach. For the specific cases considered in this paper (water and air at 1 atmosphere and 15°C), the authors obtain a domain of validity defined from the relative variations of the temperature and of the pressure, of the partial derivatives of the properties with respect to these two variables, of the Rayleigh and Prandtl numbers, and of characteristic length- and time-scales.

---

terms are respectively $(\Delta\rho/\rho)(1/\delta t)$, $(\Delta\rho/\rho)(u/L)$ and u/L. For $\Delta\rho/\rho << 1$, the second term is negligible compared to the third one; moreover, if δt is not much smaller that L/u (the time-scale characteristic of the material waves), the first term is also negligible, so that the equation reduces to $\mathrm{div}(u) = 0$.

[4] For the air, considered as a perfect gas at atmospheric pressure, the limit $\Delta\rho/\rho < 0.1$ characterizes a temperature variation $\Delta T/T < 0.1$ (with T in Kelvin), i.e. a variation of 30 K for T=300 K.

The non-dimensional numbers that must be evaluated in this configuration are the following:

- **Mach** number M = U/c (M > 0.3: compressible flow)
- **relative variation of the density** $\Delta\rho/\rho$
- **time-scale** characteristic of the flow U/L and of the boundary conditions

In *Code_Saturne*:

For M > 0.3 (compressible flows), specific numerical schemes and physical models are required. A numerical scheme for compressible flows is available in *Code_Saturne* but it has benefited from little feedback and its use requires some expertise (boundary conditions, thermodynamics for fluids other than perfect gases, multi-component mixture, …).

For M < 0.3 the standard scheme in *Code_Saturne* can be used if div(ρu) = 0 is a valid equation for mass conservation (Table 1), i.e. except for unsteady flows with $\Delta\rho/\rho \geq 0.1$ (in this case (for $\Delta\rho/\rho \geq 0.1$), a modification of the algorithm of *Code_Saturne* is necessary).

Natural/forced convection, laminar/turbulent flows

For M < 0.3, the reduced Froude number is $Fr = U/(g\,(\Delta\rho/\rho)\,H)^{1/2}$. Natural convection effects are negligible for Fr >> 1 (for example for Fr > 10); otherwise, the gravity force and the density variations must be accounted for (at least using the Boussinesq approximation, i.e. with the buoyancy term in the momentum equation).

If gravity has an influence (natural convection), one should evaluate:

- the gradient Richardson number that allows to determine if gravity effects inhibit turbulence ($R_i > 0.2$);
- the Rayleigh number that makes it possible to determine if the regime is laminar or turbulent; this is useful to *a priori* evaluate thermal fluxes (using correlations), for comparison to the computational results;

If gravity does not have any influence (forced convection), one should evaluate:

- the Reynolds number that allows us to determine if the regime is laminar or turbulent; this is useful to evaluate thermal fluxes *a priori* (using correlations, such as Colburn's) and head losses, for comparison to the computational results;

The non-dimensional numbers to evaluate in that case are the following:

- **Reduced Froude** number $Fr = U/(g\,(\Delta\rho/\rho)\,H)^{1/2}$
- **Gradient Richardson** number R_i
- **Rayleigh** number
- **Reynolds** number

In *Code_Saturne:*

When thermal phenomena are neglected or the flow is dominated by forced convection, it is advised to use:

- High Reynolds number mesh (i.e. a coarse wall grid resolution):
    - the k-epsilon with linear production as the default choice (ITURB= 21)
    - the SSG Reynolds Stress Model (ITURB= 31), whenever secondary motion or turbulent mixing is involved
- Low-Reynolds number mesh (i.e. a fine wall grid resolution):

- The v2f model as the default choice (ITURB=50)

The k-omega SST model (ITURB=60) may be used if it is not clear whether the mesh refinement at the wall makes the mesh suitable for a high Reynolds or a low Reynolds approach. It is not advised to use this model otherwise. If possible the v2f model should be used.

For mixed or natural convection phenomena, it is recommended that a sufficiently fine mesh at the wall should be used (usually possible when the Rayleigh number is small). If it is not possible to use such a fine (low Reynolds number) mesh, one may expect poor results for natural or mixed convection. In this case, there is no really best turbulence model but it is advised to use k-epsilon or compare the results for a variety of different turbulence models (eg k-epsilon with linear production (ITURB= 21), k-omega SST model (ITURB=60) and SSG Reynolds Stress Model (ITURB= 31)).

