Flow-induced Sound Radiation from Air-ducting Structures

S. C. Beck[1], S. Langer[2]

[1] TU Clausthal, Institut für Technische Mechanik, Germany, Email: s.beck@tu-bs.de

[2] TU Braunschweig, Institut für Angewandte Mechanik, Germany, Email: s.langer@tu-bs.de

1 Introduction

Sound is a constant companion in daily life. If sound is considered disturbing it is then identified as "noise". The reduction of noise has become more and more an important topic when developing products of almost any kind.

Here, the topic of research is air-ducting structures being part of pipe systems. Pipe systems, in general, can comprise the transport of liquids as well as gases. Examples include heating and sanitary ducts or ventilation channels and air intakes, respectively.

For a number of pipe systems the driving unit, e. g. the motor, ventilator or pump, has been identified as the major source of sound emission. Sound emitted from there does not only radiate directly into the surrounding space, but it also travels within the connected pipes. Another source of sound is the generation of sound due to turbulence within the flow field.

Flow through a pipe-like structure causes the structure to vibrate and as a result it radiates sound into the surrounding space.

2 Motivation

Today, suppliers in the automobile industry need to meet new demands: while they have been merely manufactures in the past, suppliers are now actively developing parts, that concludes taking the responsibility that the items fulfil the requested requirements.

In the first place, air-ducting parts in automobile industry need to meet the technical requirements as a transport medium – flow rate, pressure drop, durability.

Suppliers now also thrive to optimize their products acoustically due to the increasing aspect of sound reduction as well as to secure a competitive advantage.

Concerning air-ducting parts the supplier generally does not have any influence on the driving unit nor the flow rate. He is limited to choosing the material and the shape of the structure within close boundaries.

To optimize a product's acoustic properties within these boundaries, this research project investigates the sound radiation of a flow-induced vibrating duct using numerical simulation.

3 Problem breakdown

Figure 1 gives a schematic description of the problem. Air is flowing through a pipe structure. In general, the flow rate is constant with a Reynolds number indicating a turbulent flow. The flow is acting on the structure causing it to deform with time, i. e., to vibrate. The structure interacts with the surrounding space – normally air. Recurring surface deformations result in compression waves in the air which

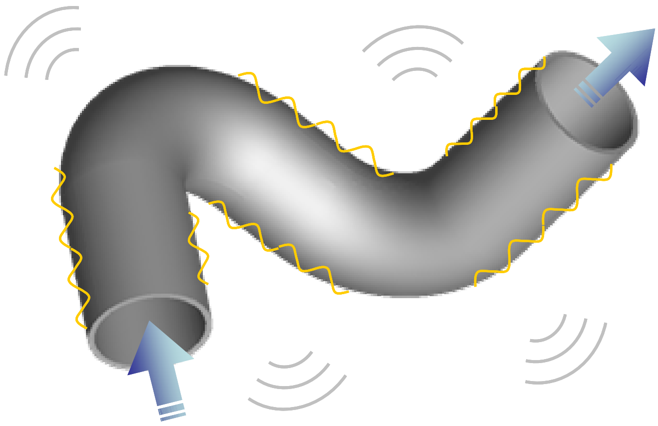

Figure 1: Schematic problem description: Air-flow propagation through a pipe causes vibration of the structure resulting in acoustical radiation into surrounding space.

are noticeable as sound if the frequency is within the audible range.

The flow's interaction with the structure is due to two main aspects:

1. The shape of the pipe (e. g. arches, changes in cross section) causes turbulence in the flow such as vortices and back flow. These disturbances can stretch over considerable lengths in flow direction and substantial proportions of the cross section (large-scale turbulence).
2. In non-laminar flow a turbulent boundary layer develops at the interface between flow and structure. Within the turbulent boundary layer perturbation is present acting as pressure fluctuations on the structure (small-scale turbulence).

4 Model

To model the aforementioned problem three different media and their properties need to be taken into account.

The internal flow (air) is modelled as an incompressible fluid. This assumption is reasonable since the flow velocities in the applications will not violate the criterion of Mach numbers being below 0.3.

