Simulations of sound propagation at a duct termination with flow

Parinaz Sedarati
Abstract

Both theoretically as well as in many applications like exhaust systems, ventilation pipes, mufflers, air intakes and also large scale industrial smoke stacks, it is of interest to understand how sound waves are scattered at duct exists. Especially for aeronautical application such as jet engines, the effect of coupling of acoustics and flow on sound radiation and reflection from a duct termination in a uniform is an important problem. In order to predict the acoustic performance in duct systems, it is essential to know how the incoming acoustic waves are propagated and transmitted and reflected.

This thesis work aims at developing suitable simulation methods as extension to existing software and to validate these methods to experimental measurements and theory.

Firstly, numerical simulations of fully developed flow through a duct exit has been carried out. The goal in this part is to obtain the mean values for the velocity and pressure. The commercial code Fluent 12.1 is used for numerical simulations in two space dimensions.

Secondly, numerical simulations of the acoustic part has been studied with the commercial software Comsol 3.5a with the objective to investigate the ability of the frequency domain Navier-Stokes equations to the characteristic properties of the acoustics at the duct termination.

Finally, numerical results are compared to available experimental results with acceptable agreement which shows successes and also constraints of the simulations.
Acknowledgments

I am grateful to my supervisors Assoc. Prof. Gunilla Efraimsson and Dr. Axel Kierkegaard. It has been pleasure to learn acoustics and to work for my master thesis with them. I specially appreciate Axel for all his guidance, sincere dedication and critical comments during this time.

I would also thank Dr. Susann Boij for her useful discussions as well as to all my friends and PhD students at the Marcus Wallenberg Laboratory (MWL), special thanks to Tristan Favre for spending his time and discussion about flow simulations and also Chenyang Weng for all his comments during this years.

Last but not least, I would like to show my gratitude to my family for their support and encouragement along these years.
Contents

Nomenclature 4

1 Introduction 7

2 Flow simulation 9
  2.1 Problem definition ................................................. 9
  2.2 Flow equations ..................................................... 11
  2.3 Turbulent .......................................................... 12
     2.3.1 Modeling of turbulence - Reynolds averaged Navier-Stokes ........................................... 12
     2.3.1.1 Near wall region and Wall Functions ............... 13

3 Simulation of the incompressible flow 15
  3.1 Boundary conditions ............................................... 15
  3.2 Mesh .............................................................. 17
     3.2.1 Determination of duct length and outer domain ........ 18
  3.3 Discussion ........................................................ 19

4 Acoustics 23
  4.1 Acoustic wave motion and plane waves ......................... 23
     4.1.1 Non-dimensional numbers .................................. 23
  4.2 Frequency scaling .................................................. 24
<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>4.3</td>
<td>Linearized Navier-Stokes equations</td>
<td>28</td>
</tr>
<tr>
<td>4.4</td>
<td>Acoustic boundary conditions</td>
<td>29</td>
</tr>
<tr>
<td>4.4.1</td>
<td>Buffer zones</td>
<td>30</td>
</tr>
<tr>
<td>4.5</td>
<td>Model constants and variables</td>
<td>31</td>
</tr>
<tr>
<td>4.5.1</td>
<td>Discussion</td>
<td>32</td>
</tr>
<tr>
<td>4.5.1.1</td>
<td>Effect of refining the mesh</td>
<td>35</td>
</tr>
<tr>
<td>4.5.1.2</td>
<td>Investigation of buffer zone area effect on the reflection coefficient</td>
<td>37</td>
</tr>
<tr>
<td>4.6</td>
<td>Summary</td>
<td>40</td>
</tr>
<tr>
<td></td>
<td>Bibliography</td>
<td>42</td>
</tr>
<tr>
<td></td>
<td>Appendix</td>
<td>44</td>
</tr>
</tbody>
</table>
Nomenclature

\[ \delta \quad \text{Kronecker delta function} \]
\[ \eta \quad \text{Area aspect ratio [-]} \]
\[ \frac{D}{Dt} \quad \text{Convective derivative} \]
\[ \gamma \quad \text{Heat capacity ratio [-]} \]
\[ \hat{p} \quad \text{Acoustic pressure [Pa]} \]
\[ \hat{u} \quad \text{Acoustic velocity [m/s]} \]
\[ \kappa \quad \text{Thermal conductivity [W/(m-K)]} \]
\[ \kappa_0 \quad \text{Cut on Helmholtz number for cylindrical duct [-]} \]
\[ \mu \quad \text{Dynamic viscosity [Pa-s]} \]
\[ \nabla \quad \text{Nabla operator} \]
\[ \nu \quad \text{Kinematic viscosity [m}^2\text{/s]} \]
\[ \omega \quad \text{Angular frequency [rad/s]} \]
\[ \phi \quad \text{Dissipation function} \]
\[ \rho \quad \text{Density [kg/m}^3\text{]} \]
\[ \rho' \quad \text{Time varying density perturbation [kg/m}^3\text{]} \]
\[ \tau_w \quad \text{Wall shear stress [N/m}^2\text{]} \]
\[ \tau_{ij} \quad \text{Viscous stress tensor [N/m}^2\text{]} \]
\[ B \quad \text{Height of the outer domain [m]} \]
\[ C \quad \text{Length of the outer domain [m]} \]
$c_0$  Speed of sound [m/s]
$c_{ph}$  Phase velocity [m/s]
$D$  Outer cylindrical diameter [m]
$d$  Inner cylindrical diameter [m]
$e$  Internal energy [J]
$f$  Frequency [Hz]
$f^c$  Cut-on frequency [Hz]
$F_i$  Volume force in the $i$ : $th$ direction [N]
$H$  Outer diameter in 2D duct [m]
$h$  Inner diameter in 2D duct [m]
$He$  Helmholtz number [-]
$He^*$  Normalized Helmholtz number [-]
$I$  Turbulent intensity [-]
$k_{2d}$  Wave number in 2D duct [1/m]
$k_{cyl}$  Wave number for cylindrical duct [1/m]
$L$  Duct length [m]
$l$  Turbulent length scale [m]
$M$  Mach number [-]
$n$  Unit vector[-]
$p'$  Time varying pressure perturbation [Pa]
$p_0$  Mean pressure [Pa]
$R$  Reflection coefficient [-]
$R$  Universal gas constant [J/mol·K]
$St$  Strouhal number [-]
$T$  Absolute temperature [K]
\(t\) Duct wall thickness [m]

\(u'\) Time varying velocity perturbation [m/s]

\(u^+\) Normalized velocity [-]

\(U_0\) Mean flow velocity magnitude [m/s]

\(u_0\) Mean axial velocity [m/s]

\(u_r\) Friction velocity [m/s]

\(v_0\) Mean transverse velocity [m/s]

\(x\) Axial coordinate [m]

\(y\) Vertical coordinate [m]