LES will be used only in cases where local and instantaneous data are required, or when such phenomena may influence the results that are looked for (for example: mixing by large structures that RANS models may not be able to capture). In such cases, the standard Smagorinsky model will be used by default: for complex configurations, the results produced with this model are most often as satisfying as those obtained with more advanced models. The LES model WALE may also be of interest (in *Code_Saturne* this model runs approximately twice as fast as the Smagorinsky model and does not generate turbulent viscosity in laminar flows which is a well known problem with the Smagorinsky model).

One may also define the roughness of the wall (dynamic and thermal roughness:in the GUI, key-word IPARUG).

For the “scalars” turbulent fluxes (tracers, concentration, temperature, enthalpy…), the only choice in version 2.2 and below of *Code_Saturne* is to use the high Reynolds number “SGDH” (single gradient diffusion hypothesis) model. In *Code_Saturne*, this model is used with wall-functions and the Boussinesq approach so that modelling of the turbulent fluxes is proportional to the gradient of the advected scalar (the coefficient of proportionality is the ratio of the turbulent viscosity to a turbulent Prandtl or Schmidt number).

This approach suffers limitations, especially for mixed/natural convection or for low Reynolds number flows (whatever the convection regime). The impact on the solutions is difficult to quantify *a priori*. For non-equilibrium anisotropic flow at high Reynolds numbers, the modelling approaches may encounter limitations; wall-functions assume that the boundary layer is in equilibrium and Boussinesq turbulence modelling supposes that the turbulence is isotropic. More models are available in *Code_Saturne* 3.0 (GGDH – general gradient diffusion hypothesis -, AFM – algebraic flux model –, DFM – differential flux model -, to be used in conjunction with the RSM) which take into account the anisotropic nature of the velocity and the scalar fields.

In all cases, the **user must refrain from varying the turbulent Prandtl and Schmidt numbers** with the configuration studied. Such an adjustment would be equivalent to a case-dependant modification of the diffusion/mixing predicted by the model: in practice, this would result in user-driven temperature or concentration results, denying to CFD computations their predictive potential.

Secondary motion

Identify the elements bound to create secondary motions (corner vortices, flow structures downstream a bend, vortices in the dead leg of a T-junction, cyclones separators…) and their potential impact (local exchange coefficient, perturbation of a stratified flow, source of thermal fluctuations, particle capture, tracer advection…). These elements should be borne in mind when choosing the turbulence model (see the previous paragraph).

Tracers or passive scalars

In *Code_Saturne*:

Nature of the scalar

One considers here the "scalars" that represent tracers in *Code_Saturne* (excluding temperature, enthalpy and the other scalar quantities that may be computed, such as the turbulent variables for example).

Code_Saturne solves a generic transport equation[5] for the scalar Y:

$$\partial(\rho Y)/\partial t + \mathrm{div}(\rho u Y) = \mathrm{div}(K \,\mathrm{grad}\, Y) + ST\_Y$$

Hence, the user must verify that this scalar equation is representative of the problem of interest *from a physical point of view*.

In the following, one considers the variable s that stands for the concentration of a tracer, in kg/m^3, (for example, s may represent the mass of salt dissolved in the water, per unit of volume of the solution). The equation that *must* be solved for s is the following[6] (obtained from considerations on the conservation of the mass of salt):

$$\partial(s)/\partial t + \mathrm{div}(us) = DST\_wor$$

However, this is not the equation that *Code_Saturne* solves. So s should not be chosen as a "scalar" in the sense of *Code_Saturne*.

A change of variable is required in order to retrieve the equation solved for "scalars" in *Code_Saturne*. If ρ stands for the density of the solution (here, salty water), a new variable s' is defined as:

$$s' = s/\rho$$

With this definition, s' is solution of the equation solved by *Code_Saturne* for "scalars":

$$\partial(\rho s')/\partial t + \mathrm{div}(\rho u s') = DST\_s',$$

The variable s', contrary to s, can be selected as a "scalar" in the sense of *Code_Saturne*.

