CHALMERS
UNIVERSITY OF TECHNOLOGY

Mathematical modelling and solutions to Flow Acoustical problems

Master's Thesis in the Master's programme in Sound and Vibration

Johan Gustafsson

Department of Civil and Environmental Engineering
Division of Applied Acoustics
Vibroacoustics Group
CHALMERS UNIVERSITY OF TECHNOLOGY
Gothenburg, Sweden 2016

CHALMERS UNIVERSITY OF TECHNOLOGY
Göteborg, Sweden 2016
Master’s Thesis BOMX02-16-136

MASTER'S THESIS BOMX02-16-136

Mathematical modelling and solutions to Flow Acoustical problems

Master's Thesis in the Master's programme in Sound and Vibration

Johan Gustafsson

Department of Civil and Environmental Engineering
Division of Applied Acoustics
Vibroacoustics Group
CHALMERS UNIVERSITY OF TECHNOLOGY

Göteborg, Sweden 2016

Mathematical modelling and solutions to Flow Acoustical problems
Master's Thesis in the Master's programme in Sound and Vibration

Master's Thesis BOMX02-16-136
Department of Civil and Environmental Engineering
Division of Applied Acoustics
Vibroacoustics Group
Chalmers University of Technology
SE-412 96 Göteborg
Sweden
Telephone: + 46 (0)31-772 1000

Chalmers Reproservice/ Department of Civil and Environmental Engineering
Göteborg, Sweden 2016

Mathematical modelling and solutions to Flow Acoustical problemsMaster's Thesis in the Master's programme in Sound and Vibration

Johan Gustafsson
Department of Civil and Environmental Engineering
Division of Applied Acoustics
Vibroacoustics Group
Chalmers University of Technology

ABSTRACT

The goal of this thesis is to create a guide containing explanations and solutions to flow acoustic problems, which can be used by engineers who are not familiar to this specific area.

Since the subject of "flow acoustics" is very wide, this thesis concentrates on solutions to a number of specific case studies, while also explaining the underlying theories. The thesis mainly focuses on internally generated sound, such as in ducts.

The thesis also explains and gives walk through guides to some simulation tools that can be used for visualizing flow acoustic phenomena. The different theoretical models that are used to make the simulations are explained and compared so that the right model can be chosen for each specific application.

Key words: Flow acoustics, CFD

Matematisk modellering och lösningar till flödesakustiska problem

Examensarbete inom Ljud och Vibrationer
JOHAN GUSTAFSSON
Institutionen för bygg- och miljöteknik
Avdelningen för Teknisk Akustik
Gruppen för Vibroakustik
Chalmers tekniska högskola

SAMMANFATTNING

Målet med detta examensarbete att skapa en instruktion innehållande förklaringar och lösningar till flödesakustiska problem som kan användas av ingenjörer som inte är förtrogna med det här specifika området.

Eftersom ämnet ”Flödesakustik” är förhållandevis brett, fokuserar det här examensarbetet på ett antal praktikfall, samtidigt som det förklarar grundläggande principer. Examensarbetet fokuserar till största delen på ljud som alstras internt, som till exempel i rör.

Examensarbetet förklarar dessutom och innehåller guider till några simuleringsverktyg som kan användas för att visualisera flödesakustiska fenomen. De olika teoretiska modellerna som används för att göra simuleringarna förklaras och jämförs med varandra så att rätt modell kan väljas för varje enskild applikation.

Nyckelord: Flödesakustik, CFD

Contents

Preface

The purpose of this master thesis is to create an instruction-sheet on how to simulate sound and how to decrease it, in some flow acoustical problems that ÅF-Ingemansson encounter in their jobs as acoustic consultants. This sheet of information can then be used by consultants not familiar to the specific area. The sheet will try to explain the physical phenomena's behind the sound generation and also give examples on what countermeasures to use for each specific problem. The actual problems handled in this master thesis will mostly be sound generation in pipes, but also sound generation caused by moving objects.

Göteborg October 2016

Johan Gustafsson

Notations

c_0	speed of sound
d	diameter
E	Young's modulus
f	frequency
$f_{V,i}$	Unsteady force acting on the fluid
M	Mach number
p'	Acoustic pressure
r	Distance between source and listener
U	mean flow velocity
u_i	Flow velocity in the flow direction
u_j	Flow velocity perpendicular to the flow direction
V	Volume
$\overline{W}$	sound Power
μ	Coefficient of viscosity
η	Dynamic viscosity
ν	Poisson's ratio
ρ	density
ρ'	Disturbance in density

1 Theory

In the development of commercial jet powered aircrafts during the 1950´s, it became apparent that a theory to handle sound generation by fluid processes was needed. As a result, James Lighthill developed a model for aero acoustic sound production. Lighthill connected fluid mechanics to acoustics by rearranging fluid flow equations into an inhomogeneous wave equation. (Åbom 2008)In the first section, 1.1, the basic theory of computational fluid dynamics will be discussed. In the second section, 1.2, the connection between the laws of fluid dynamics and the classical wave equation is established. The third section explains Lighthill's acoustic analogy, i.e. the connection between fluid dynamics and acoustics. In the fourth section, a few additional aeroacoustic models are described. Finally, in section 1.5, the theory behind sound propagation in ducts is covered.

1.1 Computational Fluid Dynamics (CFD)

Computational Fluid Dynamics (CFD) is a branch of fluid dynamics that uses numerical methods and algorithms. CFD is based on Navier-Stokes equations:

Conservation of mass: $\frac{\partial p}{\partial t} + \nabla \cdot (\rho u) = m$ (1)

Conservation of momentum: $\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} + f_{V,i}$ (2)

The partial differential equations that describe the flow are usually non-linear and are therefore solved numerically instead of analytically (Tritton 1988).

1.1.1 Turbulence modelling

Turbulent flow can be modelled using different methods. No model is considered to be the best option for all types of problems. In the subsections below, a short introduction to the methods that will be used in this thesis is given.

1.1.1.1 Direct Numerical Simulation (DNS)

When using the direct numerical simulation approach, Navier-Stokes equations are solved numerically without any simplifications. This means that the computational time is very high and the method is therefore often impractical to use. The model is mostly used in research to better understand the physics behind turbulence or to benchmark other turbulence models (Fluent Inc. 2005).

1.1.1.2 Reynolds-averaged Navier-Stokes (RANS)

In order to reduce the computational time the Reynolds-averaged Navier Stokes approach is often used to model flow. The reduction of computational time is possible due to the simplification that the flow quantities, such as flow velocity and pressure, are divided into time averaged and fluctuating quantities. RANS approach is often used when the flow is time dependent (Fluent Inc. 2005).

1.1.1.3 K-epsilon (k-ε) turbulence model

The k-ε model is the most commonly used turbulence model in CFD. It is based on the transport equations for kinetic energy (k), and the dissipation rate (ε). While the transportation equation for the kinetic energy is derived from the exact equation, the transportation equation for the dissipation rate is based on empirical data (Fluent Inc. 2005).

1.1.1.4 Large Eddy Simulation (LES)

While the RANS approach uses time averaging to reduce computational time, the LES approach uses filtering. The LES model uses the exact Navier-Stokes equations, but removes eddies smaller than the size defined by the filter.

Since the scale of the flow is reduced close to walls, the resolution needs to be increased in these areas. Since this requires a lot of computational time, wall functions are often used (Fluent Inc. 2005).

1.1.1.5 Detached Eddy Simulation (DES)

The Detached Eddy Simulation model combines the models for RANS and LES. The RANS model is used in the boundary layer, while the LES model is used further away from the wall. The DES is suitable for high Re-number, external flows, i.e. when the inertia forces dominate the viscous forces in the flow (Fluent Inc. 2005).

1.1.1.6 Spalart-Allmaras model

The Spalart-Allmaras model solves the transport equation for kinematic eddy viscosity. The model is a so called one-equation turbulence model. This means that only one transport equation is used to model the turbulent properties, compared to two-equation turbulence models that use two transport equations to model the turbulence (such as k- ε). The model was developed for aerospace applications with wall-bounded flow and is also used for turbomachinery applications (Fluent Inc. 2005).

1.2 Fluid dynamics and the wave equation

In this section the classical wave equation will be derived from the laws of fluid dynamics; the conservation of mass in a fluid (eq 1) and the conservation of momentum in a fluid (eq 2) (Åbom 2008).

For small disturbances of the fluid from its equilibrium state (p_0, ρ_0 constant and $u = 0$), one can write

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

where the ´ means disturbance (= acoustic field) from its equilibrium state.

Inserting this into equation 1 and equation 2, and keeping only linear terms gives the following:

$$\frac{\partial p'}{\partial t} + \rho_0 \nabla \cdot u = m' \qquad (3)$$

$$\rho_0 \frac{\partial u'}{\partial t} + \nabla p' = f_V' \qquad (4)$$

This is known as the classic wave equation and is valid in an ideal fluid. To reach this equation it was assumed that there are no shear forces ($\frac{\partial \tau_{ij}}{\partial x_j} = 0$). In addition, in the derivation of the classical wave equation, the source terms m´ and f_V´ are 0. For the case with source terms see next section.

Using an ideal, lossless fluid, also means that the entropy of a fluid particle is constant. A Taylor expansion for the pressure around the equilibrium state gives:

$$p' = p - p_0 = \left(\frac{\partial p}{\partial \rho}\right)(\rho - \rho_0) + \ldots = K_0 \frac{\rho'}{\rho_0} + \ldots \qquad (5)$$

Where the isentropic (constant entropy) bulk modulus:

$$K_0 = .\rho_0 \left(\frac{\partial p}{\partial \rho}\right)_0$$

The results from equation 5 is now inserted into equations 3 and 4. After performing

$$\frac{\partial}{\partial t}(eq.3) - \nabla \cdot (eq.4)$$

one finally arrive at the wave equation:

$$\frac{1}{c_0^2}\frac{\partial^2 p'}{\partial t^2} - \nabla^2 p' = 0 \qquad (6)$$

Where $c_0 = \sqrt{K_0 / \rho_0}$.

1.3 Lighthill's acoustic analogy

In order to obtain Lighthill´s acoustic analogy, one uses the conservation laws and makes the same derivation as for the classical wave equation, but without making the assumption of an ideal gas and the source terms are no longer disregarded (Åbom 2008). By using equation 1 and 2, equation 7 is derived in the following way:

$$\frac{\partial}{\partial t}(eq.1) - \frac{\partial}{\partial x_i}(eq.2)$$

$$\frac{\partial^2 \rho}{\partial t^2} - \frac{\partial^2}{\partial x_i x_j} = \frac{\partial m}{\partial t} - \frac{\partial f_{V,i}}{\partial x_i} + \frac{\partial^2 P}{\partial x_i x_j}\left(\rho u_i u_j - \tau_{ij}\right) \qquad (7)$$

The same substitutions as in the previous section are made;

$$p(x,t) = p_0 + p'(x,t)$$

$$u(x,t) = u'(x,t)$$

$$\rho(x,t) = \rho_0 + \rho'(x,t).$$

$-\frac{1}{c_0^2}\frac{\partial^2 p'}{\partial t^2}$ is then added to reformulate equation 7 as a wave equation:

$$\frac{1}{c_0^2}\frac{\partial^2 p'}{\partial t^2} - \nabla^2 p' = \frac{\partial}{\partial t}(m + \frac{1}{c_0^2}\frac{\partial}{\partial t}(p' - c_0^2\rho')) - \frac{\partial f_{V,i}}{\partial x_i}\frac{\partial^2}{\partial x_i \partial x_j}(\rho u_i u_j - \tau_{ij}) \qquad (8)$$

Equation 8 is called Lighthill´s acoustic analogy. When treating the right hand side of the equation as source terms, the equation can be simplified into the inhomogeneous wave equation:

$$\frac{1}{c_0^2}\frac{\partial^2 p'}{\partial t^2} - \nabla^2 p' = s(x,t) \qquad (9)$$

where s represents a source.

After solving the inhomogeneous wave equation using Green´s function, the three dimensional, free field solution is obtained:

$$p'(x,t) = \iiint_V \frac{s(y, t - r/c_0)}{4\pi r} dV_y \qquad (10)$$

Where:

$t - r/c_0$ is the emission time.

r is the distance between the source region and the listener.

V is the source region with the unsteady flow processes.

As seen in equation 8 (Lighthill’s acoustic analogy), the sound producing source term s can be split in three parts which represent monopole, dipole and quadropole sources:

$$s_1 = \frac{\partial}{\partial t}(m + \frac{1}{c_0^2}\frac{\partial}{\partial t}(p' - c_0^2\rho')) \qquad \text{Monopole source term} \qquad (11)$$

$$s_2 = -\frac{\partial f_{V,i}}{\partial x_i}$$ Dipole source term (12)

$$s_3 = \frac{\partial^2}{\partial x_i \partial x_j}(\rho u_i u_j - \tau_{ij})$$ Quadropole source term (13)

Where:

$\frac{\partial m}{\partial t}$ is the mass flow per time unit.

ρ' is disturbance in density.

$f_{V,i}$ is the unsteady force acting on the fluid.

u_i is the flow velocity in the mean flow direction.

u_j is the flow velocity perpendicular to the mean flow direction.

$\tau_{ij} = \eta\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial u_i}\right) - \frac{2}{3}\eta\delta_{ij} div(u)$ is the viscous stress tensor.

η is the dynamic viscosity.

The strength of the three source terms scale approximately as:

$$\overline{W}_{mono} : \overline{W}_{di} : \overline{W}_{quadro} \propto 1 : M^2 : M^4 \tag{14}$$

This comparison states that for high Mach numbers M (i.e. the flow speed normalised by the speed of sound) the quadropole sound will dominate. For lower Mach number flows the monopole or dipole sound will dominate.

According to Lighthill's acoustic analogy, it is possible to split a flow field into a flow part and an acoustic part, i.e. the wave equation. This result is also an important limitation of Lighthill's theory; in situations where the generated acoustic field modifies the flow, Lighthill’s theory cannot be used. Examples of such cases are feedback processes between the acoustic field and the flow, such as whistles and dissipation of sound due to vortex shedding.

1.3.1 Monopole source

The monopole source term represents fluctuating mass injection or volume flow (Åbom 2008). Examples of monopole sources in fluids are: sound from exhaust outlets, combustion, and collapsing bubbles in liquids. An example of a monopole source that is not flow generated is a loudspeaker mounted in a box.

Figure 1 depicts a flow outlet from a pipe, such as an exhaust, which generates a monopole source.

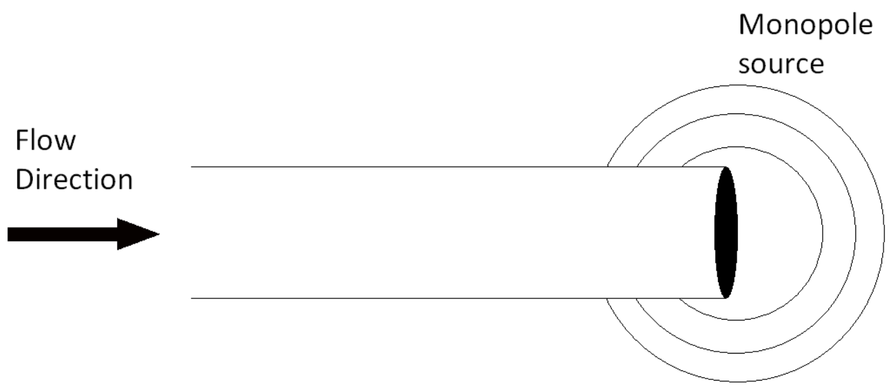

Figure 1. *Monopole source.*

The sound field generated by multiple monopole sources is given by a superposition of their individual sound fields:

$$p'(x,t) = \iiint_V \frac{\dot{m}_t(y, t - r/c_0)}{4\pi r} dV_y \qquad (15)$$

Where:

$\dot{m}_t = \partial m_t / \partial t$

$m_t = m + (1/c_0^2)\partial(p' - c_0^2 \rho')/\partial t$

As seen above it is the unsteady parts that alone generate sound.