\(y^+\) Wall unit [-]
Chapter 1

Introduction

Sound propagation in infinite ducts is a classical problem in acoustics which has been solved mathematically as an eigenvalue problem [2]. The main objective of this thesis is to develop an appropriate simulation method as an alternative method to investigate the scattering of sound at a duct termination. The range of validity of the method is studied by comparing the results to measurements and theory. Rämmal [1] has performed the experiment for plane acoustic waves and investigation of sound reflection of an open end of a circular duct emitting hot gas with low flow speeds. Rämmal has performed other experiments considering the effect of flow and flow velocity on sound reflection for heated subsonic air flow in duct termination up to $180^\circ C$ and Mach number $M < 0.31$ [1]. The experimental results obtained by using two microphone technique and results show that the magnitude of the reflection coefficient is extremely depended on jet flow Mach number and at high flow velocities when the temperature decreases, the reflection coefficient decreases. The paper states that at high temperature, more acoustic energy is transmitted to the surrounding air out of duct. The classical theoretical works by Lewine & Schwinger [4] considered sound reflection with no flow for an open unflanged pipe termination by applying the Wiener-Hopf technique. An important and major theoretical work on the sound radiation from an open ended straight pipe with flow has been done by Rienstra [5,6]. Munt [7,8] derived a theoretical model for determining the reflection coefficient for the transmission waves through the open end pipe in the presence of subsonic jet flow. In this theory, an unstable cylindrical vortex layer attached to the edge of the pipe which differences in mean subsonic flow, density and temperature are included. It is shown that flow influences the noise levels and it is increased upstream while the instability wave becomes weak at downstream [7,8]. The results show that plane wave reflection coefficients exceed
unity at low frequencies in accordance with experimental observations. By implementing Wiener-Hopf approach, an analytical solution was derived by considering vortex shedding in the problem of acoustic radiation from a semi infinite annular duct in uniform subsonic mean flow [6]. Rienstra used the method so-called complex contour deformation which is very convenient at high frequencies. It has done numerically solution by considering acoustic power loss corresponding to the vortex shedding and at frequencies close to the cut off frequency as a function of mode number of incident wave, Mach number, and hub radius. It shows for modes expect plane waves (at cut on frequency) the ratio of radiated and transmitted power is finite value however for plane waves, it is equal to zero and the acoustic power loss will be increased if Mach number or hub radius increased and frequency decreases.

The interesting part is to discuss that near the vortex sheet, the radiation condition and separating incoming and out coming waves at infinity can not be defined by energy flux vector. Acoustic transmission and reflection for for plane waves in hot region and in flow direction and transmitted into cold region which was separated by shear discontinuity was studied by Candel [9]. In this work he showed that temperature discontinuities was the main cause on the direction of propagation of transmitted waves and the results are strongly different from the uniform temperature case. Miles [10] and Ribner [11] discussed the reflection of the plane waves at an interface in relative motion, they also considered the possibility of the reflection coefficient for sufficiently high supersonic speeds. They studied on occurrence resonance at specific incident angles and the possibility of amplification and instability of interface.

Rämmal [12], applied FEM simulations for a three dimensional model to determine the acoustical passive properties of an open duct termination with a hot jet and compared the results with experimental observations. It was represented at low Mach numbers and Helmholtz numbers and the results almost agreed with the Munt’s theory.

The aim of this work, is to simulate the experimental data of [1]. Commercial software, Fluent 12.1 is used for flow field in two dimensions space, and the frequency domain and linearized Navier-Stokes equations are implemented in Comsol 3.5a for the acoustic field. The method is used to investigate the sound in the termination of ducts with incompressible isothermal flow. The comparison of results with experimental data and Munt theory, has been studied. Also the effect of the flow on result has been shown. At the end the parameters that can effect the results has been investigated.
Chapter 2

Flow simulation

2.1 Problem definition

The work presented in this thesis report emphasizes on the simulation of the plane waves for different discrete frequencies which are below the cut on frequency and the results has been compared with the experimental results by Ränmal [1]. For obtaining this, first the flow part has been solved numerically with the commercial software Fluent 12.1 RANS model with a second order accurate scheme was used in this project. The geometry is shown in Fig. 2.1 and parameters for flow and acoustic fields are defined as in Table 2.1.

![Geometry in two dimensional case study.](image)

A schematic overview of the simulation domain is shown in Fig.2.1. In order to efficiently evaluate the proposed problem and compare the results with experiments in three dimensions, we need to define an area aspect ratio $\eta$. This ratio needs to be the same for our 2D simulation as for 3D in experiment. This defines the area aspect ratio as the ratio between the area of outer to the area of the inner in 3D.
\[ \eta = \frac{\pi D^2/4}{\pi d^2/4} \]  

(2.1)

where \( D \) and \( d \) are the inner cylindrical diameter of the duct and outer cylindrical diameter of the duct respectively. The values for them can be found in Tab. 2.1. We need to determine the height of the inner and outer of the duct in 2D by keeping the \( \eta \), without any changes and just by looking at Fig. 2.1 we can write

\[ h + 2t = H \]  

(2.2)

\[ \eta = \frac{h}{H} \]  

(2.3)

then by inserting the thickness of the cylindrical duct \( t \) and \( \eta \) in Eq. 2.1, \( H \) can be obtained and then by applying it in Eq. 2.3, the inner diameter of the duct in two dimensional space will be obtained.

The outer domain of the duct is constrained by \( B \) and \( C \) which the value for them is shown in Tab. 2.1 below. This two parameters should not be too much low because of creation of reversed flow in flow simulations and also, due to the cost of simulations they should not be very far from the duct termination.

<table>
<thead>
<tr>
<th>Description</th>
<th>Symbol</th>
<th>value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Duct length</td>
<td>( L )</td>
<td>2.2</td>
<td>m</td>
</tr>
<tr>
<td>Outer cylindrical diameter</td>
<td>( D )</td>
<td>0.045</td>
<td>m</td>
</tr>
<tr>
<td>Inner cylindrical diameter</td>
<td>( d )</td>
<td>0.041</td>
<td>m</td>
</tr>
<tr>
<td>Thickness</td>
<td>( t )</td>
<td>0.002</td>
<td>m</td>
</tr>
<tr>
<td>Area aspect ratio</td>
<td>( \eta )</td>
<td>0.8301</td>
<td>[( \cdot )]</td>
</tr>
<tr>
<td>Outer 2D diameter</td>
<td>( H )</td>
<td>0.0235</td>
<td>m</td>
</tr>
<tr>
<td>Inner 2D diameter</td>
<td>( h )</td>
<td>0.0195</td>
<td>m</td>
</tr>
<tr>
<td>Mach number</td>
<td>( M )</td>
<td>0.0073</td>
<td>[( \cdot )]</td>
</tr>
<tr>
<td>Length of the outer domain</td>
<td>( C )</td>
<td>5</td>
<td>m</td>
</tr>
<tr>
<td>Height of the outer domain</td>
<td>( B )</td>
<td>0.3</td>
<td>m</td>
</tr>
<tr>
<td>Mean inlet velocity</td>
<td>( U_0 )</td>
<td>2.5</td>
<td>[m/s]</td>
</tr>
<tr>
<td>Reynolds number</td>
<td>( Re )</td>
<td>3343</td>
<td>[( \cdot )]</td>
</tr>
<tr>
<td>Frequencies</td>
<td>( f )</td>
<td>250 to 1250</td>
<td>[Hz]</td>
</tr>
</tbody>
</table>

Table 2.1: Geometry, flow and acoustic parameters according to Rämmal’s experimental data [1].
2.2 Flow equations

The fully Navier-Stockes equations for compressible and unsteady flow is considered to describe the flow field and also sound generation and propagation. Navier Stokes equations consist of the mass conservation equation, momentum equation and equation for conservation of energy. It should be mentioned that the flow separation and vorticity near the edge has a considerable effect on the results that is non-legible.

