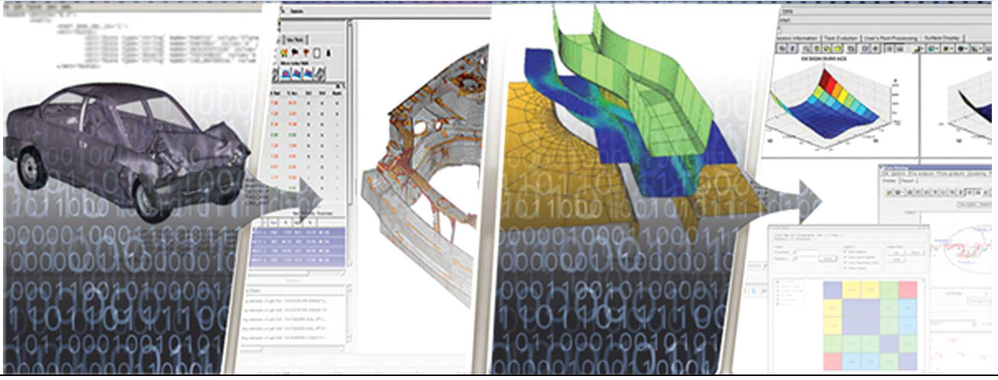

RADIOSS THEORY MANUAL

Version 2017 – January 2017

CFD

Altair

Altair Engineering, Inc., World Headquarters: 1820 E. Big Beaver Rd., Troy MI 48083-2031 USA
Phone: +1.248.614.2400 • Fax: +1.248.614.2411 • www.altair.com • info@altair.com

CONTENTS

Chapter

2

COMPUTATIONAL AERO-ACOUSTIC

2.0 COMPUTATIONAL AERO-ACOUSTIC

This chapter presents a resumed state-of-the-art of simulation in aero-acoustic domain as presented in [88]. Aero-Acoustics is the engineering field dealing with noise generated generally (but not necessarily) by a turbulent fluid flow interacting with a vibrating structure. This field differs from the pure acoustic domain where the object is the propagation of acoustic pressure waves, including reflections, diffractions and absorptions, in a medium at rest. Aero-Acoustic questions arise in many industrial design problems and are heavily represented in the noise nuisances related to the transportation industry.

A classification of Aero-Acoustic problems can be made using the following three categories:

- **External wind noise transmitted to the inside through a structure:** In the automotive industry, a pillar, side mirror and windshield wipers noise are typical problems of this category.
- **Internal flow noise transmitted to the outside through a structure**: Examples of this class of problems are exhaust, HVAC and Intakes noises.
- **Rotating machine noise:** Axial and centrifugal fans are noisy components that bring with them many interesting Aero-Acoustic problems.

Most of the Aero-Acoustic R&D works are performed experimentally but this method has some critical pitfalls. Although it is relatively simple to setup a microphone, measure a noise level and derive a spectrum at any given location in space, the correct analysis of an Aero Acoustic problem involves the use of advanced experimental techniques and is complex to use. The Aero Acoustic engineering community seeks more and more the help of CAE tools as they become available. Those tools complement the experimentations and allow a thorough visualization and understanding of the pressure and velocity fields as well as the structural vibrations. Furthermore, parametric studies can be carried out with little added cost since a numerical model modification is often straightforward and the CPU time is becoming cheaper and cheaper.

CFD codes are available since over several years, able to predict with a reasonable precision steady state flows (drag and lift) and slow transient flows like heating and defrosting. Highly transient flows involved in the Aero-Acoustic phenomena have not been treated since they were not in the bulk of the needs and they required way too much CPU to be industrially feasible. Acoustic Propagation numerical tools have also been industrially available since quite a few years. These tools operate in the frequency domain and are able to propagate a given boundary condition signal in a fluid at rest, including the noise reflections, diffractions, transmissions and attenuations thanks to the various geometrical obstacles and different materials.