The numerical simulation of the flow in the pipe is done by the commercial computational fluid dynamics (CFD) software ANSYS CFX which makes use of the Reynolds Averaged Navier-Stokes equations in a Volume-of-Fluid method to solve the simulation task. The turbulence is modelled applying the shear-stress-transport model which is

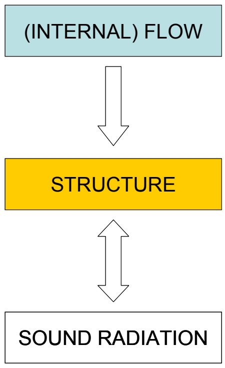

Figure 2: Modelled media and their interaction.

recommended for highly accurate boundary layer depiction or a standard k-epsilon model for large-scale turbulence modelling. [1] contains more details. .

The structure is described by a Mindlin shell model.

$$B\Delta\Delta w = f - \frac{h^2}{6k_s(1-\upsilon)}\Delta f \tag{1}$$

In Equation (1), B denotes the flexural stiffness ($B = EI / (1-v)$) with v being Poisson's ratio. w is the plate's out-of-plane deflection, h describes the plate's thickness, k_s the shear correction factor and f the force perpendicular to the plate.

The Finite-Element-Method (FEM) is adopted to simulate the structural behaviour. FEM is implemented in the in-house code of the Institut für Angewandte Mechanik, TU Braunschweig. Despite the code's capabilities to account for a variety of material laws, the shell here is considered to be purely linear-elastic (Hooke's Law).

Internal flow and structure interact by a weak coupling. It is assumed that deflections of the structure do not change the wall pressure field. This assumption is generally admitted for this type of problems; see [2] and [3]. As such, the pressure resulting from the flow simulation acts as loads on the structure.

$$f_s = \int_A p_f \, dA \tag{2}$$

In Equation (2), f represents forces and p pressure. A stands for the surface area. Indices s and f denote structural and fluid affiliation, respectively.

As the CFD simulation is carried out in the time domain and the structure is modelled in the frequency domain, a Fast Fourier Transformation (FFT) is used to convert time step data into frequency values. FFT is performed with the freely available software package fftw [4].

In the surrounding space both the Helmholtz-Equation (3) for an ideal compressible fluid and Sommerfeld's radiation condition (4) need to be fulfilled. Using Green's functions the boundary integral equation (BIE) is derived. The boundary is discretized and the BIE can be approximated using the Boundary-Element-Method (BEM). Like FEM, BEM is also implemented in the in-house code mentioned above.

$$p_{,ii} + \kappa^2 p = 0 \tag{3}$$

$$\frac{\partial p}{\partial n} = i\kappa p + O\left(\frac{1}{r}\right) \quad \text{for } r \to \infty \tag{4}$$

In Equations (3) and (4), κ represents the wave number.

Coupling between structure and sound radiation is of the strong type, i. e. the deflections of the structure accelerate the fluid and the pressure of the fluid acts as forces on the structure.

$$f_s = \int_A p_f \, dA \quad \text{and} \quad q_f = \frac{\partial p_f}{\partial n} = -i\rho_f \omega^2 w_s \tag{5}$$

In Equation (5), q_f denotes a flow in normal direction. ρ_f is the fluid's density and ω the angular frequency.

5 Examples

The first example shows coupling of CFD and FEM considering large-scale turbulence. Given is an aluminium duct with a rectangular cross section of 0.2 m by 0.1 m and a 45° change in direction. The material is considered linear-elastic and is 0.1 mm thick. The duct is excited by an air flow of 50 m/s.

Averaging the displacements of the duct over the whole surface shows that certain frequencies have a strong response to the excitation (see Figure 3). Figure 4 shows the vibration pattern for the most excited mode at 77 Hz.

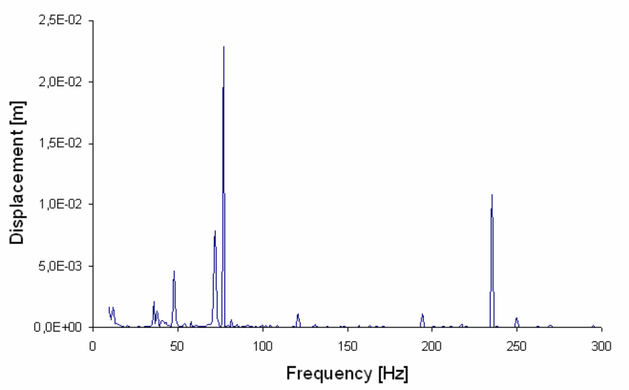

Figure 3: Average displacement of the duct evaluated in the frequency range (resulting from 50 m/s air flow excitation).