The main starting equation is the Navier-Stokes equations and it can be written as below:

mass conservation:

\[
\rho D\frac{\rho}{Dt} + \rho \frac{\partial u_k}{\partial x_k} = 0.
\] (2.4)

Momentum:

\[
\rho \frac{Du_i}{Dt} = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} + \rho F_i,
\] (2.5)

Energy:

\[
\rho \frac{De}{Dt} = -p \frac{\partial u_k}{\partial x_k} + \phi + \frac{\partial}{\partial x_k} \left( \kappa \frac{\partial T}{\partial x_k} \right)
\] (2.6)

with

\[
\phi = \tau_{ij} \frac{\partial u_i}{\partial x_j},
\] (2.7)

\[
\tau_{ij} = \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \frac{\partial u_k}{\partial x_k} \delta_{ij} \right)
\] (2.8)

Pressure obeys the ideal gas rule as \( p = \rho RT \) and \( e = e(p, T) \), where \( \rho \) is the density, \( p \) is the pressure, \( R \) is the universal gas constant and \( T \) is the absolute temperature. For two dimensional geometry, \( u_i \) is the velocity component in the \( i \)th-direction and \( u_j \) is the velocity component in the \( j \)th-direction where \( i = 1 \) is the \( x \) direction and \( i = 2 \) is the \( y \) direction. \( \frac{D}{Dt} \) is the convective derivative, \( \tau_{ij} \) is the viscous stress tensor, \( F_i \) is a volume force in the \( i \)th direction, \( e \) is internal energy, \( \kappa \) is the thermal conductivity, \( \phi \) is the dissipation function, \( \mu \) is the dynamic viscosity, and \( \delta_{ij} \) is the Kronecker delta function.
2.3 Turbulent

An important non-dimensional parameter that characterize the flow field to laminar transient and turbulent is Reynolds number. For high Reynolds number (generally greater than 3000) the flow field is turbulent and becomes unstable and chaotic. This means the convection is much stronger than dissipation.

\[ Re = \frac{UD}{\nu} \]  

(2.9)

where \( U \) is the mean velocity and \( D \) is characteristic length, here the inlet diameter of the duct and \( \nu \) is the kinematic viscosity which can be described by density and dynamic viscosity as below

\[ \nu = \frac{\mu}{\rho} \]  

(2.10)

Turbulent flow is inherently time dependent and chaotic. It is more convenience to look at the average of the flow which usually is ensemble average. for this purpose ensemble average is defined as below

\[ \bar{u}_i = \lim_{N \to \infty} \frac{1}{N} \sum_{n=1}^{N} u_i^{(n)} \]  

(2.11)

which each member of \( u_i(n) \) is regarded to independent realization of flow [13].

According to Tab. 2.1, \( Re \) in this project, the flow has been considered as turbulent flow.

2.3.1 Modeling of turbulence - Reynolds averaged Navier-Stokes

This method was presented by Reynolds in 1895, is based on the decomposition of flow variables into mean and fluctuating parts follow by time or ensemble averaging. For cases where the density is not constant, it is advisable to apply the density weighted or Favre decomposition to the velocity components. Otherwise, the averaged governing equations would become considerably more complicated due to additional correlations involving density fluctuations.
By applying the decomposition on the Navier-Stokes equations and taking
the average, it leads to the same equations for the mean variables with the
exception of two additional terms which needs to be modeled. The tensor
of viscous stresses is extended by one term-the Reynolds stress tensor. It
represents the transport of mean momentum due to turbulent fluctuations.
Furthermore the diffusive heat flux $k \nabla T$ in the energy equation is enhanced
by the turbulent heat flux vector$^2$. So the solution of RANS need to model
Reynolds stress term and turbulent heat flux vector and it cause to under-
stand the details of mean flow in turbulent flow. In this work a RANS method
with $k - \epsilon$ model has been applied.

The $k - \epsilon$ model is the most widely employed two-equation eddy viscosity
model. It is based on the solution of equations for the turbulent kinetic
energy and the turbulent dissipation rate. In this work, to get more accurate
simulation, realizable $k - \epsilon$ model has been used, this model contains a new
transport equation for the turbulent dissipation rate and the coefficient is
expressed as a function of mean flow and turbulence properties, however in
the standard model it is considered as constants. This allows the model to
satisfy certain mathematical constraints on the normal stresses consistent
with the physics of turbulence (realizability). Because of the existence of
wall boundary layer flows, modifications should be required which leads to
the use wall functions in the simulation.

2.3.1.1 Near wall region and Wall Functions

Existence of no-slip conditions for walls affect the velocity profile which causes
to define generally four zones in the inner region due to the wall bounded
flow. These zones are divided as viscous sub layers, buffer layers, logarithmic
layer and wake region. The mean velocity in near wall regions depend on $\nu$, $y$
and $u_\tau$ which are kinematic viscosity, the distance from wall and friction
velocity respectively. Then it is useful to normalize these variable as

\[ y^+ = \frac{yu_\tau}{\nu} \]  \hspace{1cm} (2.12)

\[ u^+ = \frac{u}{u_\tau} \]  \hspace{1cm} (2.13)

\[ u_\tau = \sqrt{\frac{\tau_w}{\rho}} \]  \hspace{1cm} (2.14)

here $\tau_w$ is the shear stress at the wall.
As it was mentioned above, the first region is the viscous region where $u^+$ increases linearly by increasing the $y^+$ up to 5. After that, the buffer layer has the biggest portion in turbulent production and this is usually located about $5 \leq y^+ \leq 40$. In the logarithmic layer, the Reynolds stress has the main role.

The low Reynold’s number models require very fine grids at walls. The standard condition is that the first node should be located at the distance $y^+ \leq 1$ from the wall. But here the distance between first node and the wall has to be bridged to the so called wall functions. The turbulence equations are not solved at the wall itself and at the first layer of the nodes.

The application of wall functions (provided the grid is not too coarse) has been utilized to reach reasonably accurate results for attached boundary layers.
Chapter 3

Simulation of the incompressible flow

3.1 Boundary conditions

In order to solve incompressible Navier-stokes, the commercial software Fluent 12.1 has been applied to solve the problem for pressure based, two dimensional and steady state flow.