If the monopole source is compact, equation 15 can be simplified. A source can be considered compact when the dimension of the source is smaller than the emitted wavelengths and small in comparison to the distance between the source and the observer. In these cases, equation 15 can be written as:

$$p'(x,t) = \frac{\rho_0 \dot{Q}(t - x/c_0)}{4\pi x} \qquad (16)$$

Where:

$\dot{Q} = \partial Q / \partial t$

$Q(t) = \iiint_V q'(y,t) dV_y$

q' is the unsteady volume flow/m^3

The power that radiates from a periodic monopole source is:

$$\overline{W_m} = \frac{\overline{\dot{Q}^2}}{4\pi c_0} \qquad (17)$$

1.3.2 Dipole source

A dipole source is created when an object produces a periodic flow separation, for example around a car antenna or other solid objects (Åbom 2008). This creates an oscillating force on the surrounding medium. If one continues with the example of a loudspeaker from the monopole section, a dipole would be a loudspeaker without a box. Although there is now mean flow involved, the loudspeaker without box will produce an oscillating force on the air. As long as the loudspeaker is small compared to the wavelength, it represents a dipole source.

Figure 2 depicts flow around a solid object, which causes periodic vortex shedding which in turn generates the dipole source.

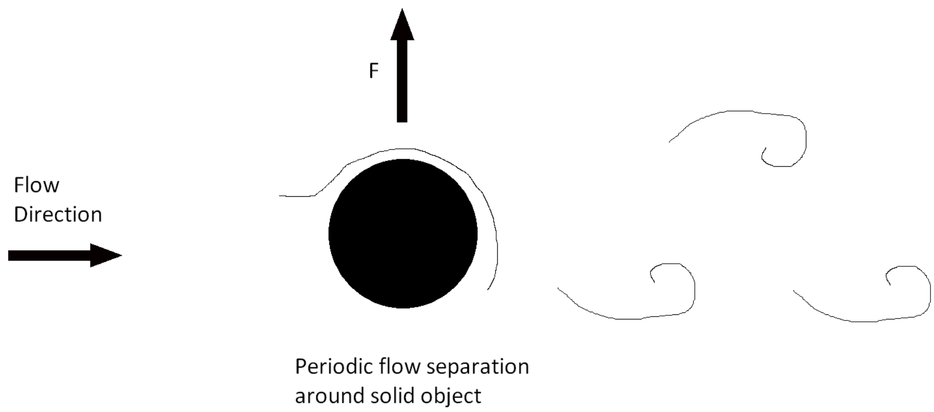

Figure 2. ***Dipole source.***

The sound field from a superposition of dipole fields is:

$$p'(x,t) = -\iiint_V \frac{\partial}{\partial x_i}\left(\frac{f_{v,i}(y,t-r/c_0)}{4\pi r}\right)dV_y \qquad (18)$$

If the dipole source is compact it can be simplified to:

$$p'(x,t) = -\frac{\partial}{\partial x_i}\left(\frac{F_i(t-x/c_0)}{4\pi x}\right) \qquad (19)$$

Where $F_i(t) = \iiint_V f_{V,i}(y,t)dV_y$ i.e. the total force acting on the fluid.

The sound power that radiates from a dipole source is:

$$\overline{W_d} = \frac{\overline{\dot{F}^2}}{12\pi\rho_0 c_0^3} \qquad (20)$$

Where $\dot{F} = \partial F / \partial t$

1.3.3 Quadropole source

The quadropole source term is generated by the unsteady part (acceleration) of the momentum transportation in a flow that causes shear forces (Åbom 2008). This is very important in turbulent flows and especially for high speed free jets.

Figure 3 below depicts a pipe outlet with a high velocity flow. The turbulence generates a quadropole source.

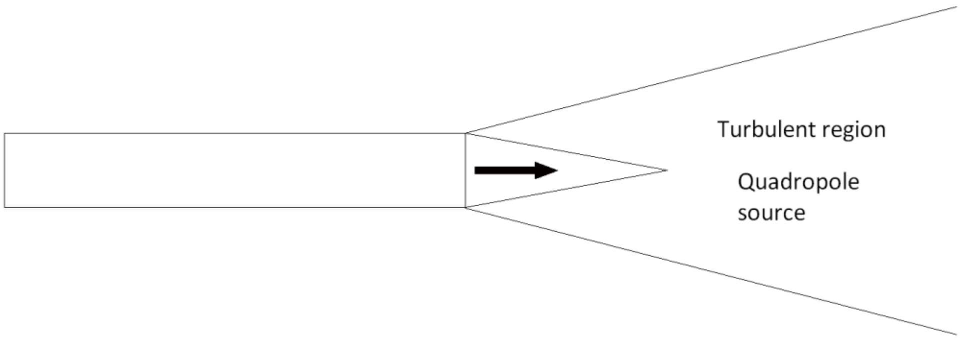

Figure 3.Quadropole source (Jet)

The total sound field from a superposition of quadropole fields is:

$$p'(x,t) = \iiint_V \frac{\partial^2}{\partial x_i \partial x_j}\left(\frac{T_{ij}(y, t - r/c_0)}{4\pi r}\right) dV_y \qquad (21)$$

Where:

$T_{ij} = \rho u_i u_j - \tau_{ij}$ (The Lighthill tensor)

If the quadropole source is compact it can be simplified to:

$$p'(x,t) = \frac{\partial^2}{\partial x_i \partial x_j}\left(\frac{Q_{ij}(t - x/c_0)}{4\pi x}\right) \qquad (22)$$

Where

$Q_{ij}(t) = \iiint_V T_{ij}(y,t) dV_y$ is the quadropole strength

The sound power that generates from a quadropole source is:

$$\overline{W_q} = \frac{\varepsilon_{ij} \overline{\ddot{Q}_{ij}^2}}{\rho_0 c_0^5} \qquad (23)$$

Where:

$\varepsilon = 1/20\pi$ when i=j and $\varepsilon = 1/60\pi$ when i≠j

$\ddot{Q}_{ij} = \partial^2 Q_{ij} / \partial t^2$

1.4 Additional aeroacoustic models

1.4.1 Curle's equation

Lighthill's theory can be extended with Curle's equation to include the effect of non-moving solid surfaces (Åbom 2008). In the derivation of Curle's equation, a turbulent flow is assumed. Figure 4 depicts a solid object in a flow that creates a sound generating volume.

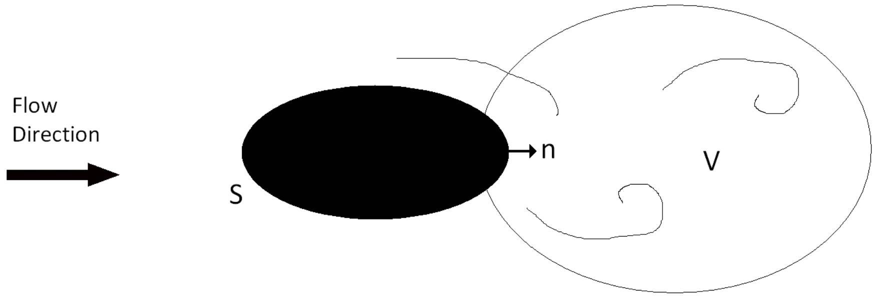

Figure 4 Sound generation explained by Curle´s equation.

Curle's equation is written:

$$p'(x,t) = \iiint_V \frac{\partial^2}{\partial x_i \partial x_j}\left[\frac{\rho u_i u_j}{4\pi r}\right]_{t_e} dV_y - \iint_S \frac{\partial}{\partial x_i}\left[\frac{p n_i + \rho u_i u_j n_j}{4\pi r}\right]_{t_e} dS_y + \iint_S \frac{\partial}{\partial t}\left[\frac{\rho u_i n_j}{4\pi r}\right]_{t_e} dS_y \quad (24)$$

where t_e means that the brackets are calculated at $t=t_e$.

The volume integral represents quadropole sound fields; the first surface integral represents dipole fields and the second surface integral represents monopole fields.

Assuming that the flow induced vibrations (monopole term) can be neglected, the sound field from an acoustically compact, non-moving solid object can be written as:

$$p'(x,t) = \iiint_V \frac{\partial^2}{\partial x_i \partial x_j}\left[\frac{\rho u_i u_j}{4\pi r}\right]_{t_e} dV_y - \frac{\partial}{\partial x_i}\left(\frac{F_i(t - x/c_0)}{4\pi x}\right) \quad (25)$$

Where $F_i(t) = \iint_S p(y,t) n_i dS_y$ is the fluctuating force from the object that acts on the fluid.

The volume integral still represents the quadropole source field and the second term represents the dipole source field.

For a small Mach number flow around a body, the dipole sound (fluctuating forces) will dominate, while the quadropole sound (turbulence) will be dominating at high Mach numbers.

1.4.2 Ffowcs Williams and Hawkins equation

In order to model a body performing an arbitrary motion through a turbulent flow one needs a generalization of Curle's equation, called the Ffowcs Williams and Hawkins equation, or in short FWH (Åbom 2008):

$$\breve{p}'(x,t) = \frac{\partial}{\partial t} \oiint_{S_e} \frac{\rho_0 V(y,t_e) \cdot n_e}{4\pi r_e (1 - M_{r,e})} dS_y - \nabla \cdot \oiint_{S_e} \frac{p_0'(y,t_e) n_e}{4\pi r_e (1 - M_{r,e})} dS_y \quad (26)$$

Where:

$\breve{p}' = p'H(f)$

$$H(f) = \begin{cases} 1, x > 0 \\ 0, x < 0 \end{cases}$$ (H is the Heaviside function)

$$f(x,t) \begin{cases} > 0, x \notin V \\ = 0, x \in S \\ < 0, x \in V \end{cases}$$

$S_e(x,t)$ is the surface that is formed by the source points.

$r_e = |x - y(t_e)|$

S is the surface of the object

V is the volume of the object

This equation contains two source terms. The first term corresponds to sound created by the relative motion between the body and the fluid (thickness noise). The second term corresponds to the pressure distribution over the surface (lift noise) which can be obtained from e.g. propellers or fans. Hence, the Ffowcs Williams and Hawkins equation is a good way to calculate sound from these sources (Åbom 2008, Fluent Inc. 2005).

If the source is compact it can be simplified to:

$$\breve{p}'(x,t) = \frac{\partial^2}{\partial t^2}\left(\frac{\rho_0 V}{4\pi r_e(1-M_{r,e})}\right) - \nabla \cdot \left(\frac{F_e}{4\pi r_e(1-M_{r,e})}\right) \qquad (27)$$

The FWH equation cannot account for the backward coupling of sound on flow. Neither can it account for reflections or obstacles between the source and receiver. Because of these limitations, the FWH equation should mainly be chosen for external flow induced noise simulations. If there is an obstacle between the source and the receiver (such as the housing surrounding a fan), the FWH equation must be used together with a BEM solution to account for the radiated sound from the object. The solid surfaces surrounding the sound source can be modelled either as impermeable (walls) or permeable (inlets/outlets).

1.5 Sound propagation

This section will cover the theory behind sound propagation in ducts.

1.5.1 Sound in ducts

Rigid walls are a standard assumption when calculating sound in ducts. (Fluent Inc. 2005). This assumption is a good approximation for ducts filled with gas but not for ducts filled with liquid. The liquid will interact with the walls which in turn are going to vibrate and radiate sound (Åbom 2008).

A propagating acoustic wave along the axis of a duct, assuming harmonic time dependence, is given by:

$$p'(x,t) = \hat{p}\Psi(y)e^{i(\omega t - k_1 x_1)} \qquad (28)$$

Where

$\Psi(y)$ is a sinus or cosines function that describes the pressure distribution over the duct's cross section.

y is the position in the x_2, x_3 plane.

For frequencies below the cut on frequency, only plane waves propagate in the duct. The cut on frequency for a mode n is:

$$f_n^c = \frac{c_0 k_{\perp,n}}{2\pi}\sqrt{1-M^2} \qquad (29)$$

The total pressure at a point inside the duct is given by a summation of all possible modes:

$$p'(x,t) = \sum_n (\hat{p}_n^+ \Psi_n(y)\exp(i(wt - k_{1,n}^+ x_1) + \hat{p}_n^- \Psi_n(y)\exp(i(wt - k_{1,n}^- x_1) \qquad (30)$$

Where + and – signs denote positive or negative x_1 direction. For a circular duct, or a rectangular duct with dimensions 2a x 2b, the eigenfunctions and eigenvalues are given by the following equations, assuming rigid walls

$$k_{\perp,mn}^2 = (\frac{m\pi}{2a})^2 + (\frac{n\pi}{2b})^2 \qquad \text{and}$$

$$\Psi_{mn}(x_2, x_3) = \Psi_m(\frac{m\pi x_2}{2a})\Psi_n(\frac{n\pi x_3}{2b})$$

Where m, n = 0,1,2,… and $\Psi_j(x) = \cos(x)$ when j is even and $\Psi_j(x) = \sin(x)$ when j is odd.

If the assumption of rigid walls cannot be made (e.g. when the present fluid is water instead of air) the sound pressure couples to the duct wall and causes it to vibrate.

If a larger duct radius is used, the noise will decrease since increased radius will decrease the mean flow speed. The noise is proportional to u^5.

1.5.2 Multi port theory

For frequencies below the cut-on frequency, only plane waves exist at the openings and the duct system can be modelled as an acoustic two-port (if wall vibrations can be disregarded) (Åbom 2008).

In multi-port theory, each element is described by a transfer matrix. The acoustic pressure and the volume velocity are used to describe the input and output states of each element. The transfer matrix is therefore:

$$\begin{pmatrix} \hat{p}_{inlet} \\ \hat{q}_{inlet} \end{pmatrix} = T \begin{pmatrix} \hat{p}_{outlet} \\ \hat{q}_{outlet} \end{pmatrix} \quad (31)$$

Where:

$\hat{p}_{inlet}$ is the acoustic pressure at the inlet of the element

$\hat{q}_{inlet}$ is the volume flow at the inlet

$\hat{p}_{outlet}$ is the acoustic pressure at the outlet

$\hat{q}_{outlet}$ is the volume flow at the outlet

If there is a discontinuity (duct area change) in the interface between two elements (called element 1 and element 2 below), the volume flow remains constant (for incompressible flows), while the acoustic pressure is influenced by an impedance:

$$\hat{p}_{outlet,1} - \hat{p}_{inlet,2} = Z_{inlet,2} q_{inlet,2} \quad (32)$$

In order to avoid mistakes, the discontinuity can be included in the two port matrix. If $\hat{p}_{outlet,1} - \hat{p}_{inlet,2} = Z_{inlet,2} q_{inlet,2}$ and $\hat{q}_{outlet,1} = \hat{q}_{inlet,2}$ is chosen, the transfer matrix is obtained:

$$T_{outlet,1} = \begin{bmatrix} 1 & Z_{inlet,2} \\ 0 & 1 \end{bmatrix} \quad (33)$$

With a number of elements in a row, the total transfer matrix is a product of all the elements transfer matrixes:

$$T_{tot} = \prod_{n=1}^{N} T_n \quad (34)$$

2 Practical problem solving

This chapter explains different flow generated sound phenomena. A few different problems and dimensioning rules will be discussed: external flow, such as for moving vehicles, fluid driven whistles and sound in ducts.