Attempts have been made to combine existing CFD and Acoustic propagation tools to predict Aero-Acoustic problems. Most methodologies are based on the Lighthill and Curle method, developed in the mid 50’s and Ffowcs Williams and Hawkings contributions made in the late 60’s [67], [68], [69], and [70]. The ideas underlying these methods are to decouple the flow pressure field and the acoustic pressure field. The fluid flow can then be computed by a standard CFD code and the noise derived from the curvature and turbulent intensities of the flow. A propagation tool is then used to compute the noise on a sub grid of the CFD computational domain loosing therefore quite some local information and high frequency content. First attempts were made with incompressible steady state CFD simulations and were not able to deliver valuable result in many cases. A good example of these limitations is highlighted by the study of the noise generated by a simple ‘side mirror’ shape written by R. Siegert [71]. Recent developments of this family of techniques require the use of transient simulations and filtering to avoid loosing to much information on the coarser acoustic mesh. Reasonable success has been met in specific areas involving low frequencies (up to a couple hundred Hz) and considerable CPU time is needed.

An alternative methodology is to incorporate in a single numerical tool, right from the beginning, the ingredients that are necessary to perform direct Aero-Acoustic numerical simulation. They are:

- **Compressible Navier Stokes:** To be able to propagate pressure waves and therefore take into account in a single simulation the flow and the noise, including all possible cavity modes.
- **Fluid structure coupling:** To be able to treat the problems involving a turbulent flow on one side of the structure and the noise radiation on the other side.
- **Small time step:** To be able to deal accurately with frequencies going up to several thousand Hertz.
- **Transient turbulence modeling:** Unlike the Reynolds Averaged Navier Stokes (RANS) methods that makes the assumption that the flow is a combination of a steady state and turbulent fluctuations. Aero-Acoustic noise is directly linked to the small scale turbulence fluctuations and strongly time dependant.
- **Acoustic boundaries with prescribed impedance:** This is a critical point of a good Aero-Acoustic simulation. Boundaries need to be able to perform tasks such as giving a free field impedance to an inlet with fixed velocity, prescribing a specific impedance at the outlet of a duct to make sure long wavelength stay trapped inside, treat exterior air impedance effect on a vibrating structure and be used to model absorbing materials (carpet, foams …) that are used to coat many components.

These ingredients have been implemented in a single numerical code. The outcome is RADIOSS solver which is different from the existing CFD codes in its capabilities and particularly well suited to short time transient analysis.

2.1 Mathematical Formulation

The objective of the development described in this document is underlined by the search for fully suited numerical technologies in order to model all physical phenomena involved in noise source generation. This criterion will be used to perform all the major numerical choices described below without any trade-off tight to other applications.

2.1.1 Compressible Navier Stokes

Correct prediction of Aero-Acoustic phenomena must obviously include a solution of the 3D Navier Stokes equations since noise sources lie in a turbulent viscous fluid flow. The equations have been written with the Arbitrary Lagrange Euler (ALE) formulation [8]. This means that an arbitrarily moving frame is used in order to be able to have a fluid grid that can undergo deformation. The deformation can range from small vibrations (an exhaust pipe) to large deformations (a door slamming). The conservation equations are written for mass, momentum and energy:

Mass $$\frac{\partial \rho}{\partial t} + \left((\mathbf{u} - \mathbf{w}).\nabla \right)\rho + \rho \nabla.\mathbf{u} = 0 \qquad \text{EQ. 2.1.1.1}$$

Momentum $$\rho \frac{\partial \mathbf{u}}{\partial t} + \rho\left((\mathbf{u} - \mathbf{w}).\nabla \right)\mathbf{u} - \nabla \tau + \nabla p = 0 \qquad \text{EQ. 2.1.1.2}$$

Energy $$\frac{\partial \rho e}{\partial t} + \left((\mathbf{u} - \mathbf{w}).\nabla \right)\rho e + (\rho e + p)\nabla.\mathbf{u} = 0 \qquad \text{EQ. 2.1.1.3}$$

Where **u** is the fluid velocity, **w** the grid velocity, ρ the density, *p* the pressure, *e* the energy and τ the stress tensor.