For modeling in this case study, a RANS method with \( k-\varepsilon \) model and second order for momentum equations and first order for turbulent kinetic equations are applied. Also in order to solve the problem accurately additional data is needed about the flow variables at the boundaries.

In the present work, velocity inlet, pressure outlet, axis, wall (no-slip) have been implemented as boundary conditions in order to determine the flow field for incompressible flow by solving the RANS equations. The details can be found as Tab. 3.1 and Fig. 3.1:
The turbulent intensity ($I$), and turbulent length scale ($l$), are two characteristic of $k − \epsilon$ model. Turbulent intensity is defined as the ratio of the root-mean-square of the velocity fluctuations, $u'$, to the mean flow velocity $U_0$ and it is chosen to be 10%.

The turbulent length scale is the size of the large eddies that contain the energy in turbulent flows and for fully developed flow in ducts, $l$ is limited by the size of the duct, and this is clear because the size of the turbulent eddies can not be larger than the duct diameter, then the approximate relation between $l$ and duct diameter can be written as
where $D$ is the two dimensional diameter of the duct in this case study which is 19.54 mm, then the turbulent length scale will be 1.4 mm.

In order to set the initial conditions, the pressure at the outflow is defined as atmospheric pressure and the operating condition is the atmospheric pressure at the left edge of air domain.

Also for simplifying the simulation and saving the time, only half of the geometry due to symmetry has been simulated.

After defining the model and boundary conditions, creating a reasonable mesh is the important part that has been explained with details in Sec. 3.2

\[ l = 0.07D \]  

(3.1)

3.2 Mesh

For solving the flow field, 324498 quadrilateral cells has been used. As can be seen from Fig. 3.2, refinements are done close to the walls and near the termination of the duct which help to have more accurate results specially at the termination where flow vorticity exists. As mentioned in Sec. 2.3.1.1, the cell size should be chosen in a way that $y^+$ values be in the order of one.

![Figure 3.2: Mesh structure for simulation of flow field.](image)

It should be noted that choosing the accurate mesh and refinement has important effect on convergence and results, however boundary conditions are
very critical in simulations. The mesh was refined in several steps, until the $y^+$ values were fulfilled.

![Figure 3.3: $y^+$ values at the duct walls.](image)

### 3.2.1 Determination of duct length and outer domain

First, we tried different geometries with same boundary conditions in order to find the optimum length for parameter $C$ that has been shown in Fig. 2.1, without any reversed flow at outflow. This means, we should consider the outer domain far enough form the duct termination however the time of calculation would be increased. Also this has been done for parameter $A$. The optimum values can be found in Tab. 2.1.

The other characteristic of flow in duct, due to the profile of velocity is the distribution of velocity across the the cross section. So we increase the length of duct to have the fully developed flow in duct. Therefore, the distribution of flow becomes independent of position along $x$ axis and is the same near the termination. It means at a sufficient distance from the entrance of the duct, a boundary layer region will be formed at the wall in the inlet expands to the center and effectively fills the entire cross section of the duct. Consequently, when we have the fully developed internal flow, the shear stress at the wall will be independent of the distance measure along the $x$ axis and the pressure will be uniform and the turbulent quantities do not vary over any cross section and also the pressure gradient $\frac{dp}{dx}$ will be constant. Then in this case study, we defined the length of the duct 2.2 m while in experiment is 1.2 m. Another
consideration in choosing the duct length is the acoustic wave lengths such as longest wave length or lowest frequency has to be considered.

3.3 Discussion

The Reynolds Averaged Navier-Stokes equations has been solved after sketching the appropriate mesh and implementing the boundary conditions. The flow profile of a fully developed incompressible flow has been created with the realizable $k - \epsilon$ model.

The Reynolds number in this work as it mentioned before was 3343 and Mach number 0.0073, while the room temperature was 300 K, So Fig. 3.4, shows the fully developed flow profile which can be observed from $x = -1.7$ m to $x = 0.1$ m which is near the duct termination. Also that near the wall the velocity is almost zero which is due to the no-slip condition.
Figure 3.4: Velocity profile $v_x$ in and out of the duct.

The contour plot for axial Mach number can be found in Fig. 3.5 and Fig. 3.6, the generation of jet field is easily identified at the termination of the duct. It shows that near the edges, there are some vorticities in turbulent flow that can affect on acoustic field and will be discuss in further sections.
Fig. 3.5: Velocity vector $v_x$ near the duct termination edges.

Fig. 3.6 shows the mean flow component by solving the RANS equations. The region in this figure is $-0.5m \leq x \leq 1m$. Also because of the symmetry, we can plot everything for half of the geometry in order to save the cost of simulation. The most considerable velocity component is axial velocity while the transverse velocity is very low and more and less constant.
Figure 3.6: The mean flow components: axial velocity $u_0$, transverse velocity $v_0$, and mean pressure $p_0$. 
Chapter 4

Acoustics

Acoustic waves can be described as small pressure oscillations in a compressible fluid and acoustic energy is propagated in the medium as waves[12]. In this work, after obtaining the flow field and mean values for velocity and pressure, the acoustic simulations are performed. All the figures and plots are performed for half of the geometry due to the symmetry of the case study.

4.1 Acoustic wave motion and plane waves

We now come to study wave motion in air which is of the high attention in the science of acoustics. For sound waves, the molecules of air move in the direction of the wave propagation, however there is alternate compression and rarefactions. One of the most important and special case in acoustics is plane wave propagation where wave crests are in planes that are perpendicular to the direction of propagation.

4.1.1 Non-dimensional numbers

The typical frequency scaling when considering acoustics with the flow is the dimensionless so-called Strouhal number defined as

\[
St = \frac{\omega L}{U}
\]  

(4.1)
where $\omega$ is the angular frequency, $U$ is the flow velocity and $L$ is the characteristic length. A small Strouhal number indicates that the flow can be assumed as quasi-stationary. In this work, characteristic length for duct can be define as inner 2D duct diameter which is written as ($h$).

Another non-dimensional number that has been used in this project is Mach number and it is followed as

$$M = \frac{U}{c_0}$$  \hspace{1cm} (4.2)

while $c_0$ is the speed of sound.

In aeroacoustics, non-dimensional numbers which can characterize a problem, are dependent to each other and can be defined as below

$$He = St \cdot M$$  \hspace{1cm} (4.3)

where $He$ is the Helmholtz number.

If the Helmholtz number is much smaller than unity then the source region is defined as acoustically compact compare to the wave length.

In this work, in order to compare the simulations and experiment results with each other, Helmholtz number is used.