The user may prefer to use a non-dimensional variable instead of s'. For example, with s'0 and a0 standing for constants coefficients, one may define X as:

$$X = s'/s'0 - a0$$

With this definition, X is solution of the equation solved by *Code_Saturne* for the "scalars":

$$\partial(\rho X)/\partial t + \mathrm{div}(\rho u X) = DST\_X,$$

The variable X, as s', can be selected as a "scalar" in the sense of *Code_Saturne*.

Remark 1: it is important to underline that the non-dimensional variable standing for the salt concentration and that may be selected as a "scalar" in the sense of *Code_Saturne* is *not* s/s0 – a0, but (with s'0 = s0/ρ0):

$$X = (s/\rho) / (s0/\rho0) - a0$$

Remark 2: the choice of the boundary conditions for the advected scalar must be done in accordance with the change of variable that defines the selected "scalar" in the sense of *Code_Saturne*. Usually, this does not lead to any difficulty for Dirichlet conditions (for example, the value imposed at the inlet would be defined as: X_inlet = (s_inlet/ρ_inlet) / (s0/ρ0) - a0). However, troublesome situations may arise for non-zero flux conditions, if the boundary condition of the density is not explicitly defined.

---

[5] ST_Y stands for the source terms that may appear for the scalar Y, excluding the molecular diffusion.

[6] DST_wor stands for the diffusion and potential source terms for s in this equation (for which the density of the fluid solution does not explicitly appear on the left-hand-side).

In conclusion, the choice of the "scalars" that can be considered as variables in *Code_Saturne* requires careful attention. For example, if s is the concentration of salt in kg/m^3 (mass of salt per unit of volume of solution) and ρ the density of the salted water in kg/m^3, the following variables can be selected as "scalars" in the sense of *Code_Saturne*:

- s' = s/ρ (mass of salt per unit of volume of solution / density of the solution, i.e. mass fraction of the salt)
- Cppm = s' * 1 000 000 (mass fraction of the salt in ppm – parts-per-million)
- X = (s/ρ)/(s0/ρ0) – a0 (non-dimensional mass fraction)

Value of the diffusivity

For a scalar Y, the equation implemented in *Code_Saturne* is, (as indicated above):

$$\partial(\rho Y)/\partial t + \mathrm{div}(\rho u Y) = \mathrm{div}(K \,\mathrm{grad}\, Y) + ST\_Y$$

However, the diffusion term may be expressed as a function of another variable C = A Y.

$$\partial(\rho Y)/\partial t + \mathrm{div}(\rho u Y) = \mathrm{div}(K_v \,\mathrm{grad}\, C) + ST\_Y$$

Under the hypothesis that the variations in space of A are negligible, a common approach is to define K from K_v as follows (the impact of this approximation is usually limited, in particular when the turbulent diffusion, which is modelled, prevails):

$$K = K_v A$$

For example:

- $K = \rho K_v$ for $C = \rho Y$ (with C a concentration s in kg/m^3 and Y the associated mass fraction s')
- $K = K_v / C_p$ for $C = Y / C_p$ (with C the temperature and Y the enthalpy for a perfect gas)

A more general approach may be envisaged, under the hypothesis[7] that C may be expressed under the form C(Y), one gets grad C = d (C)/dY grad Y, and it is possible to derive the expression for K that is necessary to complete the input data required by *Code_Saturne* to solve the equation on Y:

$$K = K_v \, d(C)/dY$$

For example:

- In a binary gaseous mixture with M_a and M_b the molar masses of the components "a" and "b" and with Y the mass fraction of the component "a", the volume fraction of the component "a" reads $C(Y) = (Y/M_a)/(Y/M_a+(1-Y)/M_b)$ and the diffusivity is $K = K_v (M_a M_b)/(Y M_b+(1-Y)M_a)^2$.
- In a salt water solution for which the density ρ depends only on the mass fraction of salt Y, the mass of salt per unit of volume of salted water reads $C(Y) = \rho(Y)Y$ and the diffusivity of the salt is $K = K_v (\rho + Y d\rho/dY)$.