2.1 External Flow

The sound created by flow separation around moving vehicles increase with velocity. A combination of a CFD solver such as Fluent and an acoustic solver such as LMS SYSNOISE, VNOISE or ACTRAN-LA can be used to simulate external flow induced sound. More information about these methods can be found in section 4.

The aim of the simulations is to design the objects and their surfaces so that the vortex shedding is reduced. Which model to use depends on the speed of the flow around the object.

- For low Mach numbers the generated sound is mostly of dipole type, caused by flow separation (Åbom 2008). In this case a model such as FWH that takes dipole sources into account, should be used (Fluent Inc. 2005).
- When the Mach number is very high but still below 1, the quadropole sound source terms will dominate (Åbom 2008). In these cases a broadband turbulence model can be used to save computational time (Fluent Inc. 2005).

2.1.1 Cars

The flow separation noise from cars is mostly created by the side mirrors and the rear wake. The tyre noise is created by vibrations but also from compressed air (monopole sources). For high speeds, tyre noise is the dominating sound source. But for even higher speeds, around 200 km/h, the sound created by flow separation dominates.

2.1.2 Trains

For velocities of around 250 km/h or more, the aeroacoustic sound is higher than the sound created by the wheels on the rail and it becomes the dominating source of sound. For future high speed trains with velocities well above 250km/h, the aeroacoustic sound will most likely be very high.

2.2 Fluid driven whistles

When an unstable flow process couples to an acoustic field, a fluid driven whistle is created (Åbom 2008). The most common example of this is periodic flow separation that occurs either around a body (type 1 whistle) or between an upstream and downstream edge (type 2 whistle). Oscillations in the flow are called Strouhal tones. When these oscillations match, or are close to, an acoustic/structural mode it forms a positive feedback loop. The combined vibration will grow until losses limit its amplitude.

It shall be noted that even if the Strouhal tone matches the resonance frequency, it does not guarantee a creation of a feedback loop. But since there are few complete models to predict whistles, the best one can do is to try to keep the Strouhal tone and the resonance frequency separated.

2.2.1 Type 1 whistle

The difference between a type 1 whistle and a regular dipole source generation is basically that for a whistle, the object exposed to the flow moves and resonates in phase with the flow separation instead of being completely fixed. One example of a type 1 whistle is a car antenna, placed perpendicular to the flow direction. The movement of the antenna causes periodic vortex shedding which in turn generates sound. The vortex shredding frequency is called the Strouhal frequency and is given by:

$$f_{st} = St\frac{U}{d} \qquad (35)$$

Where:

f_{st}= frequency of vortex shedding[s^{-1}]

U= Velocity of the flow [m/s]

d = diameter of object (cylinder) [m]

St= Strouhal number [-] (= 0,2 for a cylinder)

The periodic flow separation will create a periodic force **F** on the rod. This force excites bending wave modes in the rod so that the rod in turn acts like a dipole source with a force -**F** on the fluid. This dipole can excite acoustic modes that have a velocity component parallel with **F** so that energy can be put into the mode. When the flow separation couples to a structural or acoustic mode, the flow separation will be synchronized and the input of energy will therefore be more efficient.

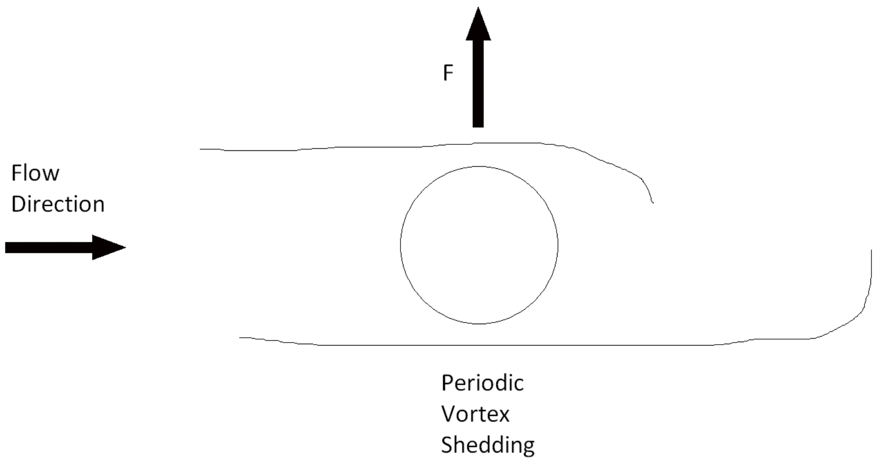

Figure 5. Type 1 whistle.

The sound from type 1 whistles can be reduced by either disturbing the flow separation or by reducing the coupling to the mechanical mode In order to disturb the flow separation, one can put irregularities on the object. The coupling can be avoided by increasing the damping in the object. If the Strouhal frequency is fixed due to unchanging flow speed, the coupling can be eliminated by changing the mechanical mode frequency. (Åbom 2008)

2.2.2 Type 2 whistles

A vortex is shed at the first edge of a wall opening. Since the flow is unstable the vortex grows while travelling with the flow downstream. When the vortex reaches the downstream edge, it creates a pressure pulse that travels upstream. This pressure pulse will create a new vortex shedding at the first wall and the process starts over. The tone produced by blowing air across a bottle is a type 2 whistle.

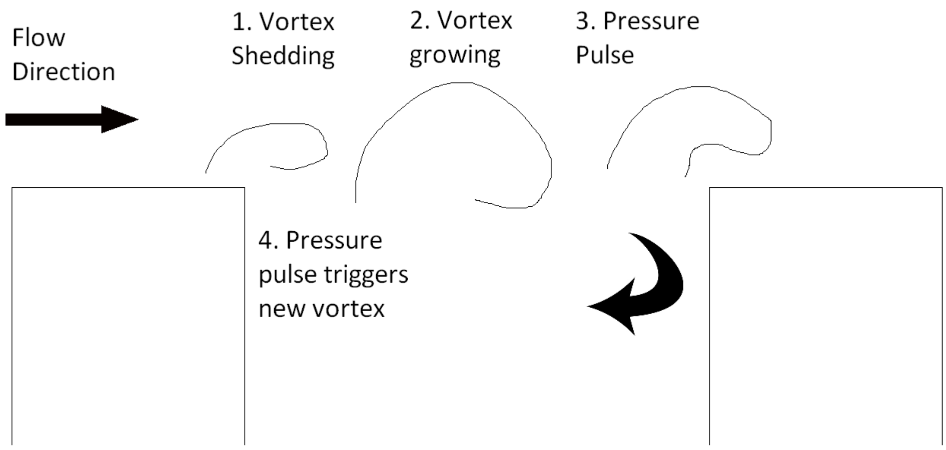

Figure 6. Type 2 whistle.

The frequency for this type of whistle is given by:

$$f_{st,n} = \left(\frac{n - \phi_r / 2\pi}{1/\gamma + M} \right) \frac{U}{d} \qquad (36)$$

Where:

d is the distance between the edges [m]

γ = U_c/U (≈ 0,4 for a cavity in a wall) [-]

U_c is the convection speed of the vortex [m/s]

ϕ_r is a phase angle (≈ π/2 for a cavity in a wall)

Whistling can be avoided by disturbing the vortex shedding/flow separation by making the edges uneven or changing the opening diameter. You can also affect the mechanical resonance by adding damping to the resonator through absorbing material or move the resonance frequency by changing the geometry. (Åbom 2008)

2.2.3 Other whistles

2.2.3.1 Subsonic jet oscillations

There are two important examples of subsonic jet oscillations: the hole/ring and the edge tones. They occur when a jet interacts with a downstream hole or sharp edge. Maximum sound pressure levels are obtained when the distance to the downstream hole/ring or wedge is between one and ten times the dimensions of the hole/ring or wedge. (Åbom 2008)

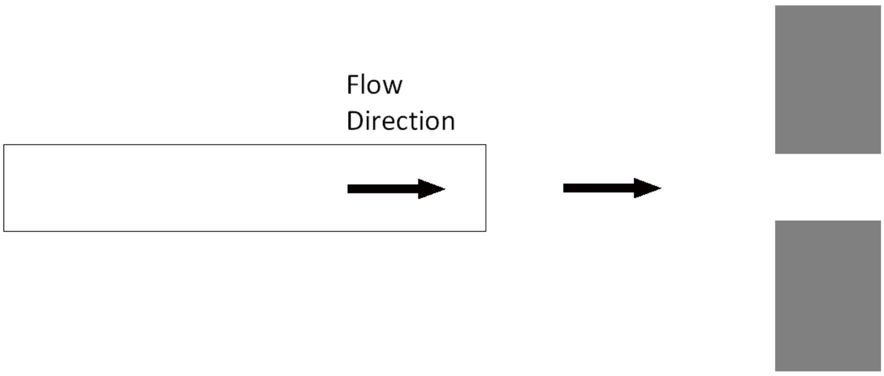

Figure 7. When a subsonic jet interacts with a downstream ring or hole, a whistle is created.

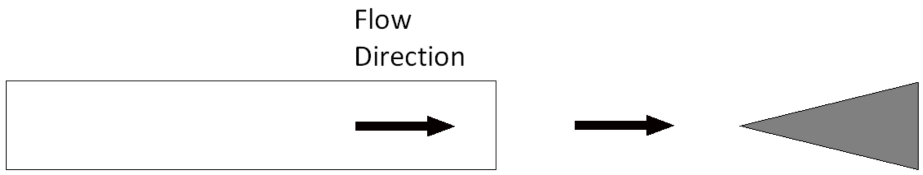

Figure 8. When a subsonic jet interacts with a downstream wedge, a whistle is created.

2.2.3.2 Supersonic jet oscillations

For supersonic flows there is one important example that is known as jet screech. It is caused by shock cells that create a feedback loop through periodic compression and expansion from a jet outlet. (Åbom 2008)

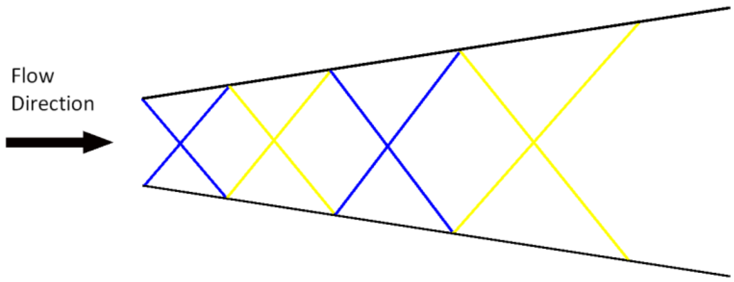

Figure 9. Jet-screech caused by periodic compression (blue) and expansion (yellow).

2.3 Ducts

In order to reduce noise generation in a duct, it should preferably be designed with graduate transitions instead of abrupt corners.

Duct changes should be separated in order to obtain a "normal" flow before the flow reaches another duct change.

Absorbing material can be added inside (if possible) or around the duct. Absorbing material placed inside the duct will require a lot of space to substantially reduce the sound. Therefore thin perforated plates are often used.

The duct walls can be stiffened in order to shift the resonance frequency and reduce the sound it radiates.

Flow noise generated by a duct change can be attenuated directly after the change with a resonator adjusted for the specific frequency. (Åbom 2008)

2.4 Structure borne sound and fan resonance

Vibrations from a fan or a duct can be transmitted to the carrying structure, which in turn can radiate sound in the room. This structure borne sound can be reduced by adding vibration isolation between the fan or duct and the structure.

If the generated frequency is close to the coincidence frequency of the structure, the acoustic forces will be transmitted to the structure and sound will radiate from the structure. However, the frequency of the structure can be avoided, and the volume flow kept unchanged. This is done by either changing to a faster spinning fan with smaller blades, or by changing to a slower spinning fan with larger blades.

In some areas of use, such as cooling fans in electronic equipment, the fan can be replaced by a passive heat sink solution or, if necessary, a passive solution together with a slow moving, quiet fan.

If one wants to simulate the acoustic–structure interaction, software such as COMSOL Multiphysics can be used (see simulation section).

Calculations of some basic pipe and beam resonance frequencies can be found in the Appendix.

3 Simulations

In order to get to know the different solving approaches and the different software some simulations were performed. This section covers different solving approaches, choice of software and ends with a walk through.

3.1 Performed Simulations

Flow across a wedge (2D) and flow through a pipe (2D) was meshed in Gambit and simulated in Fluent. Each geometry was analyzed both with a FWH model using a transient solution, and a broadband noise source model using a turbulence solution. The aim of the simulations was to compare the transient solution to the turbulence solution, and to see if sound sources could be visualized using this method.

The FWH simulation was performed by first running a transient solution until it became statistically satisfying. Then the FWH model was activated and the sound sources was calculated by continuing the simulation.

3.1.1 Pipe flow

The pipe that was simulated was 8 m long and 0,2 m in diameter. The medium was air. Flow speed was 50 m/s at the inlet. The figure below depicts the near wall model of the grid. A k-ε model was used.

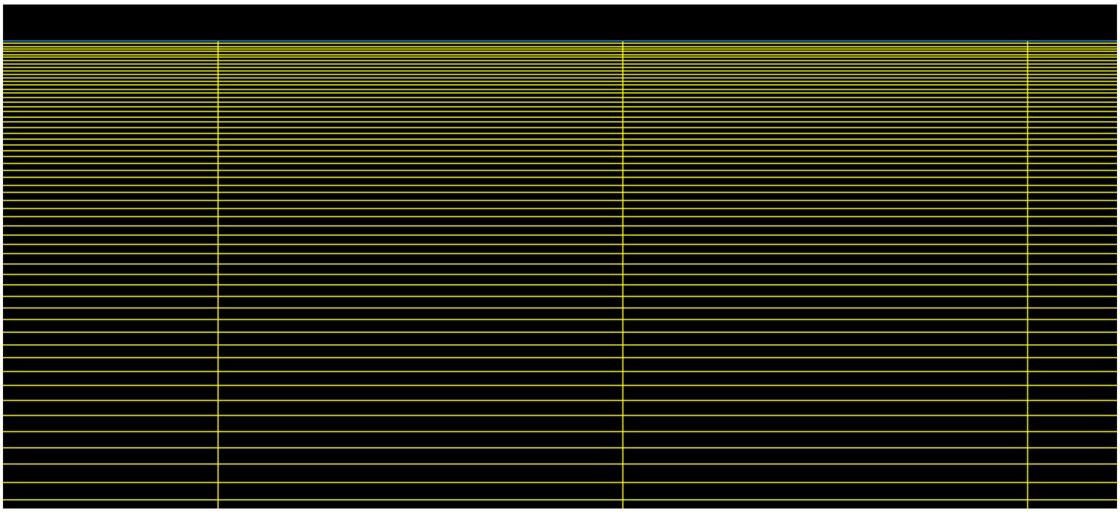

Figure 11. The grid near the wall of the pipe.