### 4.2 Frequency scaling

In a 2D geometry, acoustical properties occur at different frequencies compare in 3D cylindrical pipe. By introducing the so-called normalized Helmholtz number $He^*$, it is possible to compare acoustic propagation in 2D and 3D with each other. The normalized Helmholtz number is defined as the Helmholtz number divided by the Helmholtz number of the cut-on frequency of the first higher order mode wave which is propagating in duct

$$He^* = \frac{He}{He_{cut-on}}$$  \hspace{1cm} (4.4)

Then this $He^*$ should be equal for the 2D and 3D case. After that it is possible to compare the results of the 3D experiment to simulations which has been carried out in 2D. The normalized Helmholtz number for the cylindrical case can be defined as below
\[ He^{*}_{cyl} = \frac{He_{cyl}}{\kappa_0} \quad (4.5) \]

where \( He_{cyl} \) is the Helmholtz number in a 3D cylindrical pipe.

\[ He_{cyl} = k_{cyl}d/2 \quad (4.6) \]

where \( d \) is the inner diameter of the cylindrical pipe has been used in experiment and \( k_{cyl} \) is the wave number that can be stated as \( k_{cyl} = \frac{\omega_{cyl}}{c_0} \) and \( \omega_{cyl} = 2\pi f_{cyl} \), then the Eq. (4.5) becomes as

\[ He^{*}_{cyl} = \frac{d\pi f_{cyl}}{c_0\kappa_0} \quad (4.7) \]

and also the same method can be applied for 2D as

\[ He^{*}_{2D} = \frac{He_{2D}}{\pi} \quad (4.8) \]

where \( He_{2D} \) is Helmholtz number in 2D duct.

\[ He_{2D} = k_{2D}h \quad (4.9) \]

\( h \) is the inner diameter for 2D duct and \( k_{2D} \) is the wave number in 2D as \( k_{2D} = \frac{\omega_{2D}}{c_0} \) and \( \omega_{2D} = 2\pi f_{2D} \) so by inserting these in Eq.(4.7), we have

\[ He^{*}_{2d} = \frac{f_{2D}h}{c_0} \quad (4.10) \]

as it was expressed before, the \( He^{*}_{cyl} \) should be equal to \( He^{*}_{2D} \) therefore by considering the Eqs. (4.10) and (4.7) the frequency in 2D can be represented as

\[ f_{2D} = \frac{\pi d}{h\kappa_0} f_{cyl} \quad (4.11) \]

where \( \kappa_0 \approx 3.832 \) is the cut on Helmholtz number for a cylindrical duct, while the cut on Helmholtz number for 2D duct is \( \pi \) [14]. Also, the cut-on frequency for mode \( n \) can be obtained as [14]

\[ f_n^c = \frac{c_0 k_{\perp,n}}{2\pi} \sqrt{1 - M^2} \quad (4.12) \]
This argument indicates that if the frequency is lower than the cut on frequency for mode $n$, then the $n$:th mode dies out exponentially in the direction of the propagation and transmit no acoustic power. This happens in acoustic near fields. If the frequency is higher than the cut-on frequency, higher acoustical modes will propagate. In this project all the frequencies are less than the cut on frequency for the first higher order mode, thus only plane waves propagates.

The frequency range which has been applied for measurement is $0.1 \leq He \leq 0.8$ and it is due to the two microphone technique that has restricted the frequency range in [1] therefore the $0.026 \leq He^* \leq 0.21$ is applied for this project.

The simplest way to to calculate the reflection for sudden changes into the cross section is using the plane wave decomposition. Fig. 4.1 shows the plane wave decomposition in the duct and whole of the case.

$$\hat{p}_{\text{duct}}(x, \omega) = (\hat{p}_+ e^{-ik_+ x} + \hat{p}_- e^{ik_- x}) e^{i\omega t}$$

$$\hat{u}_{\text{duct}}(x, \omega) = \frac{1}{\rho_0 c_0} (\hat{p}_+ e^{-ik_+ x} + \hat{p}_- e^{ik_- x}) e^{i\omega t}$$

where $\hat{p}$ and $\hat{u}$ is the acoustic pressure and acoustic velocity in frequency domain, respectively. $\hat{p}_+$ is the acoustic pressure for wave propagating to the downstream and $\hat{p}_-$ is the acoustic pressure for wave propagating to the upstream. Also $k_+$ is the wave number propagating in positive $x$ direction along the duct and $k_-$ is in the opposite direction and this is defined as

$$k_\pm = \frac{k_0}{1 \pm M}$$
where \( k_0 = \frac{\omega}{c_0} \) and \( M = \frac{U}{c_0} \), and \( U \) is the mean velocity of flow.

The phase speed for plane wave also can be represented as below

\[
c_{ph} = \frac{\omega}{k} = c_0(\pm 1 + M \cos \theta)
\]  

(4.16)

where \( \theta \) is the angel between mean velocity vector and normal vector to plane wave propagation [14].

Then by applying a plane wave decomposition we can write the propagating components as [3]:

\[
\hat{\rho} = \hat{\rho}_+ + \hat{\rho}_-
\]

(4.17)

the relation between the acoustic density and acoustic velocity is considered as following:

\[
\hat{\rho} = \pm \frac{\rho_0}{c_0} \hat{u}
\]

(4.18)

so the characteristic impedance is not change by adding flow from the no flow case.

Therefore the density for incoming and out-coming wave can be written such as:

\[
\hat{\rho}_+(x) = \frac{1}{2}(\hat{\rho}_{mean} + \frac{\rho_0}{c_0} \hat{u}_{mean})
\]

(4.19)

\[
\hat{\rho}_-(x) = \frac{1}{2}(\hat{\rho}_{mean} - \frac{\rho_0}{c_0} \hat{u}_{mean})
\]

(4.20)

\( \hat{\rho}_{mean} \) and \( \hat{u}_{mean} \) is the averaging of acoustic density and velocity density over the duct cross section such as :

\[
\hat{\rho}_{mean} = \frac{1}{H} \int_0^H \hat{\rho}(x, y) dy
\]

(4.21)

\[
\hat{u}_{mean} = \frac{1}{H} \int_0^H \hat{u}(x, y) dy
\]

(4.22)
So the reflection coefficient is defined as:

\[ R = \frac{\hat{\rho}_-}{\hat{\rho}_+} \]  

(4.23)

The reflection coefficient is a complex value which depends on the boundary impedance, medium and frequency.

### 4.3 Linearized Navier-Stokes equations

In order to simulate the acoustics for the problem with considering flow in Comsol, the linearized Navier-Stokes equations in frequency domain has been applied. It helps to reduce the time of simulations significantly.