Coupling with the conduction in solids (conjugate gradient with SYRTHES)

When the objective of the study is to determine a thermal load in a structure, an independent thermal study with boundary conditions representing the fluid thermal load with correlations may be sufficient. Equally the computation of the thermal field in the solid is not always compulsory and predefined correlations may be sufficient to provide boundary conditions for the fluid calculation. A fully coupled calculation between the fluid and the solid is required if one is interested in a transient load, local

[7] C may not be a function of Y exclusively: for example, for a concentration of salt, when the density also depends on the temperature T, C is a function of Y and T (C = ρ(Y,T)Y = C(Y,T)). In this case, the approach remains valid if it is possible to neglect the effects of the transport of mass by the diffusion created by the temperature gradient (otherwise, some terms depending on the gradient of the temperature should be added to the equation of the scalar Y).

characteristics, or if the geometry is too complex for reliable correlations to be available.

The coupling between fluid and solid computations must be taken into account only if a mutual influence is suspected. This will be the case especially if the solid is bound to create thermal bridges between fluid zones that (would otherwise not "see" each other and that) would not mix spontaneously (for example: the thermally conductive wall of a pipe containing a stably stratified flow)

- In that case, it is necessary to evaluate the time-scale of the conduction in the solid (L_s^2/λ_s, ratio of the square of a characteristic length of the solid to the thermal conductivity of the solid) and the time-scale characteristic of the fluid, i.e. the minimum between at least the characteristic convective time-scale U/L (velocity/characteristic length), a characteristic time-scale representative of the possible gravity effects over a height H, $(H/(g\Delta\rho/\rho))^{1/2}$, and a characteristic time-scale for turbulence, k/ε.
- These estimations will show if it is necessary to adopt a specific approach to implement a coupled modelling (for example, if the characteristic time-scale of the solid if several orders of magnitude larger than that of the fluid, an artificial acceleration of the convergence of the solid conduction may be employed). These calculations may even help determining if the coupling is simply relevant (it may be useless to couple a solid that has a very large thermal time-scale to a fluid subjected to very rapid variations about a permanent state).

The non-dimensional numbers to evaluate in that case are the following:

Ratio of the time-scales solid/fluid: (L_s^2/λ_s) / min $(U/L, (H/(g\Delta\rho/\rho))^{1/2}, k/\varepsilon)$

Steady / Unsteady flow

Three types of flows may be considered:

- statistically permanent flows: one will try and model a permanent state
- flows with an inherent unsteadiness: one will compute the time-scales of the different phenomena that may play a role. This makes it possible to determine whether some of them may be neglected (for example, if they are too slow with respect to the others)[8] or to determine the time necessary to reach a possible permanent state. Examples of time-scales:
  - convection: L/U
  - gravity: $(H/(g\Delta\rho/\rho))^{1/2}$
  - turbulence: k/ε
  - vortex shedding: L/ (U S), where S stands for the Strouhal number, i.e. the non-dimensional frequency of vortex shedding behind an obstacle of characteristic size L.
- Flows for which the unsteadiness is driven by the boundary conditions: one must compute the time-scales associated with the boundary conditions and compare them to the time-scales of the other phenomena. Of course, the analysis depends of the boundary conditions considered (explosion, valve closing**,** mass flow rate ramp…).

The (non-) dimensional numbers to evaluate in that case are:

- **Fluid time-scales:** $U/L, (H/(g\Delta\rho/\rho))^{1/2}, k/\varepsilon$
- **Strouhal** number

Domain of interest

The boundaries of the domain will be defined and their position justified. This may require using correlations that provide, for example, the minimal length for a specific type of flow development (pipe flow, jet, mixing layer…) so as to define the inlet or the outlet locations.

[8] A priori, this analysis is covered by the calculation of non-dimensional numbers such a as the Reynolds number, the Froude number, the Mach number, that compare the convection, diffusion, gravity and acoustic phenomena.

The source and sinks for the momentum (head loss, deviation…) and temperature (heat) will be defined and the precision with which they shall be dealt with will be specified.

The inlet and outlet boundary conditions will be defined and the uncertainties indicated (at least by providing a minimal value, a maximal value and a probable value). The upstream and downstream regions of the domain will be described as much as possible (singularities, head losses of the circuits…).