Note: In case of **w=u,** the system degenerates into the Lagrangian formulation, meanwhile **w=0** describes the classical Eulerian formulation. Lagrangian formulation will be used for the structures when needed. An ALE fluid mesh is attached to the structure mesh, therefore able to undergo deformations tight to the structural vibrations.

Acoustic phenomena are driven by pressure waves propagating in the materials. Those waves velocity, the sound speed velocity *c* can be expressed by the following equations:

$$c = \sqrt{\frac{\partial P}{\partial \rho}} \qquad \text{EQ. 2.1.1.4}$$

Another useful relation defining *c* as a function of the material characteristics is:

$$c = \sqrt{\frac{K}{\rho}} \qquad \text{EQ. 2.1.1.5}$$

where *K* is the bulk modulus of the material.

In the usual engineering fields, the sound speed is about 340 m/s for air and 1500 m/s for water.

Most of the CAA research in the industry is aimed at flow motions of relatively small Mach numbers.

Therefore, CFD codes have widely used the incompressibility hypothesis which allows a fair simplification of the Navier Stokes equations. An *incompressible flow* is a flow for in which:

$$\left|\frac{D\rho}{Dt}\right| = \left|\frac{\partial \rho}{\partial t} + \mathbf{u}.\nabla\rho\right| << \rho\left(\left|\frac{\partial u}{\partial x}\right| + \left|\frac{\partial v}{\partial y}\right| + \left|\frac{\partial w}{\partial z}\right|\right) \qquad \text{EQ. 2.1.1.6}$$

If this condition is fulfilled the mass conservation equation can be rewritten.

$$\nabla \mathbf{u} = 0 \qquad \text{EQ. 2.1.1.7}$$

It has to be noted that an incompressible flow does not necessarily implies that ρ remains constant throughout the flow but that ρ remains constant along a streamline since one can substitute EQ. 2.1.1.6 in EQ. 2.1.1.1. Practically, this condition can be reasonably assumed for Mach numbers lower than 0.3 and in the case of transient flows, if the time rate of velocity change is long compared to the time for a sound wave to traverse the flow field. Thus, the incompressibility hypothesis yields an instantaneous transmission of pressure waves in the fluid domain. Incompressible approach will not be able to simulate the propagation of acoustic waves nor the cavity modes tight to composition of reflected waves on a given geometry. It then becomes necessary to apply one of the following techniques:

- Solve the compressible Navier Stokes equations to capture correctly the propagative nature of the noise waves.
- Use additional equations to derive the noise sources from the flow field.

Accordingly to the base criteria of always to the best solution for CAA, it has been decided to follow the first path that requires no specific assumptions and is physically and mathematically much better grounded. By solving the fully compressible Navier Stokes equations, it becomes possible to compute in a *single simulation* the flow and the sound pressure wave's propagation. Since there is a large gap between the flow pressure variations (in the order of hundreds of Pa in most cases) and the acoustic pressure variations (in the order of 1 Pa ~94 dB), it has become critical to use 64-bits double precision arithmetic. Using 32-bits would yield flawed results because the acoustic pressure levels would not be handled properly in many cases.

The spatial integration of momentum equation is performed with a Finite Element integration using *Streamline Upwind Petrov Galerkin* (SUPG) scheme [72], which was shown accurate enough to capture flow instabilities. Advection of state variables is achieved via a simple finite volume technique as in Donea's original paper [8].

2.1.2 Transient analysis, explicit formulation

Sound pressure waves and acoustic phenomena are transient per fundamental nature. Therefore, a transient solution of the full compressible Navier-Stokes equations has been developed, allowing the propagation of pressure waves as well as transient fluid flow simulation.

For time integration, two techniques are available, implicit and explicit. Implicit methods are unconditionally stable; therefore allowing the use of an arbitrary large time step; meanwhile the explicit formulation is conditionally stable. A Courant Friedrichs Levy condition [56] has to be verified at each time step of the simulation:

$$dt \leq Min\left(\frac{l}{\left(c+\|\mathbf{u}-\mathbf{w}\|\right)}\right) \qquad \text{EQ. 2.1.1.8}$$

where l is the characteristic element size, c the sound speed, **u** the flow velocity and **w** the grid velocity. Typical values of dt for CAA applications are in 1 to 5 μ 's range.