The methodology has been described in details by Kierkegaard et al. [3]. He used the full compressible Navier-Stokes equation as it was clarified in Sec. 2.2 and using the decomposition that variables such as density, pressure and velocity can be decomposed into an a priori time independent mean flow and unknown time-dependent perturbations thus,

\[
\begin{align*}
\rho(x, t) &= \rho_0(x) + \rho'(x, t) \\
u(x, t) &= u_0(x) + u'(x, t) \\
v(x, t) &= v_0(x) + v'(x, t) \\
p(x, t) &= p_0(x) + p'(x, t)
\end{align*}
\]  

(4.24)

where \( \rho_0, u_0, v_0 \) and \( p_0 \) are the mean values and \( \rho', u', v' \) represent the time-varying perturbations [3]. Due to isentropic flow in this work, the relation between pressure and density is followed by

\[
\frac{\partial p'}{\partial x_i} = c_0^2 \frac{\partial \rho'}{\partial x_i}
\]  

(4.25)

where \( c_0 \) is the speed of sound and \( i = 1 \) is the \( x \)-direction and \( i = 2 \) is the \( y \)-direction. As was mentioned before, harmonic time dependence for the perturbation quantities has been assumed which means

\[
q' = \text{Re}(\bar{q}(x)e^{-i\omega t})
\]  

(4.26)
where \( \hat{q} \) is a complex value and \( \omega \) is the angular frequency. Then we insert Eqs. (4.24)-(4.26) into Navier-Stokes equation that has been described in Sec. 2.2, consequently linearized Navier-Stokes equations can be defined as [3].

\[
\hat{\rho} : (u_0 \ v_0) \nabla \hat{\rho} + \left( \frac{\partial u_0}{\partial x} + \frac{\partial v_0}{\partial y} - i\omega \right) \hat{\rho} = -\left( \frac{\partial \rho_0 \hat{u}}{\partial x} + \frac{\partial \rho_0 \hat{v}}{\partial y} \right), \tag{4.27}
\]

\[
\hat{u} : \nabla^T \left[- \left( \begin{array}{cc} \frac{4}{3} \mu & 0 \\ 0 & \mu \end{array} \right) \nabla \hat{u} \right] + \rho_0 (u_0 \ v_0) \nabla \hat{u} + \rho_0 \left( \frac{\partial u_0}{\partial x} - i\omega \right) \hat{u} = \rho_0 \hat{F}_x - (u_0 \frac{\partial u_0}{\partial x} + v_0 \frac{\partial u_0}{\partial y}) \hat{\rho} - c^2 \frac{\partial \hat{\rho}}{\partial x} + \frac{1}{3} \mu \frac{\partial^2 \hat{\rho}}{\partial x \partial y} - \rho_0 \frac{\partial u_0}{\partial y} \hat{\rho}, \tag{4.28}
\]

\[
\hat{v} : \nabla^T \left[- \left( \begin{array}{cc} \mu & 0 \\ 0 & \frac{4}{3} \mu \end{array} \right) \nabla \hat{v} \right] + \rho_0 (u_0 \ v_0) \nabla \hat{v} + \rho_0 \left( \frac{\partial v_0}{\partial y} - i\omega \right) \hat{v} = -c^2 \frac{\partial \hat{\rho}}{\partial y} - (u_0 \frac{\partial v_0}{\partial x} + v_0 \frac{\partial v_0}{\partial y}) \hat{\rho} - \rho_0 \frac{\partial v_0}{\partial x} \hat{\rho} + \frac{1}{3} \mu \frac{\partial^2 \hat{\rho}}{\partial x \partial y}, \tag{4.29}
\]

where \( \nabla = (\frac{\partial}{\partial x} \ \frac{\partial}{\partial y})^T \). Then all these Linearized Navier-Stokes equations has been implemented in Comsol in order to simulate and solve the acoustic part.

### 4.4 Acoustic boundary conditions

In this work, various acoustic boundaries has been defined for solving the problem in the duct. On the surface of the pipe, rigid wall and slip boundary condition in \( x \)-direction has been applied, however in \( y \)-direction for that wall, rigid wall with no slip boundary condition is assumed,

\[
\hat{v} = 0, \ n \cdot \nabla \hat{u} = 0 \text{ and } \hat{n} \cdot \nabla \hat{\rho} = 0.
\]

Where \( \hat{n} \) is a unit vector perpendicular to the walls. Similarly, at the vertical duct wall due to the thickness of the 2D duct, rigid wall with no slip boundary condition in \( x \) direction can be written as following used

\[
\hat{u} = 0, \ n \cdot \nabla \hat{v} = 0 \text{ and } \hat{n} \cdot \nabla \hat{\rho} = 0.
\]
This means that at rigid walls, the velocity in axial direction and the gradient of the pressure will be zero.

The benefit of using the slip boundary condition is that the acoustic boundary layers along the duct would not be solved along the duct walls. Consequently the computational cost will be decreased significantly.

Considering plane waves of a discrete frequency traveling along the duct, non-reflecting boundaries can be used at the in-and outflow boundaries, this can be expressed as

\[ \hat{n} \cdot \nabla \hat{\rho} = ik \hat{\rho}, \quad \hat{n} \cdot \nabla \hat{u} = ik \hat{u} \]

and \( \hat{n} = 0 \) except for upper domain of the out flow which \( \hat{n} \cdot \nabla \hat{v} = ik \hat{v} \).

Here, \( k \) is the convective wave number and is defined as in Eq. 4.15

So by looking at Eq.4.15 it can be concluded that the axial wave number for downstream, \( k_+ \), is lower than the axial wave number is upstream, \( k_- \). This mean that traveling waves in downstream are propagating faster that upstream.

For the symmetry line the boundary conditions of the pressure gradient and velocity gradient along \( x \) direction and velocity along \( y \) direction is zero.

### 4.4.1 Buffer zones

In order to have the non reflecting boundaries at inflow and outflow, we assumed buffer zone in that area, this means additional artificial viscosity is applied for this purpose. It is one way to sacrifice some part of the computational domain near the outlet where perturbation damped by almost high artificial viscosity before the wave reaches to the outlet. It will help to prevent the reflection effect at the in-outflow boundaries. Therefore, artificial viscosity is added to physical viscosity in the buffer zones Fig.4.2.

\[ \mu = \mu_{\text{physical}} + \mu_{\text{artificial}} \quad (4.30) \]

Accordingly it can be said artificial dynamic viscosity inside the computational domain is zero and inside the buffer zone it smoothly increased to a specific values which in this work, the maximum is 20 at corners. Fig. 4.2 shows the position of artificial viscosity in inside and outside the duct.
4.5 Model constants and variables

As an acoustic source, a time harmonic body force function \( \hat{F}(x) \) is applied over the upstream region in the domain. The body force function is defined as bell-shaped piece-wise cubic interpolation polynomial with maximum amplitude \( 10^2 \). The position of the body force is \(-2m \leq x \leq -1m\). Table 4.1 shows the constants and variables that has been set for simulation the acoustics by Comsol 3.5a.