3.1.1.1 Broadband model results

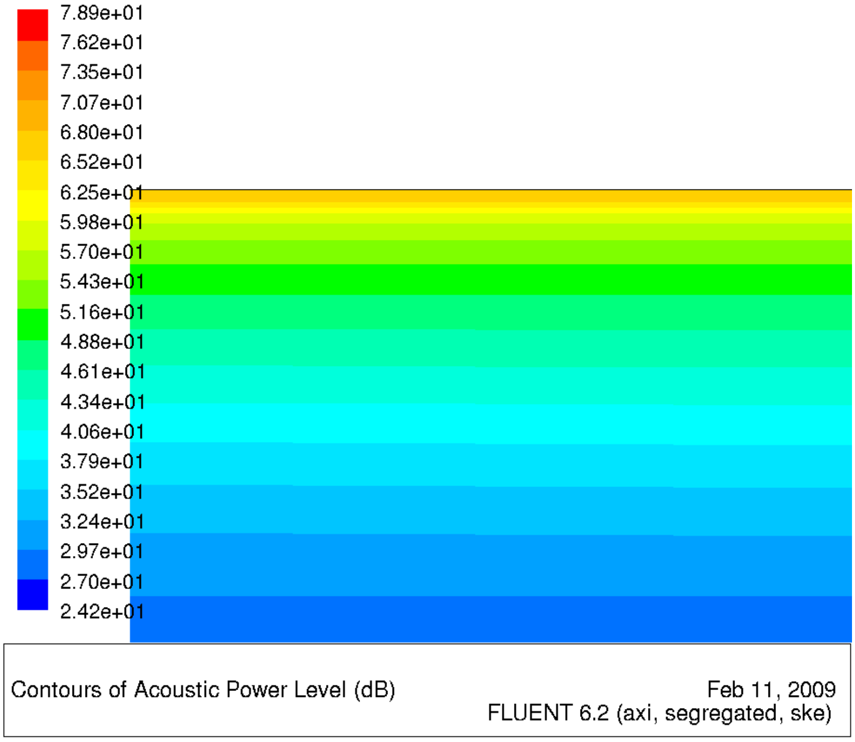

Figure 12. Acoustic power level obtained using the broadband noise model.

3.1.1.2 FWH results

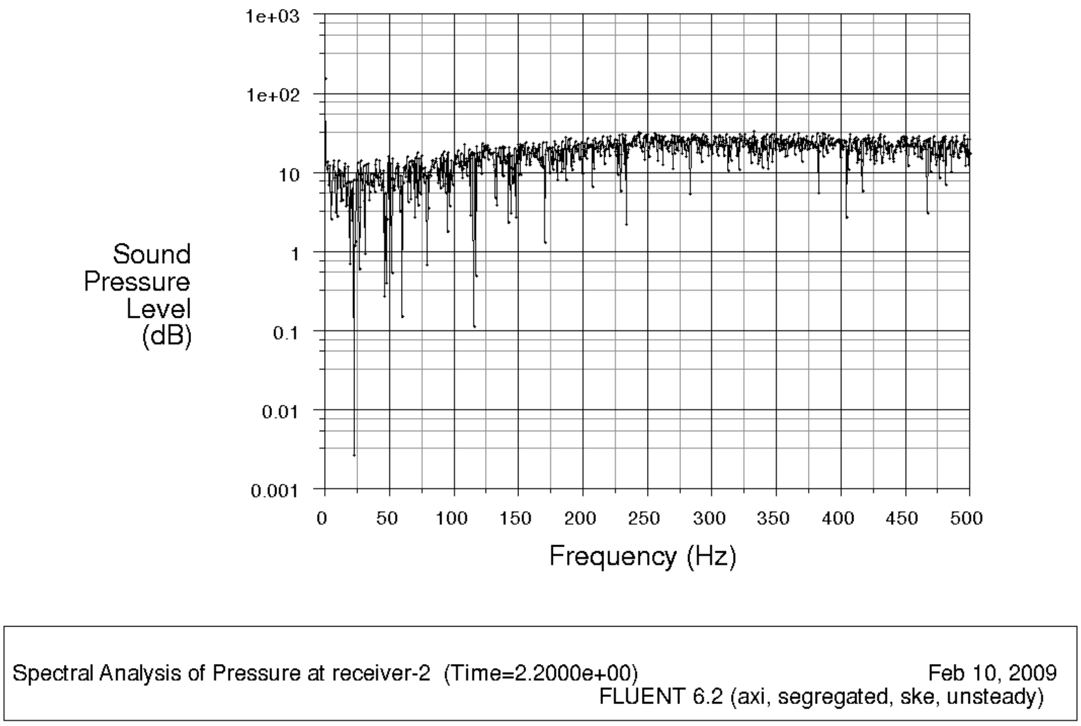

Figure 13. Frequency spectra of sound pressure level obtained in mid flow.

3.1.1.3 Comparison of velocity magnitude

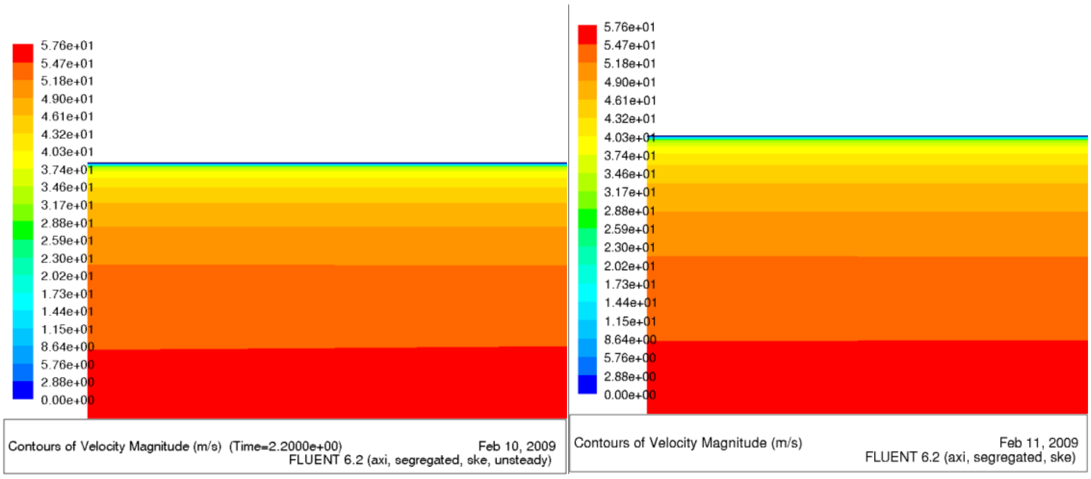

Figure 14. Contours of velocity magnitude using transient solution (left) and a turbulence model (right).

3.1.1.4 Comparison of turbulence kinetic energy

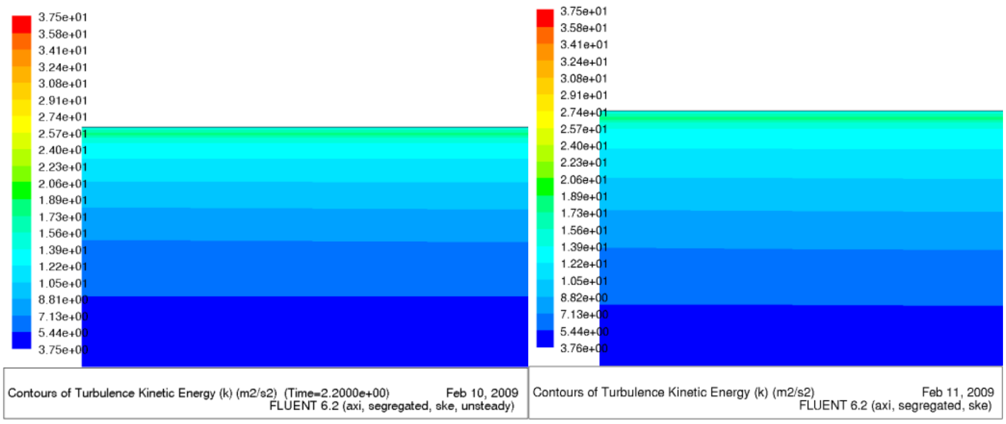

Figure 15. Contours of turbulence kinetic energy using transient solution (left) and a turbulence model (right).

3.1.2 Flow over wedge

The wedge is 0,2 m long. The flow is coming from the left at 3º angle of attack. The medium was air. Flow speed was 86 m/s at the inlet.

3.1.2.1 Broadband model results

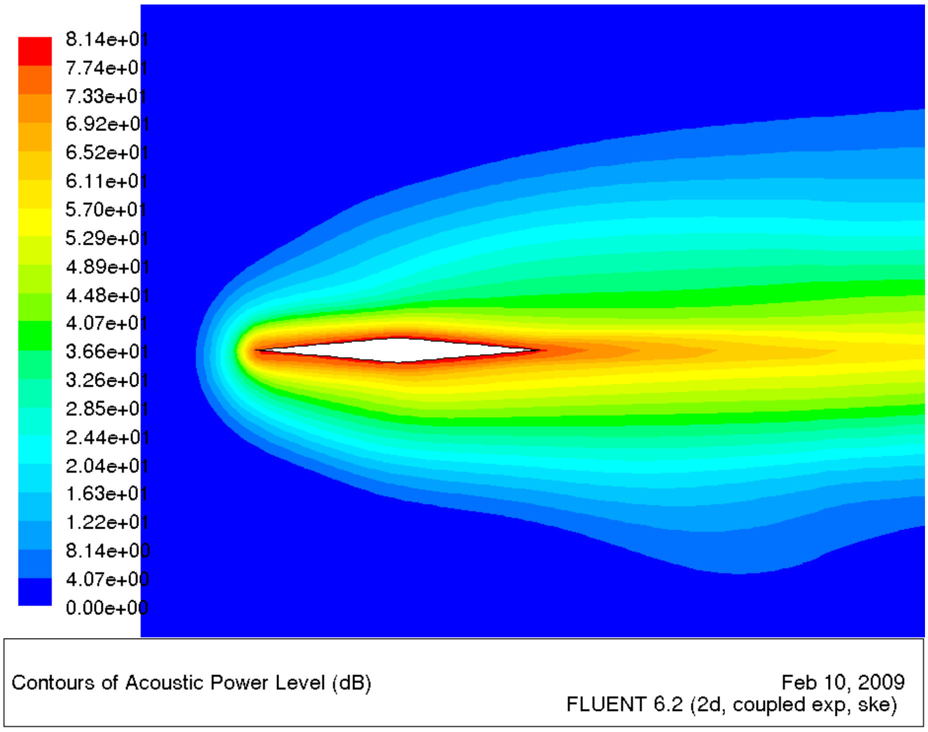

Figure 16. Acoustic power level obtained using the broadband noise model.

One of the most important results from the turbulence simulation is the shear noise sources. This figure below shows where most of the turbulence generated sound is created.

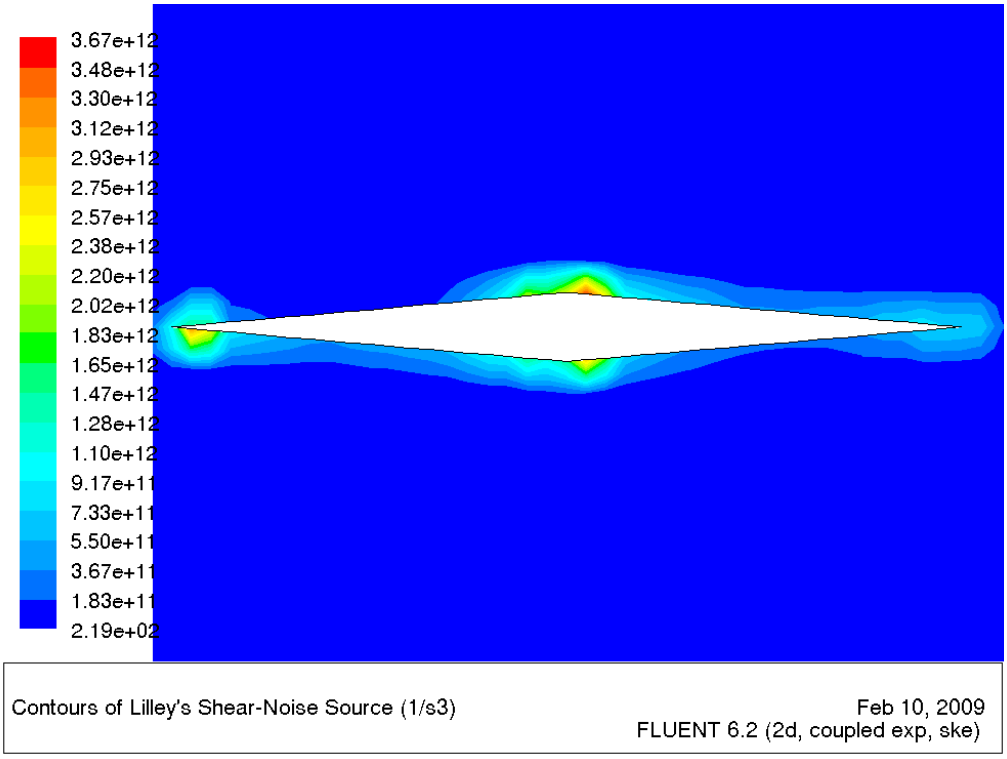

Figure 17. Contours of Shear-noise sources obtained using the turbulence model.

3.1.2.2 FWH results

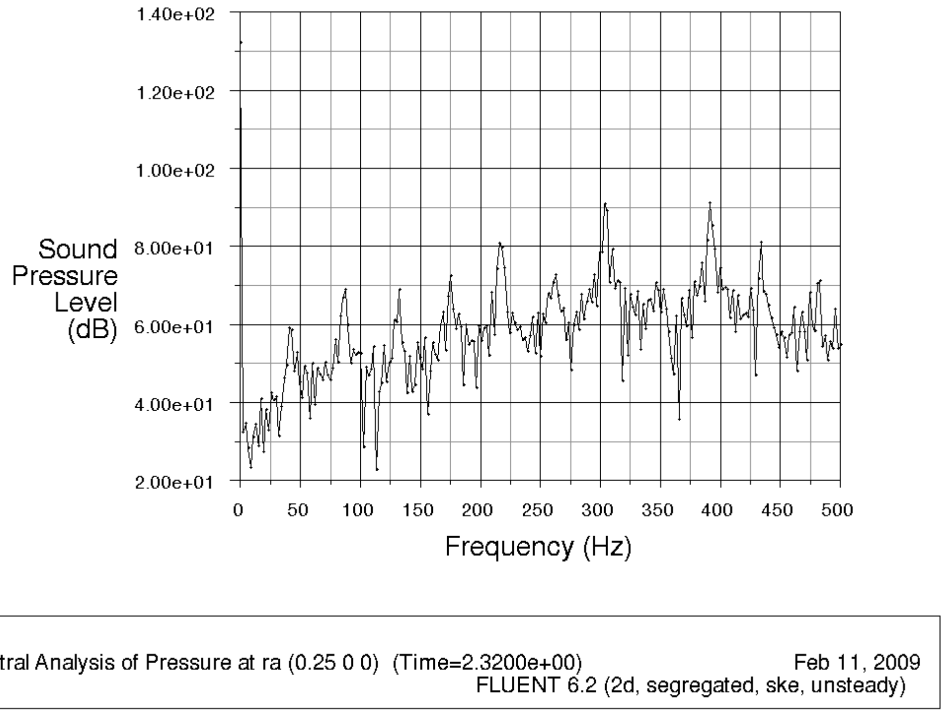

Figure 18. Frequency spectra of sound pressure level obtained behind the wedge using the FWH-model.

For the wedge peaks appear in the spectrum. These are probably due to the dipole sources that are created in the flow across the wedge.