The choice of one formulation or the other will actually be dictated by the minimization of the CPU time for the considered application. Shall the goal be to simulate the defrosting of a windshield lasting about 5 minutes, with low flow velocities, the several second time step allowed by the implicit scheme will be much more efficient. In the CAA case however, the frequencies of interest are going up to 5 to 10 kHz and the Nyquist criteria and anti aliasing filtering imposes a sampling for the time domain recording of 20 to 40 kHz. Therefore, the maximum time step that can be used in implicit will be $5x10^{-5}$s. The explicit time step being in the range of $2x10^{-6}$s, one has to compare the cost of one implicit time step to 25 explicit time steps. A survey yield a ratio of about two orders of magnitude between the CPU cost of both schemes in favor of the explicit and therefore, the explicit scheme CPU efficiency has been considered superior for CAA applications.

2.1.3 Large Eddy Simulation Turbulence modeling

Another critical aspect of a correct CAA modeling is to take into account properly the noise induced by turbulent structures. Unfortunately, the turbulent structures that are simultaneously active at any given time range from the full size of the problem to the microscopic Kolmogorov size. The ideal solution would be to use DNS but this is unfortunately out of reach of today computers. Consequently, a turbulence model has to be used. The choice among the turbulence models will be performed by evaluating their interest for the CAA simulation. Today, there are two major families of turbulence models that are available for implementation in an industrial oriented CFD code.

The Reynolds Average Navier-Stokes (RANS) turbulence models family relies on the assumption that the flow can be separated between a steady state flow and a turbulent fluctuation. The steady state flow can be interpreted as the spatial average of the flow field. Transient problems featuring vortex shedding cannot be treated this way and therefore Unsteady Reynolds Average Navier-Stokes (URANS) have been developed in which the averaging is performed on a considerably smaller time scale. Although RANS and URANS models are easy and cheap to implement, the basic assumption that there is a combination of a steady state flow and turbulent fluctuations is not very well suited to the CAA analysis where the key to the noise generation is the transient turbulent behavior. The Large Eddy Simulation or LES family is based on a fully transient flow simulation. Large turbulent scales which depend heavily on the boundary conditions are solved directly by the computational grid and the small scales are assumed to be more or less non problem dependant, are solved by a model called a Sub Grid Scale model (SGS). The Navier-Stokes equations are filtered in space and the spatial high frequency terms (beyond the filter cutoff frequency) are modeled by the SGS. The SGS main role is to model the viscous dissipation of the energy within the small scale eddies and absorbs energy from the large scales to this effect. The filter used most often is actually the grid itself and the spatial cutoff is about 6 element size. The drawback that prevents the LES to be used in many cases is the need for small elements for the base assumption to be valid (independence of the modeled scales from the boundary conditions) and the transient assumption which requires the use of a CFL condition on the time step to guarantee a correct simulation of the larger scales and yields large CPU time when combined with classical implicit resolution techniques.

In our case, since it has already been chosen to solve the transient Navier Stokes equations with an explicit scheme yielding a very small time step, the CFL condition does not imply any major constraint. Mesh size criteria have been developed in order to ensure a good CAA behavior and LES accuracy (also see Modeling Methodology in the next chapter).