Figure 4.2: An overview of the artificial viscosity inside and outside the duct
<table>
<thead>
<tr>
<th>Expression</th>
<th>Value</th>
<th>unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\mu$</td>
<td>$1.8 \cdot 10^{-5}$</td>
<td>[kg/m.s]</td>
</tr>
<tr>
<td>$H$</td>
<td>0.02354</td>
<td>[m]</td>
</tr>
<tr>
<td>$h$</td>
<td>0.01954</td>
<td>[m]</td>
</tr>
<tr>
<td>$D$</td>
<td>0.045</td>
<td>[m]</td>
</tr>
<tr>
<td>$\gamma$</td>
<td>1.4</td>
<td>[-]</td>
</tr>
<tr>
<td>$\kappa_0$</td>
<td>3.832</td>
<td>[-]</td>
</tr>
<tr>
<td>$\rho_0$</td>
<td>1.2</td>
<td>[kg/m$^3$]</td>
</tr>
<tr>
<td>$\eta$</td>
<td>0.8301</td>
<td>[-]</td>
</tr>
<tr>
<td>$f_{cyl}$</td>
<td>250-1650</td>
<td>[Hz]</td>
</tr>
<tr>
<td>$f_{2D}$</td>
<td>$\frac{\pi d}{2m_{ref}} f_{cyl}$</td>
<td>[Hz]</td>
</tr>
<tr>
<td>$\omega$</td>
<td>$2\pi f$</td>
<td>[rad/s]</td>
</tr>
<tr>
<td>$h/2_{up}$</td>
<td>0.00977</td>
<td>[m]</td>
</tr>
<tr>
<td>$H/2_{down}$</td>
<td>0.0118</td>
<td>[m]</td>
</tr>
</tbody>
</table>

Table 4.1: Model constants and variables

4.5.1 Discussion

After calculating the profiles for fully developed flow and obtaining the mean values in Chap.1, the linearized Naiver-Stokes equations (4.29) were applied in Comsol 3.5a, solved with the direct UMFPACK direct solver, the mesh contains 6953 triangular elements of cubic Lagrangian interpolation polynomials. Consequently the problem is solved for 99183 degrees of freedom. The mesh is constructed such that near the termination of duct and edges were much finer where small scale vorticities was appeared. Then according to the normalized Helmholtz number, the problem has been solved for different frequencies and reflection coefficients according to the Eq. 4.23 has been obtained.

Fig. 4.3, shows the reflection coefficient regarding to wave decomposition for a normalized Helmholtz number, $He^*=0.15$, with $f_{cyl}=1550$ Hz in the experiments for the cylindrical pipe. The other frequencies can be found in the Appendix. The termination of the duct is located at $x=0$. 

32
Figure 4.3: Plane wave decomposition of simulated data at $f_{cyl}=1150$ Hz, $U_{mean}=2.5$ m/s. Absolute value of total density $|\hat{\rho}|$: solid line. Magnitude of the density of the downstream propagating wave $|\hat{\rho}^+|$: dash line. Magnitude of the density of the upstream propagating wave $|\hat{\rho}^-|$: dash dotted line.

This mesh is also applied for the case with no mean flow i.e., the simulation with 9653 triangular elements of cubic Lagrangian interpolation polynomials has been implemented for the case with no flow, and the results for reflection coefficient at $x=-1$ is compared with experiment, Munt theory and simulation result for the case with mean flow. All these information can be seen in Fig. 4.4, for all the lines. Effects of the flow ($U_{mean}=2.5$m/s) has been considered except for the dash dotted line with red cross, in that case the result of simulations neglects the effect of the mean flow. As can be figured out from the Fig. 4.4, the reftection coefficient is more or less the same when we have flow in duct or not and that is because of the low Mach number. However if the Mach number or the mean velocity increases, this difference will be increased noticeably.
For investigation of the results, according to Fig. 4.4, two statistical quantities such as root mean square (RMS), and standard deviation have been calculated for the simulations data and measurements data.

**Root mean square**: The RMS value for a set of values is defined as the square root of average of the squares of the original values. It is described for $n$ values as below

$$x_{rms} = \sqrt{\frac{x_1^2 + x_2^2 + \ldots + x_n^2}{n}} \quad (4.31)$$

According to Eq. 4.31, RMS value of the simulations data is 0.78 and for the measurements data is 0.81, so the difference of simulation data with considering the convection and flow is about 0.032 or 3%.
**Standard deviation**: Another statistical quantity that has been calculated for the results is the standard deviation. Standard deviation shows how much variation or dispersion exist from the average. It is defined as

\[
\sigma = \sqrt{\frac{1}{N} \left( x_i - \bar{x} \right)^2}
\]  

(4.32)

where N is the number of data. We can use Eq. 4.32 for every normalized Helmholtz number, the result can be observed in Fig. 4.5, it seems around \( \text{He}^* = 0.12 \), the standard deviation reaches the maximum value where the difference between the simulations and measurement is 0.004.

![Graph showing standard deviation for every normalized Helmholtz number](image)

Figure 4.5: Standard deviation for every normalized Helmholtz number between the experimental values and simulation

### 4.5.1.1 Effect of refining the mesh

The grid for the duct has been refined specially near the termination. Tab. 4.2, shows the effect on the case which the cylindrical frequency is 650Hz and \( U_{\text{mean}} = 2.5 \text{m/s} \). \( R \) is the reflection coefficient.
Table 4.2: Comparing the results by refining the mesh at 650Hz, \( U_0 = 2.5 \text{ m/s} \) and \( T = 21^\circ C \)

For instance, Fig. 4.6, represents two different meshes which have the fine mesh close to the duct termination.

Figure 4.6: Different meshes. a is for 3612 elements and b is for 14448 elements.
Table 4.2 shows that there is 0.04% improvement with increasing the number of elements from 3612 to 7853. So refining the whole geometry once is not enough. Another interesting result is that, the reflection coefficient when refining the whole case twice is the same while refining just near the termination however it will reduce the solution time significantly. It can be added 7853 elements are fair enough to get the results with the same accuracy because if near the termination fine more such as 15573 is implemented, it just increases the time and D.O.F while the reflection coefficient is the same as 7853 elements or even a little bit decreasing without changing the results like 6953 elements that has 99183 D.O.F

4.5.1.2 Investigation of buffer zone area effect on the reflection coefficient

In this part different buffer zones has been applied to see the effect of the upper buffer zone on the reflection coefficient. Figures 4.7-4.9 show the different cases that have been studied.

In case 1, the height of the outer domain is increased thus it causing the buffer zone to increase, the reflections are damped by artificial viscosity of buffer i.e., the height of it is 0.4118-0.2=0.2118 m, (the initial setting was 0.1118m). The reflection coefficient for this case for $f = 650$ Hz is 0.9214.

Figure 4.7: Distribution of artificial viscosity for case 1.
In case 2, the geometry is the same but the area is slightly increased i.e. the height is 0.4118 - 0.15 = 0.2688 m. The reflection coefficient becomes 0.9215, therefore compared to case 1, it is almost the same.

Fig. 4.8, shows the distribution for artificial viscosity for case 2.