3.1.2.3 Comparison of velocity magnitude

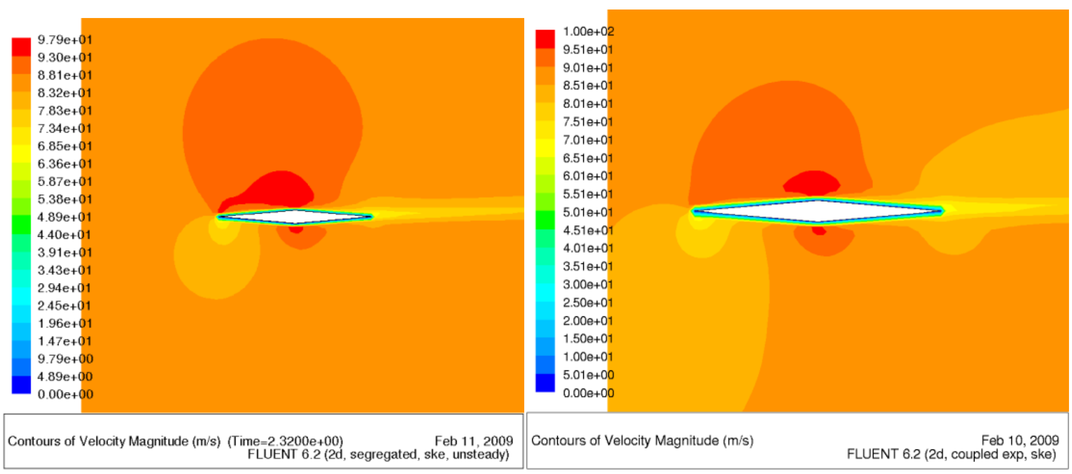

Figure 19. Contours of velocity magnitude using transient solution (left) and a turbulence model (right).

3.1.2.4 Comparison of turbulence kinetic energy

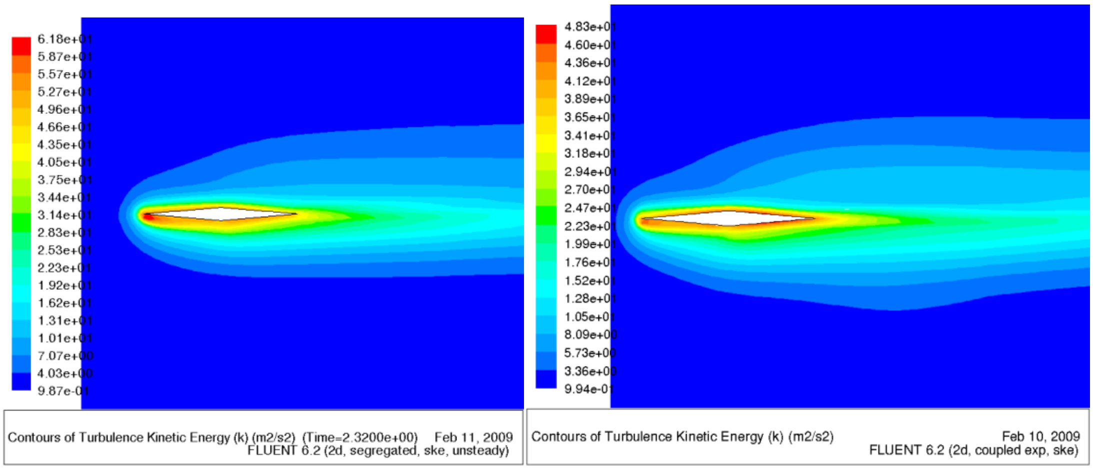

Figure 20. Contours of turbulence kinetic energy using transient solution (left) and a turbulence model (right).

In these figures you can see that a lot of turbulence is generated at the front of the wedge.

3.1.2.5 Comparison of static pressure

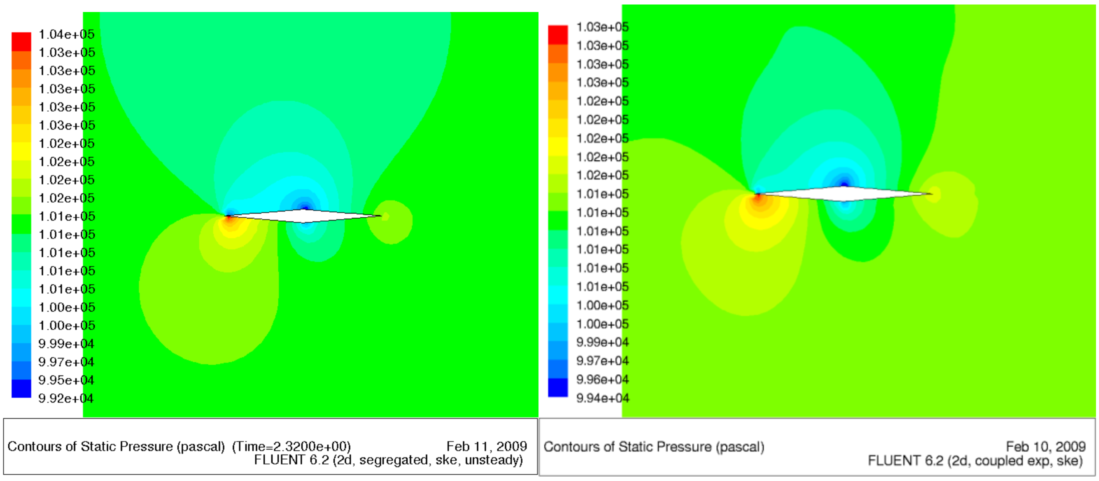

Figure 21. Contours static pressure using transient solution (left) and a turbulence model (right).

3.2 Solving approaches

Commercial solvers are based on different approaches for simulating aeroacoustics.

3.2.1 Direct approach

The direct approach is the most accurate but also the most computational demanding. It is too expensive for most practical problems.

It uses a CFD code to calculate the sound generation and propagation from the Navier Stokes equations. Since the method is a direct simulation it does not make any simplifications to reduce computanion time. It simply computes a transient CFD solution and the static pressure at different microphone positions, all done in one program. It can also handle sound to flow coupling. This method is the easiest to use and understand, but it requires far too much CPU-time.

The main problems with this method are:

- Acoustic timescales different from turbulence timescales.
- Large number of time steps.
- Computational domain must reach from source to observer.
- Far field problems must use very large mesh sizes (car heard from a distance, airplane heard on the ground etc.).
- Acoustic pressure much smaller than atmospheric pressure.
- Very high order of discretization.

Because of these computational demanding drawbacks, this method can seldom be used in practise. If it would be used it would be for low frequencies, small distances between sources and high sound pressure levels.

As processing power increases the CAA simulations will become economically feasible. This will result in more accurate results with no simplifications. (Fluent Inc. 2005)

3.2.2 Hybrid approach

The combination of CFD and a wave equation solver is called "hybrid approach" and significantly reduces the computational time. It allows complex flows and geometries. The acoustics and the flow of a compressible fluid are closely coupled together but it is possible to split the problem in two parts; the fluid flow problem and the acoustic problem. However, important to remember is that these models cannot account for the sound to flow coupling, they can only calculate the flow to sound coupling. A numerical CFD-solver such as Fluent, using Navier-Stokes equations solves the fluid flow problem and gives you the acoustic sources. In order to get the far field sound, the acoustic problem is solved by an acoustic solver such as VNoise or Actran, using the acoustic sources found by the CFD-analysis. Some commercial solvers (including Fluent) include both the CFD- and a simplified version of the acoustic method. The acoustic solver is based on Lighthill's equation or its expansions made by Curle and Ffowcs Williams and Hawkins (see theory section).

The FWH method is an integral method which makes the solution valid only for external propagation problems with a free field assumption. But in reality, walls or casings are often in the way of the acoustic waves.

Therefore some simulation tools, such as VNoise uses the FWH method to find the sources and then uses the BEM to calculate the radiated sound.

The integral approach allows lower mesh density for the acoustic mesh than for the CFD mesh.

It can be used for high Mach number aeroacoustic problems via a permeable surface method.

The integral approach, i.e. the FWH method, is known to be too demanding for high numbers of observer positions and number of sources. The FEM method is only sensitive to the complexity of the source. So if the number of observer locations is low, the geometry of the problem is small and CPU time is limited then FWH can be used. (Fluent Inc. 2005)

3.2.3 Broadband noise source models

Broadband noise source models use RANS-equations (Reynolds-averaged Navier-Stokes) instead of the LES technique to compute the sound sources in the CFD simulation. It can only calculate quadropole source terms and is less accurate but at the same time a lot less time consuming. Therefore it is well suited to first find the most dominant broadband (turbulence) sources of sound in a construction and then try different designs to investigate possible improvements. This method might also be the only practical way to achieve simulations of larger objects such as a whole car, duct systems etc. This method only predicts the sound locally and not at a receiver location. It doesn't need a transient CFD solution; it just needs the mean velocity field, the turbulent kinetic energy (k) and the dissipation rate (ε) from the RANS model. This is what makes it so fast in comparison to other models. (Fluent Inc. 2005)

3.3 Working method for simulations

Flow acoustic simulations can be very helpful when you want to predict the acoustic properties or quickly find the cause of flow induced problems. For detailed information on how to perform the simulations see "Walk Through" section.

3.3.1 Choice of software

There are many ways to simulate the same problem, and many programs to do it with. What you choose depend on what you want to model, how much computing power you have, and what program licences are available. Here follows a list of different ways to model flow generated noise for external or internal flow.

3.3.1.1 Fluent with FWH module (FWH)

This method is appropriate to use for external flow. Fluent CFD-code solves the transient flow simulation in and around the source regions. The time varying surface pressure on the source surfaces is saved. Since Fluent already includes a FWH-solver, the sound pressure level at predefined locations is found quickly. Both compressible and incompressible fluid flow can be handled.

Examples of suitable applications are: side view mirrors on cars, landing gear on airplanes, rotor noise, and jet noise.

The FWH module is included in the standard Fluent package which is a licence that ÅF already have. If more advanced acoustic calculations are needed, it is recommended to use Fluent together with an acoustic simulation tool such as SYSNOISE or VNoise (see below).

Requires:

- Fluent 6.2 or later

3.3.1.2 Fluent + SYSNOISE (BEM)

The data from Fluent's CFD analysis is read into SYSNOISE which then calculates the resulting radiated or scattered noise using BEM. Since SYSNOISE uses BEM, it only needs the pressure fluctuation on the boundaries, which reduces the necessary computational time.

Requires:

- Fluent 6.2 or later
- LMS SYSNOISE Kernel
- LMS SYSNOISE Harmonic Acoustic BEM
- LMS SYSNOISE Aero-Acoustic Modelling

SYSNOISE is a license that ÅF have, but not all of the required parts.

3.3.1.3 Fluent + VNOISE (FWH+BEM)

This method is appropriate to use for problems with external flow. It is based on the same FWH theory as built into Fluent, but it adds a lot more acoustic simulations. For example it can use the source data created with the FWH method and use it in a BEM analysis. This creates more application areas such as noise radiated from fans in casings.

Requires:

- Fluent 6.2 or later
- VNoise

3.3.1.4 Fluent + Actran-LA (FEM)

This method is appropriate to use for external or internal flow. It does not neglect the quadropole term like the BEM technique.

Begin with exporting Lighthill's tensor and other variables from Fluent to Actran. Then perform the sound propagation computation.

Requires:

- Fluent 6.2 or later
- Actran-LA

3.3.1.5 OpenFOAM (CFD)

OpenFoam is an open source CFD solver. It is stated to be able to calculate vibrations induced by flow, and noise analysis. However, since the solver is open source, it contains no documentation or explanations of the methods used. If OpenFOAM is selected it can be used together with the open source mesh generator NETGEN. This program can import both STEP and IGES files. All tough this solver is free it is not compatible with any external acoustic solver and its usefulness is therefore very limited.

Requires:

- OpenFOAM

3.3.1.6 SIDLAB (Two-port theory)

SIDLAB is a very easy to use tool for low frequency sound in ducts that is based on the two port theory. However, it requires any CAD drawing to manually be transformed into a 1-D duct network and its usefulness is very limited and the sound sources are predefined. But for simple geometries and low frequency sound in ducts it is very easy to use. See figure below for example of a duct network:

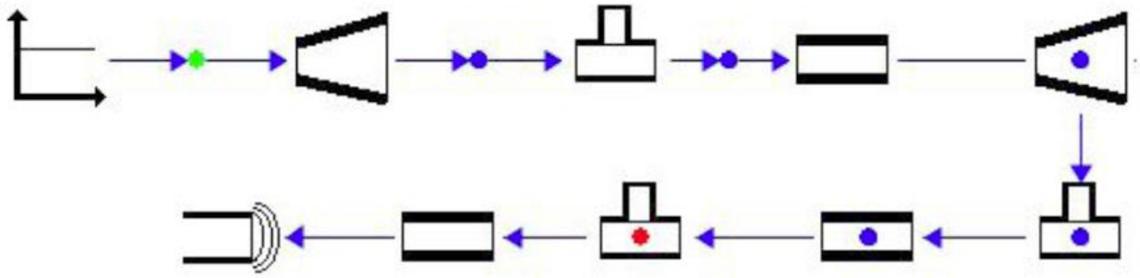

Figure 22. Duct network in SIDLAB.

SIDLAB can also be used to make two-port measurements. These measurements can then be compared to the simulated transmission loss:

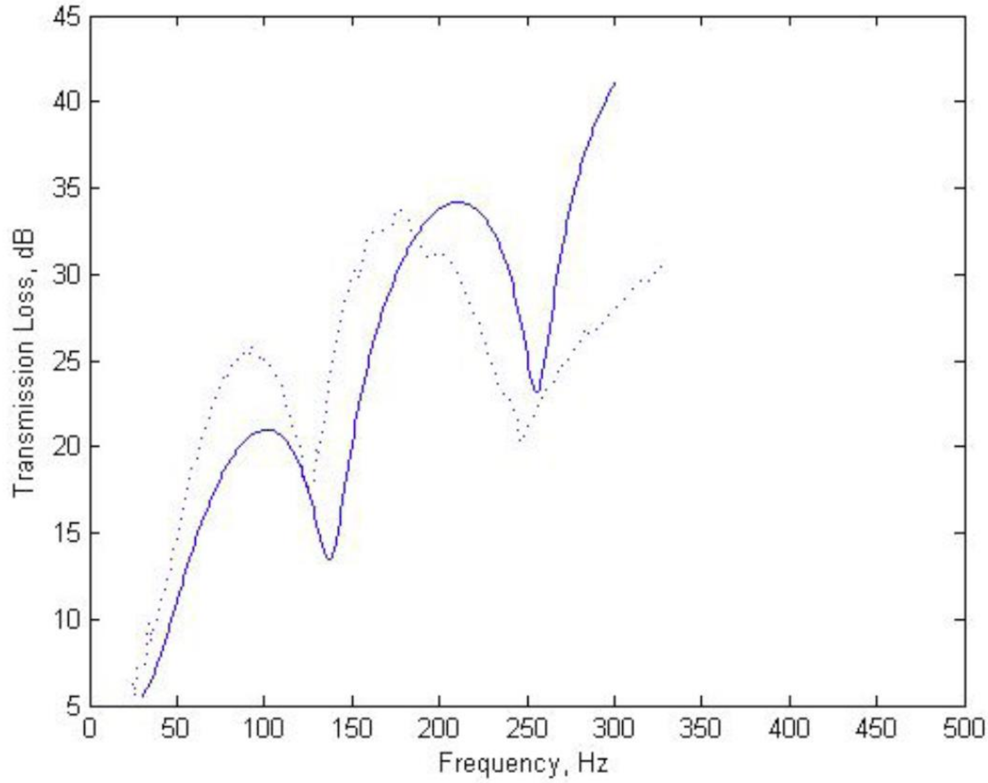

Figure 23. Simulation results (full line) compared to Measurements (dotted line) of a duct network obtained with SIDLAB.