In noise generation zones where most of the turbulence is generated and small vortices are very active, the mesh criteria is:

$$h < 0.1\left(\frac{u}{f}\right) \qquad \text{EQ. 2.1.1.9}$$

Where *h* is the element size, *u* the flow velocity and *f* the highest frequency of interest. The last equation yields the expression of local Reynolds number:

$$\mathrm{Re} = \frac{u.h}{\nu} < 0.1\left(\frac{u^2}{f.\nu}\right) \qquad \text{EQ. 2.1.1.10}$$

Usually, u is in the 20 m/s range, f is 2500 Hz and the air viscosity ν =1.5x10^{-5}m^2/s. This yields a local Reynolds number of 1000. A reinforced mesh criteria applied close to the walls is to have a y_+ lower than 100. Although, these numbers might be considered as large by academic standards, it has been verified for various industrial applications that the numerical predictions reasonably match the experimental data [73]. In the SGS model, the sub grid scale level energy dissipation ε_{SGS} is proportional to the resolved scales stress tensor S_{ij} and the modeled small scales stress tensor τ_{ij} through a turbulent viscosity coefficient ν_{SGS} . This turbulent viscosity is a function of the filter width l (mesh size) and of the resolved scales (via the stress tensor). C_s is known as the Smagorinsky constant and has been experimentally evaluated to be 0.18 (this constant can be tuned for specific applications and 0.1 is used by default in RADIOSS).

$$\varepsilon_{SGS} = -2\nu_{SGS}.\overline{S}_{ij}.\overline{S}_{ij} \qquad \text{EQ. 2.1.1.11}$$

$$\nu_{SGS} = (C_S l)^2 \left(\left|\overline{S}\right|\right) \qquad \text{EQ. 2.1.1.12}$$

It has to be noted that an original pressure damping factor has been developed in order to better take into account the absorption of the acoustic waves on the boundary layers.

The elements neighboring the walls have a specific treatment depending upon the element size. A logarithmic velocity profile is assumed in the first layer of elements and the corresponding modified viscosity is derived.

2.1.4 Boundary Conditions

Non-reflective boundary conditions are critical for CFD simulations and it is obviously going to be even more critical for CAA analysis. Acoustic boundary conditions treatment can be classified in three different categories:

- Non-reflective fluid boundaries in the open field. Under this class lie the external wind noise problems. Not only outlet and sides need to be treated but unless and infinite impedance or supersonic conditions are assumed, the inlet shall be able to let acoustic waves go out of the computational domain.
- Boundaries including geometrical details. Exhaust tailpipe for instance have a cutoff frequency related to its diameter and shape.
- Exterior air boundaries. In order to correctly treat problems like an exhaust noise, the exterior air impedance has to be applied to the structure in order to get the correct vibrations and noise. It has to be noted that in the case of a duct surrounded by other structures (for example, an exhaust line under a car), the acoustic impedance of the exterior air is vastly different from the free field conditions and non trivial to compute.

The goal of boundary conditions is to replace the influence of the exterior of the computational domain by a condition that will let some frequencies go out and reflect others accordingly to the physical properties of the surrounding materials. Acoustic impedance, defined by the ratio of the acoustic pressure $\tilde{p}$ at a given location over the acoustic velocity u_n at the same location is a key notion.

$$Z = \frac{\tilde{p}}{u_n} \qquad \text{EQ. 2.1.1.13}$$

The important point to be reminded here is that $\tilde{p}$ and u_n differs from the traditional fluid dynamics pressure and velocity since they are complex functions, solutions of Helmoltz equation which is describing the propagation of waves in the frequency domain ω:

$$\Delta\tilde{p}(\vec{r},\omega) + k^2\tilde{p}(\vec{r},\omega) = F(\vec{r},\omega) \qquad \text{EQ. 2.1.1.14}$$

With:

$$k = \frac{\omega}{c} \qquad \text{EQ. 2.1.1.15}$$

where *c* is the sound speed. In the acoustic works, items are classically described as a function of the frequency. In particular, impedances describing the boundary conditions acoustic behavior. RADIOSS on the other hand is based on a time domain description of the physical phenomena, it is therefore not possible to use directly those data and it is required to impose time dependant boundary conditions. For the non-reflective boundary conditions, it has been decided to implement the linearized Euler equation by Bayliss and Turkell [74]:

$$\frac{\partial\rho}{\partial t} = \rho c\frac{\partial u_n}{\partial t} + \frac{c}{2l_c}\left(p_\infty - p\right) \qquad \text{EQ. 2.1.1.16}$$

Where *n* is the normal to the boundary surface and l_c a relaxation factor toward the desired pressure p_∞. Practically, the last equation matches the radiation of a monopole situated at a distance 2 l_c inside the computational domain.