The second example shows another CFD-FEM coupling, considering excitement by a turbulent boundary layer (TBL, small-scale turbulence). A flow of 44.7 m/s propagates over a steel plate with the dimensions 0.47 m by 0.37 m and a thickness of 0.016 m. The plate is clamped along all edges and linear-elastic material behaviour is assumed. The set-up of this numerical example is chosen to resemble the experimental set-up of [5].

As in the example before, excitation differs over the frequency range (Figure 5). An example for the plate's displacements is shown in Figure 6.

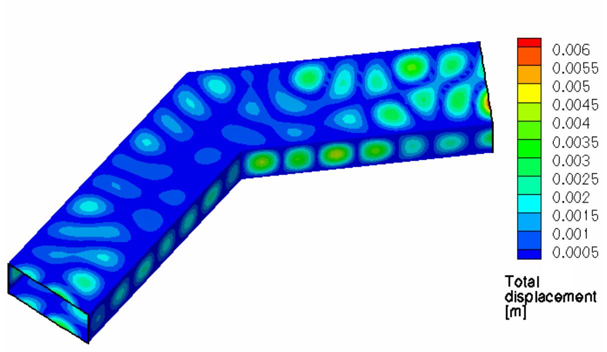

Figure 4: Displacement of the duct for most responsive mode (77 Hz).

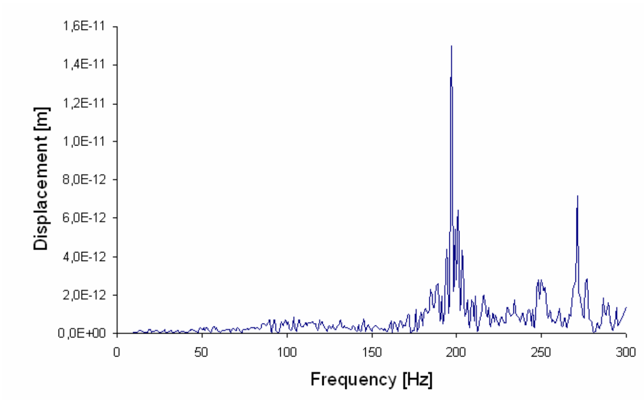

Figure 5: Average displacement of the plate due to turbulent boundary layer excitement.

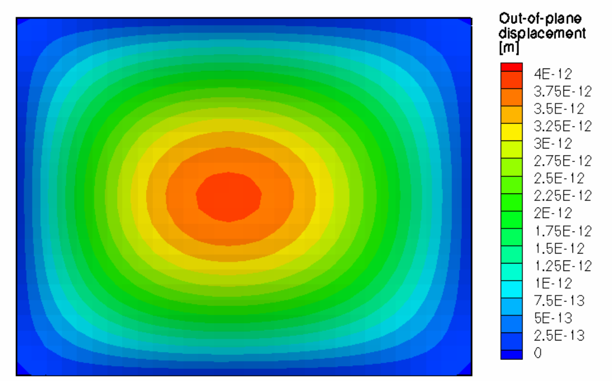

Figure 6: Displacement of the steel plate at resonant mode (197 Hz).

6 Outlook and further research

The most apparent task in the future will be to simulate completely coupled CFD-FEM-BEM examples.

Another topic of further research will focus on investigations concerning the modelling of the turbulent boundary layer. This will include comparison of own simulated TBL excitation with experimental results and TBL descriptions of various authors. In addition, the influence of the radiated sound due to TBL excitation on the overall sound radiation will be evaluated.

It is further planned to carry out experiments in cooperation with Physikalisch Technische Bundesanstalt (PTB) in Braunschweig. The results should verify the presented approach.

In the long run it is intended to predict the sound radiation of air-ducting parts for mass production and to show the optimization potential in terms of geometry and/or material modifications.

References

[1] ANSYS CFX-Solver Modelling Guide. Release 11.0

[2] Durant et al., Journal of Sound and Vibration (2000), **229**(5), 1115-1155

[3] Maury et al., Journal of Sound and Vibration (2002), **252**(1), 83-113

[4] URL: `http://www.fftw.org`

[5] Han et al., Journal of Sound and Vibration (1999), **227**(4), 685-709