![Figure 4.8: Distribution of artificial viscosity for case 2.](image)

In case 3, the buffer zone is heightened up more thus the outer physical domain is increased. The buffer zone is 0.6118 - 0.5 = 0.118, yielding a reflection coefficient of 0.9221 at 650 Hz.
In case 4, the height of the artificial viscosity is $0.6118 - 0.2 = 0.4118$, and reflection coefficient is 0.9225. Fig. 4.10 shows the distribution of artificial viscosity for the case 4.
4.6 Summary

The simulation of acoustic wave propagation in a duct with mean flow effects has been studied. The acoustic wave propagation is carried out with a frequency domain linearized Navier-Stokes methodology which has been described by details in [3]. The mean flow is obtained by solving the fully turbulent developed flow in the duct and using RANS. As a test case, the acoustic response for a duct termination without mean flow is simulated. After that the effect of flow has been considered, however the magnitude of the mean velocity was high enough to significantly affect the acoustic scattering. The results are compared to the Munt theory and measurements with more and less good agreement. The comparison has done for normalized Helmholtz number that makes it possible to compare calculations for rectangular two dimensional geometry with cylindrical duct.
Finally, the effect of Buffer zones and meshes have been discussed. We can conclude that this simulation is a good start to evaluate the effect of velocity and flow scattering. This means if the flow mean velocity increases, the more vorticities near the duct termination appears and has more effect in coupling of fluid and acoustics. These vortex shedding can cause a an important phenomena such as whistling.
Bibliography


Appendix

Acoustic simulation

The figures below show the magnitude of the density perturbations after applying a plane wave decomposition of the simulated data at different frequencies. The range for normalized Helmholtz number is $0.0245 \leq He^* \leq 0.161$. For these normalized Helmholtz numbers the $f_{2D}$ are 215 Hz to 1418 Hz while frequencies in the experiment for the cylindrical pipe were from 250 Hz to 1650 Hz.

Figure 4.11: Plane wave decomposition of simulated data at $f_{cyl}=250$ Hz, $U_{mean}=2.5$ m/s. Absolute value of total density $|\hat{\rho}|$: solid line. Magnitude of the density of the downstream propagating wave $|\hat{\rho}^+|$: dash line. Magnitude of the density of the upstream propagating wave $|\hat{\rho}^-|$: dash dotted line.
Figure 4.12: Plane wave decomposition of simulated data at $f_{cyl}=350$ Hz, $U_{mean}=2.5$ m/s. Absolute value of total density $|\hat{\rho}|$: solid line. Magnitude of the density of the downstream propagating wave $|\hat{\rho}^+|$: dash line. Magnitude of the density of the upstream propagating wave $|\hat{\rho}^-|$: dash dotted line.
Figure 4.13: Plane wave decomposition of simulated data at $f_{cy}=450$ Hz, $U_{mean}=2.5$ m/s. Absolute value of total density $|\rho|$ : solid line. Magnitude of the density of the downstream propagating wave $|\rho^+|$ : dash line. Magnitude of the density of the upstream propagating wave $|\rho^-|$ : dash dotted line.
Figure 4.14: Plane wave decomposition of simulated data at $f_{cyl} = 550$ Hz, $U_{mean} = 2.5$ m/s. Absolute value of total density $|\hat{\rho}|$: solid line. Magnitude of the density of the downstream propagating wave $|\hat{\rho}^+|$: dash line. Magnitude of the density of the upstream propagating wave $|\hat{\rho}^-|$: dash dotted line.
Figure 4.15: Plane wave decomposition of simulated data at $f_{cyl}=650$ Hz, $U_{mean}=2.5$ m/s. Absolute value of total density $|\hat{\rho}|$: solid line. Magnitude of the density of the downstream propagating wave $|\hat{\rho}^+|$: dash line. Magnitude of the density of the upstream propagating wave $|\hat{\rho}^-|$: dash dotted line.
Figure 4.16: Plane wave decomposition of simulated data at $f_{cyl}=750$ Hz, $U_{mean}=2.5$ m/s. Absolute value of total density $|\hat{\rho}|$: solid line. Magnitude of the density of the downstream propagating wave $|\hat{\rho}^+|$: dash line. Magnitude of the density of the upstream propagating wave $|\hat{\rho}^-|$: dash dotted line.
Figure 4.17: Plane wave decomposition of simulated data at $f_{cycl}=850$ Hz, $U_{mean}=2.5$ m/s. Absolute value of total density $|\hat{\rho}|$: solid line. Magnitude of the density of the downstream propagating wave $|\hat{\rho}^+|$: dash line. Magnitude of the density of the upstream propagating wave $|\hat{\rho}^-|$: dash dotted line.
Figure 4.18: Plane wave decomposition of simulated data at $f_{cyl} = 950$ Hz, $U_{mean} = 2.5$ m/s. Absolute value of total density $|\hat{\rho}|$: solid line. Magnitude of the density of the downstream propagating wave $|\hat{\rho}^+|$: dash line. Magnitude of the density of the upstream propagating wave $|\hat{\rho}^-|$: dash dotted line.
Figure 4.19: Plane wave decomposition of simulated data at $f_{cyl}=1050$ Hz, $U_{mean}=2.5$ m/s. Absolute value of total density $|\hat{\rho}|$: solid line. Magnitude of the density of the downstream propagating wave $|\hat{\rho}^+|$: dash line. Magnitude of the density of the upstream propagating wave $|\hat{\rho}^-|$: dash dotted line.
Figure 4.20: Plane wave decomposition of simulated data at $f_{cyl}=1150$ Hz, $U_{mean}=2.5$ m/s. Absolute value of total density $|\hat{\rho}|$: solid line. Magnitude of the density of the downstream propagating wave $|\hat{\rho}^+|$: dash line. Magnitude of the density of the upstream propagating wave $|\hat{\rho}^-|$: dash dotted line.
Figure 4.21: Plane wave decomposition of simulated data at $f_{cyl}=1250$ Hz, $U_{mean}=2.5$ m/s. Absolute value of total density $|\hat{\rho}|$ : solid line. Magnitude of the density of the downstream propagating wave $|\hat{\rho}^+|$ : dash line. Magnitude of the density of the upstream propagating wave $|\hat{\rho}^-|$ : dash dotted line.
Figure 4.22: Plane wave decomposition of simulated data at $f_{cyl} = 1350$ Hz, $U_{mean} = 2.5$ m/s. Absolute value of total density $|\hat{\rho}|$: solid line. Magnitude of the density of the downstream propagating wave $|\hat{\rho}^+|$: dash line. Magnitude of the density of the upstream propagating wave $|\hat{\rho}^-|$: dash dotted line.
Figure 4.23: Plane wave decomposition of simulated data at $f_{cyl}=1450$ Hz, $U_{mean}=2.5$ m/s. Absolute value of total density $|\hat{\rho}|$: solid line. Magnitude of the density of the downstream propagating wave $|\hat{\rho}^+|$: dash line. Magnitude of the density of the upstream propagating wave $|\hat{\rho}^-|$: dash dotted line.
Figure 4.24: Plane wave decomposition of simulated data at $f_{cyl}$=1550 Hz, $U_{mean}$=2.5 m/s. Absolute value of total density $|\hat{\rho}|$: solid line. Magnitude of the density of the downstream propagating wave $|\hat{\rho}^+|$: dash line. Magnitude of the density of the upstream propagating wave $|\hat{\rho}^-|$: dash dotted line.