Requires:

- SIDLAB

3.3.1.7 COMSOL Multiphysics

If the flow generated sound source is known and one wants to simulate the sound field or the "acoustic – structure coupling" based on this field; a program such as COMSOL Multiphysics can be used. Below is a visualization of a sound generating point source that influences a hollow cylinder. Other types of sources and setups can be chosen in order to best fit the current flow acoustic problem. This particular problem requires the acoustics and structural mechanics modules. The figure below shows displacement and sound pressure level for the cylinder and the surrounding region.

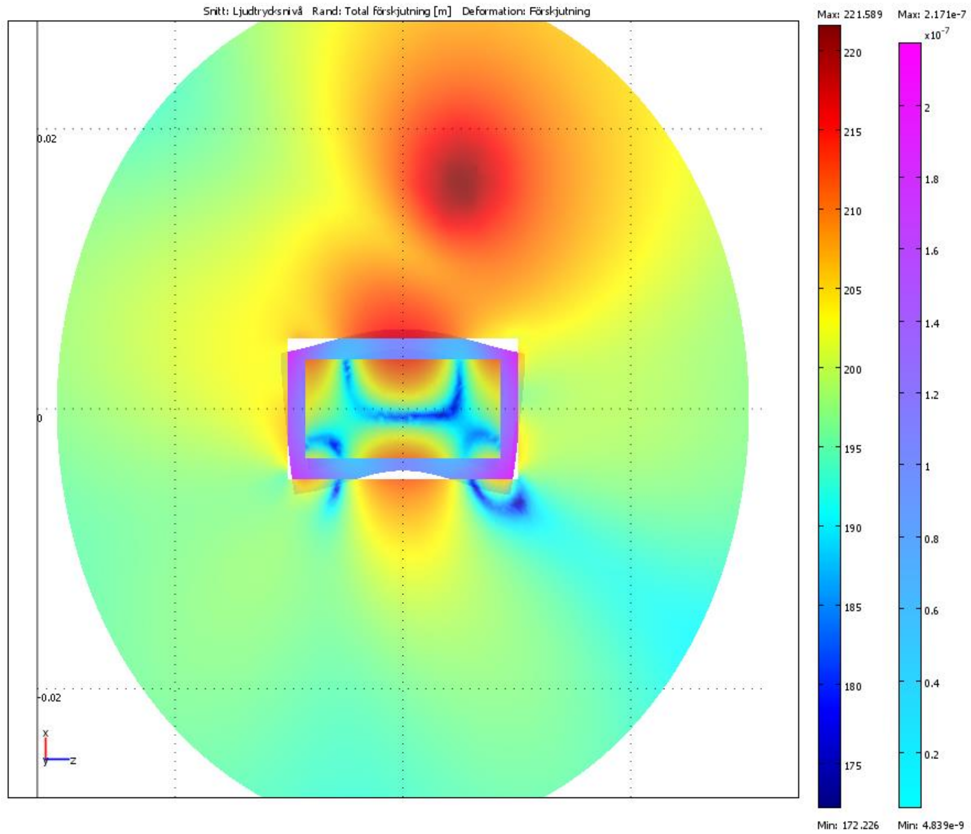

Figure 24. Sound pressure levels and displacement calculated from a point source, acting on a cylinder.

COMSOL Multiphysics can also be used to calculate the sound pressure levels and visualize how the acoustic field looks inside a pipe. Comsol cannot calculate how sound is generated from a flow, but it can calculate the sound propagation in a flow for a known source. Inlet velocity and boundary values must be set in agreement with the current flow acoustic problem. This problem requires the acoustics module.

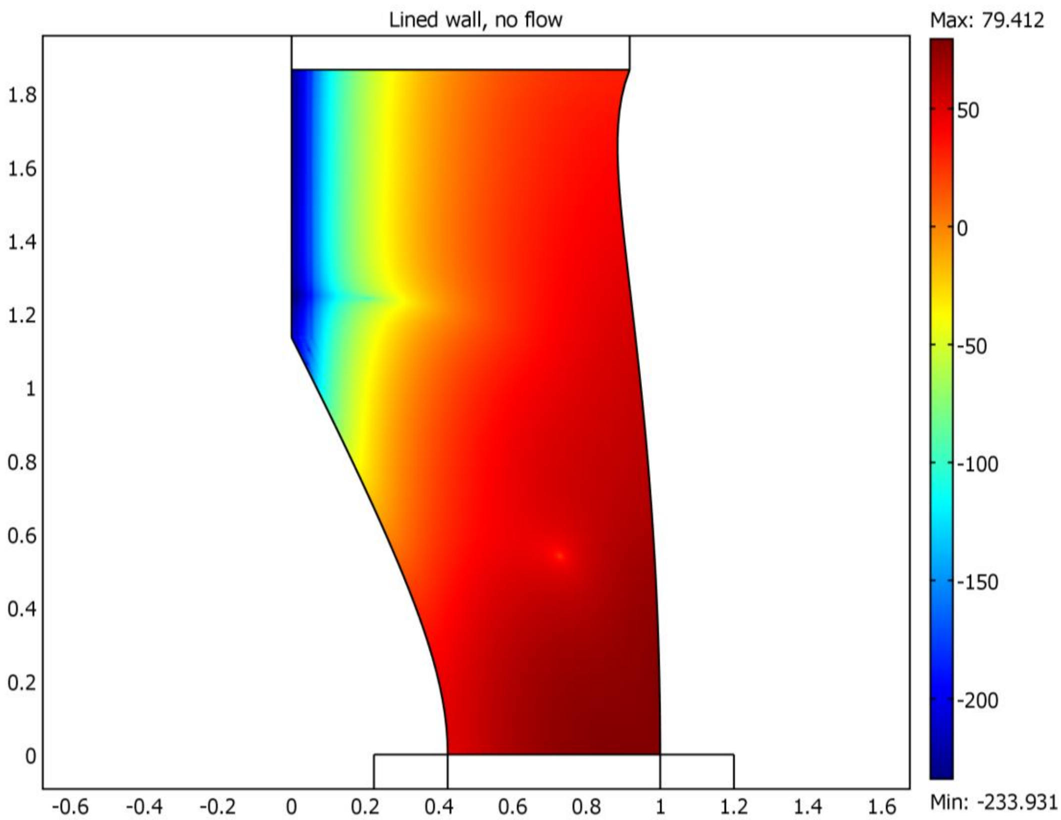

Figure 25. Sound pressure levels calculated from a moving source at the inlet.

COMSOL can also calculate the eigenfrequencies of an object. This can then be used in the acoustic-structure interaction calculation in order to see how the object behaves at these potentially important frequencies. This can be of use in flow acoustic problems where the sound field is generated by high velocity flows. This problem requires the structural mechanics module.

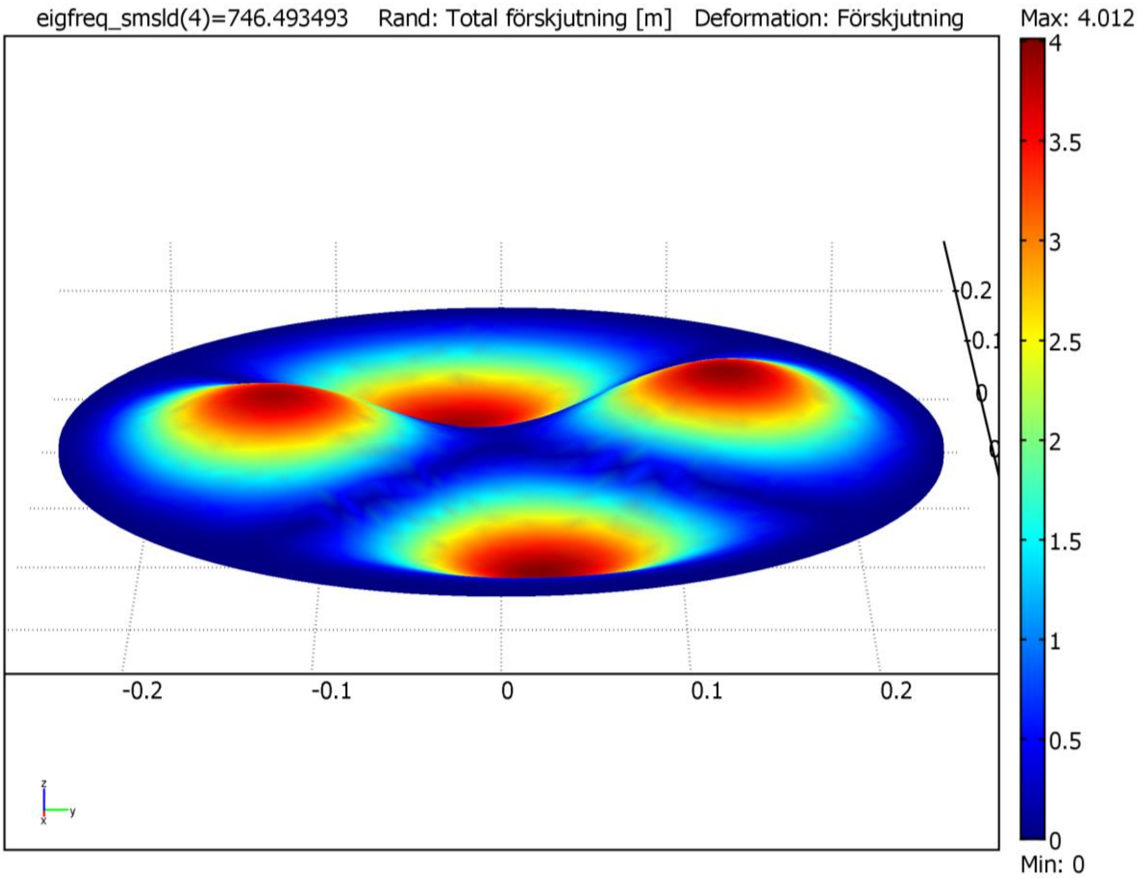

Figure 26. An eigenfrequency of an object.

COMSOL can import STEP, IGES and STL CAD formats which makes the geometry creation very simple.

Requires:

- COMSOL Multiphysics
- COMSOL Acoustics Module
- COMSOL Structural Mechanics Module (for acoustic-structure interaction)

3.4 Walk Through: Simulation of pipe flow using Fluent

3.4.1 Mesh creation using Gambit software

- Create geometry
- Mesh geometry
- Specify boundary types

3.4.2 Using Fluent FWH solver

1. **Calculate a converged flow solution**. A transient case should run until a “statistically steady state solution” is obtained. This means that all important flow variables are fully developed and that their statistics does not vary with time. To see if this is fulfilled, the major flow variables can be monitored at selected points.

2. **Activate the FWH acoustic model**. This is done by clicking define→models→acoustics. Select “Export Acoustic Source Data” if the data is to be saved or exported to another program such as SYSNOISE, or select “Compute Acoustic Signals Simultaneously” to use the built in FWH solver simultaneously.
   Set the model constants. All default values are air at atmospheric pressure and temperature. The “Compute Acoustic Signals Simultaneously” option performs a simultaneous calculation of the sound pressure levels at the receivers without writing to a data file, which saves a lot of disc space. At the same time it increases the computation time a little bit, but it also makes it possible to see when the signals have become statistically steady.

3. **Specify Source surfaces**. Click on “sources” to select surface sources. Then choose write frequency for the data export and the number of time steps per file. From the acoustic model panel, click “Sources”.
   If multiple source surfaces are selected, no source surface may enclose any of the other source surfaces. If you want to define an interior surface, you will have to define this in advance when meshing the grid. The two cell zones must have different cell-ID. When you select the interior surface in Fluent, a pop up box will appear that lets you specify which of the two zones that are the interior cell zone. This will help Fluent to let the sound propagate in the right direction.
   If a permeable surface is chosen as a source, other surfaces inside the permeable surface should not be chosen for acoustic calculation or they will be counted twice.

   To export sound source data without enabling the FWH model you will still have to define source sources. Also you can choose quadropole sources. To do this, use “export volumetric sources” to choose fluid zones as emissions sources. This also requires you to use “write centroid info”

4. **Saving source data**. The “write frequency” means how often the source data will be written. 1 is the default value and should be used unless the time step size in the transient simulation is smaller than required to calculate the frequency of interest.

"No of Time Steps per File" splits the data into different files. Each file will include source data for x time steps. This can be useful if some of the calculated frequencies have not reached a fully statistically stable state, you can still use the frequencies that did. Remember that saving source data can take up a lot of disc space.

5. **Specify receiver locations**. Click on "Receivers" and give their locations. It can be inside or outside the CFD domain. If you have chosen to export the acoustic data without computing the acoustic signals simultaneously, the receiver locations will not need to be specified.

6. **Continue the transient simulation**. Click Solve→Iterate. The acoustic source data are saved to .index and .asd files. The time step should be 1/(10*max frequency) and the simulation run time 10/(min frequency).

7. **Read .index and .asd files** to compute the acoustic signals (pressure vs. time). Start with loading the .index file, which updates the source data list. Then select the .asd files (source data) to be processed and the source zones (surfaces) to be used. Click on "Compute/Write" to calculate the acoustic signal for the receivers. The receiver data is saved in .ard files.

8. **Use the FFT tool** in order to transform the acoustic signal to the frequency spectrum. Click Plot→FFT. Choose "Power Spectral Density" or "Sound Pressure Level" on the Y-axis and "Frequency" or "1/3 Octave band" on the X-axis. Click on "Plot/Modify Input Signal" to modify the signal further.

3.4.3 Using Fluent Broadband noise source model (turbulence modelling)

1. **Calculate a steady or unsteady RANS solution.**
   Begin with importing the mesh that was created in Gambit. Do this by clicking File→Read→Case and choose your file. Then click Grid→Check to verify that there are no errors in the grid.
   Click Define→Models→Solver. Choose the options suitable for your specific case (defaults are pressure based solver, implicit formulation, steady flow and absolute velocity formulation).
   Click Define→Models→Viscous. Select your preferred model (in this case a broadband noise model). Choose for example the k-ε model. Select your desired wall model.
   Click Define→Models→Energy. If the flow is incompressible, the energy equation is decoupled from the momentum equations and the energy equation should then be unselected. The energy equation

should be selected when the temperature distribution is supposed to be calculated.
Click Define→Materials to setup the material parameters.
Click Define→Boundary conditions. This lets you define the required boundary conditions for the solution variables. Which boundary conditions you have to define depends on what model you have chosen. Click on inlet to define inlet conditions such as inlet velocity. If “Intensity and Hydraulic Diameter” is chosen as the Turbulence specification method, these values must be specified. For circular ducts the hydraulic diameter = inlet diameter of the duct. The turbulence intensity is between 5% and 20% for high-speed cases (such as flow in heat exchangers, turbines etc.), 1% to 5% in not so complex flows (like large pipes and ventilation flows etc) or low-speed flows, below 1% for flow originating from a fluid standing still (external flow across a car, submarine, aircraft). The turbulence intensity and hydraulic diameter is described further in the theory section.
Click Solve→Initialize→Initialize to give an initial guess for the solution variables. The final solution will be independent of this guess, but a good initial guess will reduce computational time. Click “Init” when done.
To start iterating click Solve→Iterate

The Spalart-Allmaras (low Re numbers) and the k-ω (high and low Re) models are designed to be used throughout any boundary layer as well. There are two methods to deal with the boundary layers near walls: “wall functions” and the “near wall model”. The “wall function” approach uses mathematical functions to model the flow region closest to the wall, while the “near wall model” creates a fine mesh all the way to the wall, including the viscous sub layers. The “wall function” approach saves computational time and is used for most high Reynolds number flows. For lower Reynolds number flows, the wall functions are not valid and the “near wall model” should be used. The figure below depicts the “near wall model” applied to a 2D pipe.