2.1.5 Fluid Structure Coupling

Fluid Structure Interaction (FSI) is an issue in many important sub domains of the CAA. Important applications include and are of course not limited to:

- Duct noise: The noise radiated by a structural system of ducts whose vibrating walls are excited by a turbulent flow inside. Among the many examples, exhausts and Heat Ventilation and Air Conditioning (HVAC).

- Passenger perception of exterior wind noise transmitted by a side glass or plane cabin.

One might suggest that rather than using a complex FSI solution, a decoupled approach could be used. That is performing a CFD (or pure fluid CAA) simulation on one hand and then apply to a structural model as a boundary condition the computed pressure field. Beyond the mere inconvenience tight to the external coupling of two numerical programs (flow and structure), this approach, which is reasonably grounded steady aero elasticity phenomena and for low frequencies applications is not well suited to broad band CAA applications. It is not practical to apply the transient pressure field with a sampling of 20 kHz on a fine mesh and even more important, there is a dependency of the structure vibrations to the surrounding fluid impedances. This impedance can be eventually modeled on the side where there is no flow but is extremely complex on the side exposed to the turbulent airflow. Consequently, in most cases, a decoupled modeling will be flawed from the beginning.

Accordingly to our goal of modeling all physical phenomena important for CAA, a full FSI has been developed between the well-established structural module of RADIOSS and the CFD/CAA module. Some specific options have been developed in order to make this interface practical among which:

1. ALE Subcycling. In most applications, there are 95% fluid elements and 5% or less structural elements. Unfortunately the sound speed is often one order of magnitude higher in the structure yielding a time step decreased by the same factor. The 5% structure element will impose to the 95% a time step 10 times smaller, which is obviously not desirable. A specific subcycling has been developed in order to compute the fluid elements with a decoupled time step from the structures.
2. Fluid – Structure interfaces have been developed to ease the mesh development. Those interfaces can be used to stitch arbitrary meshes together and can be used in either fixed or sliding mode (fluid-fluid fixed and sliding interfaces have also been developed to take care primarily of fan problems).

2.2 Modeling Methodology

The process of defining a numerical model for a given CAA application unfolds in four consecutive phases. The quality of the results is directly tight to the quality of the numerical model developed in the points 1 and 2 below. It is therefore critical to define precisely the questions the model is supposed to answer to before starting any development:

1. Mesh definition: Targeting specific answers to engineering questions and constrained by the available computer resources.
2. Numerical model construction, including material, boundary conditions and desired output.
3. Run monitoring.
4. Post-processing of the time domain data. Including 3D visualization, time history and frequency content analysis.

2.2.1 Mesh definition

The mesh building process is the art of building models able to solve the CFD and the CAA problem for a given set of boundary conditions and range of frequencies while keeping the model as small as possible to minimize the compute time. To do so, a set of practical rules have been developed. These rules should be used as initial guidelines to get started on a given problem.

However each class of problems has its own requirements and subtleties and a good knowledge of the problem physics through experimental data and/or numerical simulations will be necessary to refine these rules and get the best possible results.

2.2.2 Post-processing

Post-processing of the numerical simulation is very similar to the post-processing of a detailed experimental study of the same problem. The analysis will be carried out by using:

- FFT’s of recorded time domain signal to access the frequency domain content at any given location of the computational domain.
- Visualization and analysis of intensities on the structures
- Propagation in the far field (if and when needed) of the pressure signal. Typically, this is required for simulations where the measurement locations are not located in the computational domain. In most of the internal flow problems for instance the limit of the domain is the structure and a boundary elements layer to represent the outside air impedance (exhaust, HVAC …). The noise is often measured at a given distance outside the ducts in still air where there is no reason to have an expensive CAA solution. This propagation can be performed by a simple monopolar approximation that gives satisfactory results in the free conditions or by more sophisticated tools such as BEM methods.
- …