The surface condition will be indicated (smooth/rough), and the height of roughness ζ will be provided under a non-dimensional form $\zeta^+ = \zeta u^*/\nu$ ($\zeta^+ < 5$: smooth, $\zeta^+ > 70$: rough).

The non-dimensional numbers to evaluate in that case:

- **Non-dimensional size of the surface roughness**

Non-dimensional numbers

F	Froude	$U/(g\,H)^{1/2}$	inertia / gravity (or velocity / wave speed)	F > 1: supercritical flows, F < 1: subcritical flows The Froude number plays the same role for free surface flows as the Mach number does for aerodynamics
Fr	Reduced Froude	$U/(g\,(\Delta\rho/\rho)\,H)^{1/2}$	inertia / reduced gravity	Fr >> 1: differential gravity forces are negligible with respect to forced convection
Gr	Grashof	$g\,\beta\,\Delta T\,L^3/\nu^2$	reduced gravity / viscous effects	Equivalent to the square of a Reynolds number built on a natural convection velocity
M	Mach	U/c	inertia / wave propagation	M > 1: supersonic M < 1: subsonic $c^2 = (\partial P/\partial\rho)\|_s$ and for a perfect gas in particular $c^2 = \gamma\,P/\rho$ c is of the order of 300 m/s in air and of 1500 m/s in water (1 bar, 25°C)
Nu	Nusselt	$\Phi\,L/(\lambda\Delta T)$	non-dimensional thermal flux	Correlations such as Colburn, Mac-Adams (depending on Re, Pr, Ra)
Pr	Prandtl	ν/a	viscous effects / conduction effects	0.6 to 1: gas 1 to 20: usual liquids 1000 to 10 000: oils 0.005 to 0.05: liquid metals
Ra	Rayleigh	$g\,\beta\,\Delta T\,L^3/(\nu a)$	Gr Pr	Characteristic of the natural convection regime (laminar below 10^5 for example)
Re	Reynolds	UL/ν	inertia / viscous effects	Re > 5000: turbulent flow (this limit may be lower for specific types of flow)
R_i	Gradient Richardson	$\beta g(\partial T/\partial z)/(2s{:}s)$	gravity / turbulence	R_i > 0,2: turbulence inhibited pure shear: $2s{:}s = \frac{1}{2}(\partial U/\partial y)^2$ pure impact: $2s{:}s = 2(\partial U/\partial x)^2$
Sc	Schmidt	ν/D	viscous effects / diffusive effects	Equivalent of the Prandtl number for the diffusivity of the species
S	Strouhal	$f\,L/U$	non-dimensional frequency	-
We	Weber	$\rho U^2 L/\sigma$	inertia / surface tension	We >> 1: surface tension has no influence

References

(Aelbrecht 2000) Cours sur la houle, Ecole Nationale des Ponts et Chaussées, D. Aelbrecht, Note EDF R&D HP-72/2000/030/A, 2000

(Gray 1976) The Validity of the Boussinesq Approximation for Liquids and Gases, Donald D. Gray and Aldo Giorgini, International Journal of Heat and Mass Transfer, 19, 545-551, 1976

(LeQuéré 1992) A Chebyshev Collocation Algorithm for 2D Non- Boussinesq Convection, Le Quéré P., Masson R., Perrot P. Journal of Computational Physics 103, 320-335, 1992

(Paillère 2000) Comparison of low Mach number models for natural convection problems, Paillère H., Viozat C., Kumbaro A., Toumi I., Heat and Mass Transfer, 36, 567-573, 2000

(Peirano 2006) Mean-field/PDF numerical approach for polydispersed turbulent two-phase flows, Peirano E., Chibbaro S., Pozorski J. et Minier J.P., Progress in Energy and Combustion Sciences, 32(3): 315-371, 2006

(Viollet 1997) Mécanique des fluides à masse volumique variable, Viollet P.L., Presses des Ponts et Chaussées, 1997

(Wilcox 1997) Basic Fluid Mechanics, Wilcox D.C., DWC Industries, 1997 (Roache 1997) Quantification of Uncertainty in Computational Fluid Dynamics", Annual Rev. Fluid Mech., Vol. 29, pp.123-160, Roache, P.J., 1997