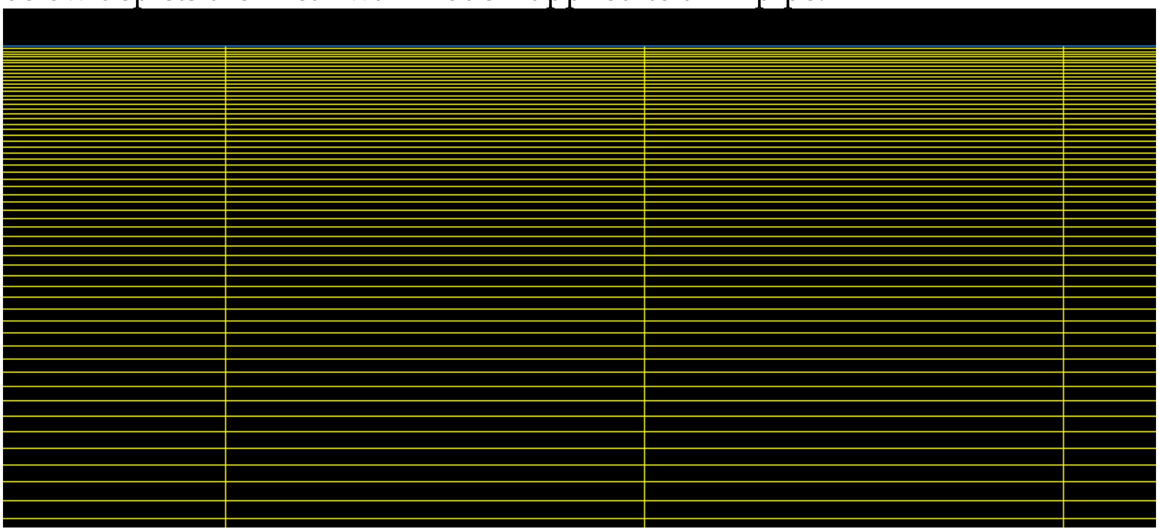

Figure 27. Near wall model used to mesh the boundary layer of a 2D pipe.

2. **Enable the broadband noise model and set up model parameters.** Click Define→Models→Acoustics. Choose "Broadband Noise Sources". Setup the acoustic parameters. "Number of Realizations" is the number of samples that are used to calculate the averaged source terms. "Number of Fourier Modes" is the number of modes that are used to calculate the turbulent velocity field and its derivatives.

3. **Post processing.** Click Display→Contours and choose among the available variables. The figure below shows contours of Sound Power Level generated in the pipe mentioned above. Areas marked with high values (red), creates a lot of sound, while areas with low values (blue) does not contribute to the sound generation.

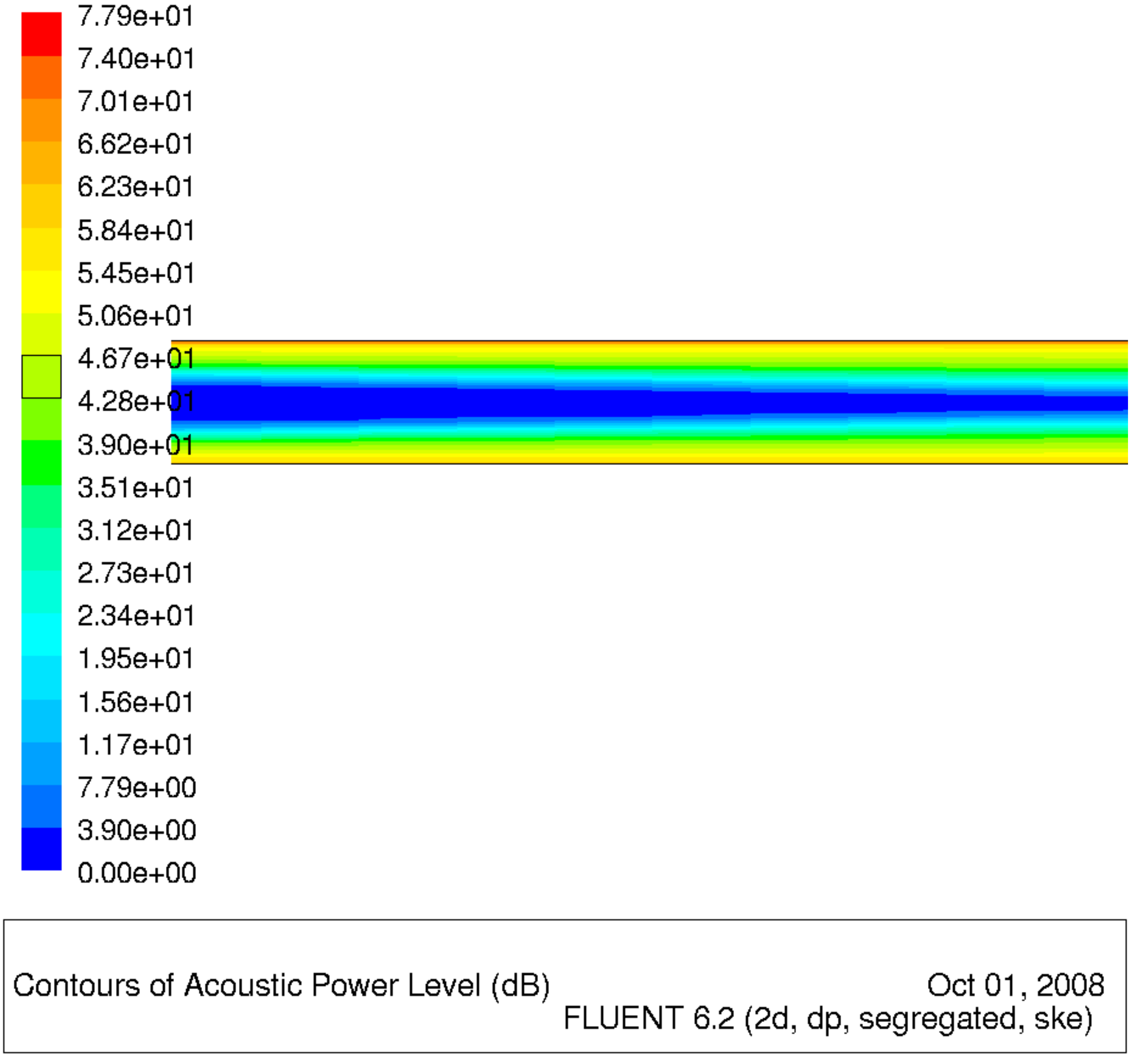

Figure 28. Contours of sound power level generated from turbulent flow in a pipe.

3.5 Walk Through: Fan simulation using FWH + BEM method

The difference between this and the previous example is that here the acoustic sound field can couple to the structure and then radiate sound towards an observer outside the duct. Therefore this tutorial includes the same steps made in Fluent but with some add-ons. Because of the usage of real acoustic solver software, this method is also a good way to simulate externally generated sound.

1. **CAD drawing**
   First of all you need a CAD drawing. If you plan to use Gambit for meshing, it supports all common formats (Parasolid, ACIS, STEP, IGES, and CATIA V4/V5. Drawings can be directly exported from among others, Pro/ENGINEER and SolidWorks. However, this requires a geometry conversion tool licence in gambit.

2. **Meshing**
   Gambit is used to mesh the CAD model and its surroundings.

3. **CFD solution**
   Perform the transient flow simulation in Fluent and save the data.

4. **FWH solution**
   Save the sound sources found with Fluent's FWH module.

5. **BEM analysis**
   Use an acoustic solver that can use the source data from Fluent, and perform the BEM analysis. Examples of solvers like this are VNoise and LMS SYSNOISE. Advantages and disadvantages of these programs can be found in section 4.2.

6. **Calculation of external radiation**
   The sound radiated from the structure is then calculated by the Acoustic solver program. Sound pressure levels can be displayed for different observer positions.

Design changes
If the main cause of the flow induced problem is found, a proposed solution should be added to the CAD drawing and the process starts over. This is an iterative process that will end when an adequate solution is found.

Actran-LA can also be used together with Fluent and uses FEM method instead of BEM. The proceedings using that software are similar to what is described above.

4 Case studies

This section aims to use the knowledge obtained from the previous sections to understand some specific cases that was being worked on at the consultancy company at the time.

4.1 Problem 1, Sound generation in steam pipes

4.1.1 Problem description

A steam generation plant had a problem with an unexplainable high pitch tonal noise.

4.1.2 Theory

The tonal noise was caused by a cavity in a valve that generated a type 2 whistle.

4.1.3 Visualization

To make a complete simulation of a pipe system like this, that includes whistling phenomena would take enormous computational resources in order to find the source of the sound. What one can do is calculations of how sound propagates from a given sound source in the duct. Alternatively, one can analyze smaller pieces of the system numerically and hopefully find a piece with high risk of resonance. If the sound instead had been of broadband character, the simulations would become a lot simpler and faster to perform.

4.1.4 Solution

Sound measurements were made and a strong peak was found at 2500 Hz with harmonics at 5000 Hz and 7500 Hz. The source of the sound could not be found by only measuring and listening.

By systematically calculating resonances for different parts in the piping system, a valve was found with a resonance frequency close to 2500 Hz. It is likely that flow separation occurred in the valve and increased in strength because of the resonance. When this valve was changed the generated tone disappeared. Furthermore, the steel beams in contact with the pipe have efficient radiation at 2500 Hz.

It is essential that the speed of sound is correctly determined from gas composition, temperature and pressure, in order to find the right eigenfrequencies in the acoustic cavities and resonators.

4.1.5 Summary of noise abatement measures for this and similar problems

- Resonance frequencies should be different from flow generated frequencies everywhere.

- No objects, big or small, should interrupt the flow in order to avoid flow separation and tonal noise.
- If vibrations are transmitted from the duct to the carrying structure; vibration isolating material can be added between the duct and the structure.
- If possible, a large duct radius with a low flow velocity should be used.
- Duct changes should be separated so that a normal flow is obtained before the flow reaches another duct change.
- Abrupt changes in the duct should be avoided as much as possible. Gradual changes are preferred.
- A noise shield can be mounted outside (or sometimes inside) the object in order to reduce high frequency noise.
- A CFD analysis for an entire duct system is computationally very demanding. However, a broadband noise model can be performed in order to find the turbulence generated noise sources. This analysis will not find tonal noise. (For further information see the simulation section)

4.2 Problem 2, Torch at refinery

4.2.1 Problem description

An oil refinery had a problem with flow induced noise from their torch. To put a stop to this problem, a new torch nozzle was ordered from a supplier, and ÅF-Ingemansson's job was to evaluate this new nozzle design from an acoustic perspective. The old nozzle is more or less just a straight pipe while the new nozzle splits the flow in many different pipes with a larger diameter in total. This reduces the flow speed.

A torch is a kind of safety valve that burns all excessive gas from the refinery. The flow in the original torch nozzle reaches speeds above 100 m/s. The gas that flows through the nozzle is known.

4.2.2 Theory

If the Mach number is low, the quadropole source term will not be as important as the monopole and dipole terms and radiated sound power depends on the velocity as U^6.

If the Mach number is higher but still below 1, the radiated sound is of quadropole type and the sound power is proportional to U^8. For this torch the flow velocity is above 100 m/s and the sound source can therefore probably be considered as a quadropole term when neglecting heat.

A problem in the analysis of the torch nozzle is that the sound increases a lot during operating disturbances. Another factor that can influence the results is the effect of varying combustion. This was not handled by the ÅF-Ingemansson report and would not be handled in a possible simulation part either. The simulation will only take care of the flow acoustic phenomena's in the torch and not the sound created by any other sources such as compressors etc. Inside the nozzle, time varying and unsteady fluid forces will act on the surfaces and create dipole source term sound. This sound is probably increased in the new nozzle design due to the branched outlet. Since this sound is created on the inside of the nozzle it can to a large extent be shielded with noise shields on the outside. The turbulence generated quadropole source terms at the nozzle outlet then remains the main source of sound.

If a simplified simulation, i.e. a broadband noise model, is applied it will only take into account the quadropole source term. If this is the dominating source of sound this is not a problem, but if monopole, and foremost, dipole sources are thought to be of the same order as the quadropole sound, then a different simulation method should be considered.

4.2.3 Visualization

An alternative/complementary approach is to make a simulation first in order to get an idea of how much sound the new nozzle will radiate, and where the sound is created. This approach can save the company a lot of money in that it can evaluate the radiated sound from the nozzle, and make it possible to make changes in its design before it has been constructed. Since the nozzle is quite large and the noise mostly consists of jet-noise, a broadband approach can be used. This would not be as detailed as a direct or integral approach, but it would be noticeably faster which is crucial when dealing with these large constructions. If necessary computing power and time is available then the integral approach should definitely be chosen.

Since the CAD-drawing of the nozzle and all necessary data of the gas was available, it is just a matter of starting with the mesh before the CFD analysis can be run. To be able to import the CAD file to the meshing software (Gambit), it must have geometry conversion tool installed. If this function is unavailable, as it was on Chalmers, the drawing must be sketched up again in the meshing software. The simulation would not be able to give sound pressure levels at a receiver location, but it could be used to compare different nozzle designs to see which one is the most efficient at reducing the noise. In this case, the aim of the analysis would be to compare the old nozzle design to the new.

A simplified version of this comparison can be made using a software named ENC. It uses Hydraulic Diameter and Turbulence intensity to calculate radiated sound from a jet or furnace.

4.2.4 Solution made by ÅF-Ingemansson

Noise measurements of the torch nozzle were made to see if it reduces the noise as much as the supplier stated. Measurements were taken at 80 m and then recalculated to 1000 m, which is the distance that had sound pressure levels stated by the supplier. The comparison was made at a gas flow of 7000 kg/h. ÅF-Ingemansson measurements show good agreement with the values stated by the supplier. All measurements were taken in the "downstream"/tailwind direction of the torch which gives the most fair results. The flow data were given from the refinery.

The subjective judgement was that the torch nozzle is quiet.

The solution made by the supplier is based on the idea that it is the difference in velocity between the outlet gas and the surrounding air that creates the noise through a shear phenomenon. Therefore they created a nozzle that has a larger outlet surface. This lowers the speed of the gas and therefore also the generated sound, especially at low frequencies. The sound increases somewhat for higher frequencies but they are damped by an external sound insulation, also created by the nozzle supplier, and high frequencies are also damped more easily over large distances. The construction also separates the flow into separate channels with outlets close to each other in order to reduce the difference in speed between the outlet gas and the surrounding air. This lowers the noise a lot since the sound is proportional to U^8 for these flow velocities. The flow varied between 2000 and 11000 kg/h during the measurement. The sound pressure level increased more or less linearly with the flow speed. Below 5000kg/h, the sound radiated to the surroundings is very low. Besides the flow speed, the combustion should be kept at a minimum since large combustion leads to very high sound pressure levels.

For more information about the underlying theory of this and similar problems see "Subsonic jet oscillations" and "Quadropole source term" in the theory section.

4.2.5 Summary of noise abatement measures for this and similar high velocity flow noise problems

- As mentioned above the pipe diameter should be as large as possible and the speed reduced as much as possible.
- The diameter of the pipes should preferably not decrease in the flow direction.

- Noise generated by high velocity outlet flow can be reduced by letting a part of the flow go outside the main flow with a lower flow velocity, thus making the velocity difference between the outlet flow and the surrounding air smaller.

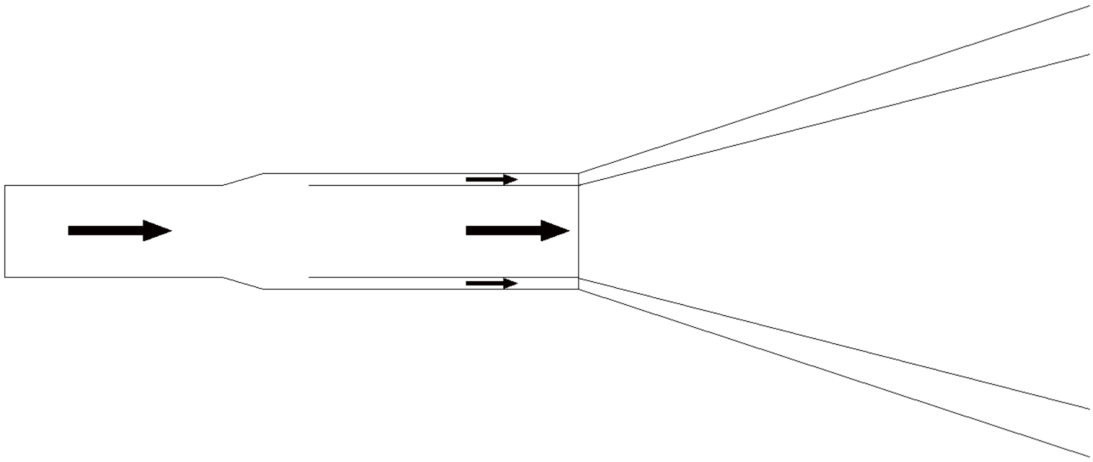

Figure 29. Low velocity flow outside main flow.

- A noise shield can be mounted outside (or sometimes inside) the object in order to reduce high frequency noise.
- A CFD analysis can point out other details in the design that generate sound. (For further information see the simulation section)

4.3 Problem 3, Water piping system at paper mill

4.3.1 Problem description

A paper mill in Blackburn made by Alstom Power had a problem with a strong, unidentified tonal noise generated somewhere in the piping system of the mill. The piping system contains moving water. The tonal noise could be heard everywhere near the piping system.

4.3.2 Theory

The source of the sound was a flow meter mounted inside the piping system. This flow meter generated a periodic flow separation and a type 1 whistle that created a dipole source term with a specific Strouhal frequency.

The sound radiated from the source through the water and coupled to the wall of the duct. The vibrating walls then couple to the outside air and the carrying structure. The flow meter itself can also begin to vibrate and in turn cause the duct to vibrate.

4.3.3 Visualization

See 5.1.3.

4.3.4 Solution made by ÅF-Ingemansson

Sound pressure measurements where in the year 2002 carried out on many locations outside the piping system but since the tonal noise was spread all over the system the source could not be found. Measurements and calculations were conducted according to BS5228: Part 4 and BS 4142:1997.

Through a trial and error approach, several parts of the piping system (such as valves) that was thought to be cause of the tonal noise was changed or removed. However none of these modifications solved the problem.

Analyses of the fluid flow were also conducted but nothing that generates tonal noise could be found. This approach led to some changes in the piping system but none of them removed the noise.

(Other noise abatement methods were also performed.)

Eventually the flow meter that was mounted inside the pipe was found to be the source of the sound. This flow meter created a disturbance in the flow which generated a Strouhal frequency (dipole source term) by flow separation. When this flow meter was removed the tonal noise disappeared. In order to avoid this tonal noise and still being able to measure the flow, one could instead install a flow meter that does not affect the flow. One example of this is an ultrasonic flow meter that measures the volume flow without being in contact with the flow. This flow meter is mounted outside the pipe instead of being inserted directly into the flow. It thereby prevents the source of the problem, namely the periodic flow separation, to occur.

For more information about the underlying theory of this and similar problems see “type 1 whistle”, “dipole source term” or “sound in ducts” in the theory section.

4.3.5 Summary of noise abatement measures for this and similar problems

- If a flow meter of this type must be used it should be stiffened and have it geometry changed to avoid the “type 1 whistle phenomena”. Irregularities can be put on the object or it can be stiffened.

See 5.1.5 for more noise abatement measures.

5 Conclusion

To be able to perform simulations of flow generated sound, Fluent together with the meshing tool "Gambit", with geometry converter, seems to be the essential software to use. Of course there are other alternative CFD solvers but none of them is as versatile or compatible with external acoustic solvers as Fluent is. Furthermore it is a licence that ÅF already owns, and knowledge of how it works is therefore available inside the company. For more advanced problems, a licence for a Fluent- compatible acoustic solver such as SYSNOISE should be acquired. This combination can solve a number of practical problems such as sound in ducts with sound radiation, fans, turbines, sound from rear view mirrors etc. If a flow acoustic problem would arise, the CFD expertise from other departments together with the acoustic expertise from Ingemansson could successfully simulate and solve the problem. Since it is quite time consuming for a beginner to get into these kinds of simulations, they might be best suited for larger, ongoing projects.

For simulations of larger duct systems, mufflers, or other low velocity flows where the sound source is known, a program such as SIDLAB can be used. It is not computationally demanding and it is comparably cheap. The program is quite easy to learn, but it takes some experience using the program in order to draw correct duct systems and perform accurate simulations. SIDLAB can also be used to compare two-port measurements with simulation results.

Flow acoustic simulations are viable today, but they will have an even more important role in the future (see next section).

6 Future Work

In the future when simulations using the direct approach is easy to perform on a regular cluster of computers, this option should definitely be looked into. It will probably be a lot easier to use and to understand than the hybrid approaches of today.

With the expected increased computational power of the future, larger objects and duct systems can be simulated and with increased accuracy.

7 References

[1] Tritton, DJ. (1988): *Physical fluid dynamics*, Second edition, Oxford University Press, New York, {United States}.

[2] Anderson, J. (2004): *Modern compressible flow*, third edition. McGraw-Hill Education, New York, {United States}.

[3] Åbom, M. (2008): *An introduction to flow acoustics*. KTH Aeronautical and Vehicle Engineering, Stockholm, {Sweden}.

[4] Lighthill, J. (1978): Waves in fluids. Camebridge university press, Cambridge, {United Kingdom}.

[5] Fluent Inc: *Fluent 6.2 Users guide*, Fluent Inc, {United States}, {2005}.

[6] Ducret, F. (2006): *Studies of sound generation and propagation in flow ducts*. {Licentiate Thesis} Department of Aeronautical and vehicle engineering, The Royal institute of technology (KTH), Stockholm, {Sweden}.

[7] Gallez, X. Caro, S. Ploumhans, P. Morgenthaler, V. Mathey, F. Ciapa, G. Ma, J: *Identification of the appropriate parameters for accurate CAA*. Presented at 11th AIAA/CEAS Aeroacoustics Conference, Montery CA, USA, 23-25 May 2005.

[8] Von Karman Institute. (2006): *Aeroacoustics*. Von Karman Institute for Fluid Dynamics, Rhode-St-Genèse, {Belgium}.

[9] Kleiner, M. (2005): *Audioteknik och akustik*, eighth edition. Institutionen för teknisk akustik, Chalmers, Göteborg, {Sweden}.

[10]Larsson, J. (2002): *Computational aero acoustics for vehicle applications*. {Licentiate Thesis}. Departement of Thermo and Fluid Dynamics, Chalmers University of Technology, Göteborg, {Sweden}.

[11]Manning, T A. Lele, S K. (2000): *A numerical investigation of sound generation in supersonic jet screech*, Departement of Aeronautics and Astronautics, Stanford University, California, {United States}.

[12]Nordling, C. Österman, J. (2002): *Physics handbook*, sixth edition. Studentlitteratur, Lund, {Sweden}.

[13]Råde, L. Westergren, B. (2004): *Mathematics handbook*, fifth edition. Studentlitteratur, Lund, {Sweden}.

[14] Sovani, S. (2005): *Aeroacoustics modeling*. Fluent Inc., Ann Arbor, MI, {United States}.

[15] Tam, CKW. (2001): *Computational aeroacoustics: an overview*. Departement of Mathematics, Florida State University, Tallahassee, FL, {United States}.

Appendix

Pipe Resonance Frequencies

The strong tones created by flow phenomena can be increased if they are coupled to an acoustic mode of the pipe.

For a **half wave resonator**, resonance occurs for:

$$f = \frac{nc}{2L} \qquad (37)$$

where:

$n = 1, 2, 3, \ldots$

L is the length of the pipe.

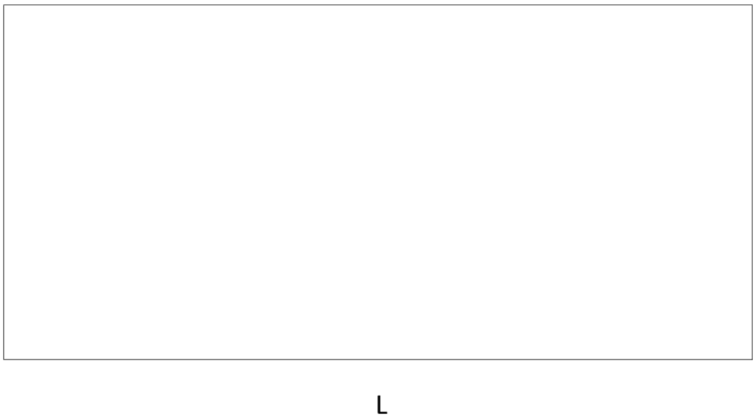

Figure A1. Half wave resonator

For a quarter wave resonator, resonance occurs for:

$$f_n = \frac{(2n-1)\cdot c}{4L} \qquad (38)$$

Figure A2. Quarter wave resonator

For a **Helmholtz resonator**, resonance occurs at:

$$f_r = \frac{c}{2\pi}\sqrt{\frac{A}{LV}} \qquad (39)$$

Where:

L is the length of the throat

V is the resonator volume

A is the throat area

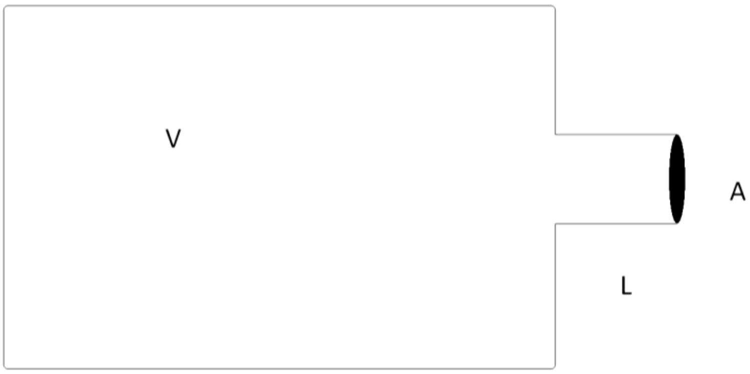

Figure A3. Helmholtz resonator

Beam Resonance Frequencies

Tones can also be increased if they are coupled to a structural mode.

A **console beam** has a bending wave resonance at:

$$f_n = \frac{\pi}{2L^2}\sqrt{\frac{E\kappa^2}{\rho}}\beta_n^2 \qquad (40)$$

Where:

L is the length of the beam

β_1 = 0.597, β_2 = 1.494, β_3 = 2.500, $\beta_n \approx n$-0.5 for $n \geq 3$

κ is the inertial radius of the cross section =a/2 for a circular cross section with radius a.

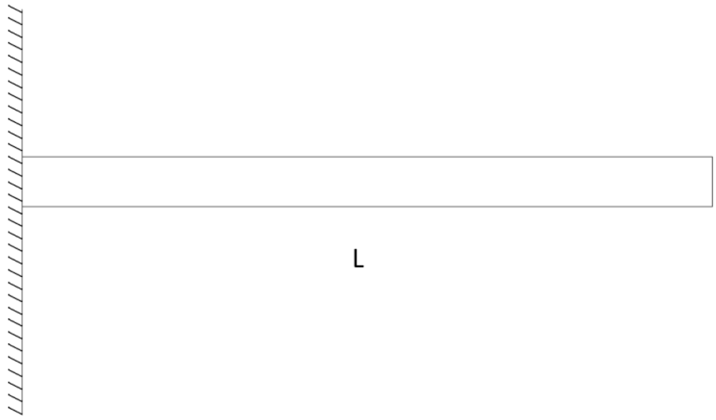

Figure A4. Console beam

A **simply supported beam** has a bending wave resonance at:

$$f_n = \frac{\pi}{2L^2}\sqrt{\frac{E\kappa^2}{\rho}}n_n^2 \qquad (41)$$

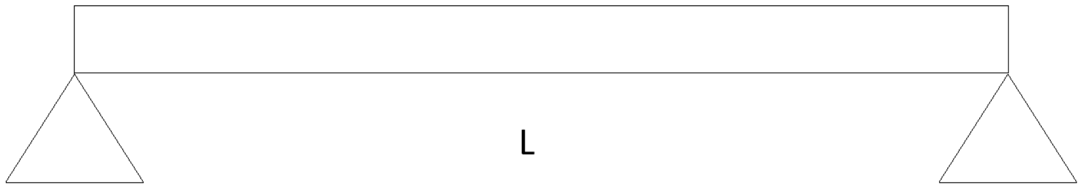

Figure A5. Simply supported beam

A **beam that is free in both ends or fixed in both ends** has a bending wave resonance at:

$$f_n = \frac{\pi}{2L^2}\sqrt{\frac{E\kappa^2}{\rho}}\beta_n^2 \qquad (42)$$

Where β_1 = 1.5056, β_2 = 2.4997, $\beta_n \approx n$-0.5 for $n \geq 2$

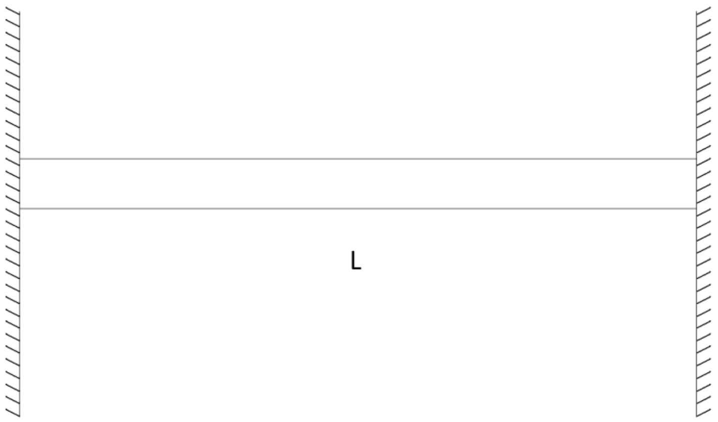

Figure A6. beam fixed in both ends

A **cylindrical pipe** has shell resonances at:

$$f_R = \frac{c_L}{2\pi a} \qquad (43)$$

Where:

$c_L = \sqrt{\frac{E}{\rho(1-\nu^2)}}$ is the longitudinal wave speed in a plate.

a is the average radius

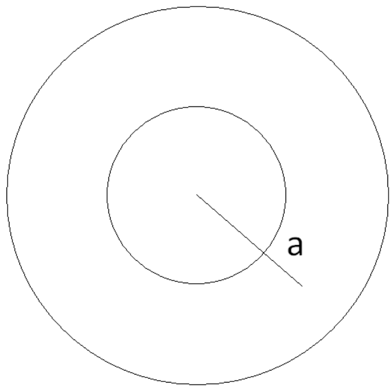

Figure A7. Cylindrical pipe