Towards sibilant /s/ modelling: preliminary computational results

X. Grandchamp\textsuperscript{a}, A. Van Hirtum\textsuperscript{a}, X. Pelorson\textsuperscript{a}, K. Nozaki\textsuperscript{b} and S. Shimojo\textsuperscript{b}

\textsuperscript{a}D\textdegree{}partement Parole \& Cognition, GIPSA-lab, 46, avenue Félix Viallet, 38031 Grenoble Cedex, France
\textsuperscript{b}Cyber Media Center, Osaka University Cybermedia, 5-1 Mihogaoka Ibaraki, 567-0047 Osaka, Japan

xavier.grandchamp@gipsa-lab.inpg.fr
The importance of turbulence in human speech sounds production is generally accepted in the case of fricatives. Nevertheless, this phenomenon is marginally taken into account in physical modelling of speech production. The current study considers steady flow at a given Reynolds number \( \text{Re} = 4000 \). Large Eddy Simulation has been used to simulate the flow evolution in case of two-dimensional geometries. The used geometrical and flow characteristics are severe simplifications of the human upper airways beyond the glottis. The impact of those typical geometries on the computed flow field is searched in order to detect the aerodynamical sources produced by turbulent flow, crucial in /s/ production. The presence of the obstacle in a uniform channel and the curved pipe induce reversed flow. Nevertheless, the turbulence intensity, in case of the obstacle configuration does not correspond with turbulent flow, whereas the curve tends to give higher turbulent velocities along the curvature.

1 Introduction

Human speech sounds can be classified into two distinctive categories depending on their production mode: voiced and unvoiced sounds. The production of voiced sound was deeply studied, and among of research is available in the literature. The airflow expelled by the lungs interacts with the vocal folds leading to an auto-oscillation of the vocal folds. The other mechanism is purely aerodynamical and corresponds to the production of unvoiced sound. Nevertheless, less attention was given to the mechanism of production of unvoiced sound. The complex geometry of the human upper airways, and the flow conditions yield the turbulent air flow. The importance of the turbulence in the production of unvoiced sound is noted [2]. Numerous experimental studies reveal the importance of the interaction between the flow and obstacle, in a.o [10, 12]. The mechanism of sound production and the importance of vorticity and the necessary flow-separation conditions in the production of fricatives are studied [7, 11]. The study of jet vorticity and its evolution for unvoiced sound is summarised in [6] as “the sound field was seen to be determined uniquely by the interaction of jet vorticity and the shape of the vocal tract”.

One of the main articulator concerning obstacle-flow interaction is the teeth. The impact of teeth on the production of sibilant /s/ is studied by [5, 9]. Large Eddy Simulation (LES) is applied in the mid-sagittal plane of a human oral cavity [9]. Nevertheless, the complexity of the geometry gave us a difficult understanding of the phenomena.

As a first step for a future application in speech production, the aim of this paper is to analyse with LES, the evolution of 2 dimensional laminar inflow in the case of schematic human upper airways. The influence of the vocal folds and false vocal folds are left untreated. The study is restricted on typical geometries present in the upper airways. A rectangular straight pipe, a rectangular straight pipe with an obstacle, and finally a rectangular curved pipe are considered. The three geometries are studied separately and their impact on the flow evolution is discussed.

The paper is organised as follows. Section 2 presents the numerical method and numerical domains. The results for the different geometries are discussed in section 3. The conclusion is formulated in section 4.

2 Computational Method

The airflow is simulated within the different geometries by using a commercial explicit finite element code “Front Flow Blue/FFB”, developed by Advance Soft Corporation [3]. In the following section the method is outlined [6]. Next, the computational domains are briefly presented.

2.1 Governing equations

This code solves spatially-filtered incompressible continuity equation and Navier-Stokes equations.

\[
\frac{\partial \bar{u}_i}{\partial t} + \frac{\partial}{\partial y_j} (\bar{u}_i \bar{u}_j + \tau_{ij}) = - \frac{1}{\rho_0} \frac{\partial p}{\partial y_j} + \frac{\partial}{\partial y_j} \left\{ \nu \left( \frac{\partial \bar{u}_i}{\partial y_j} + \frac{\partial \bar{u}_j}{\partial y_i} \right) \right\},
\]

\( \bar{u} \) represents the grid-scale static pressure, \( \nu \) the kinematic viscosity, \( u_i \) the grid-scale velocity component. The subgrid-scale (SGS) stress tensor is defined by:

\[
\tau_{ij} = \bar{u}_i \bar{u}_j - \bar{u}_i \bar{u}_j.
\]

The Smagorinsky closure is applied to the SGS stress \( \tau_{ij} \):

\[
\tau_{ij} = -\frac{1}{3} \delta_{ij} \delta_{kk} \nu_{SGS} \bar{S}_{ij},
\]

Where

\[
\nu_{SGS} = (C_s \Delta)^2 \left| \bar{S} \right| = \frac{\left( 2 \bar{S}_{ij} \bar{S}_{ij} \right)^{1/2}}{2} = \frac{1}{2} \left( \frac{\partial \bar{u}_i}{\partial y_j} + \frac{\partial \bar{u}_j}{\partial y_i} \right)
\]

\( C_s \) is the Smagorinsky coefficient and \( \Delta \) is the size of the grid filter. In our case, the dynamic Smagorinsky model with modification due to [8] is used. \( C_s \) is determined locally in time and space by

\[
C_s^2 = \frac{\langle M_{ij} \bar{y}_j \rangle}{2\Delta^2 \langle M_{ij} \bar{M}_{ij} \rangle},
\]
\[ M_y = \tilde{\beta}_y - \alpha^2 \tilde{\Delta} S_y, \quad \alpha \cong \frac{\tilde{\Delta}}{\Delta} = 2, \]  
\[ l_y = (\tilde{\gamma}_y - \tilde{\Delta} u_j) - \frac{1}{3} \delta_j (\tilde{\gamma}_{ik} - \tilde{\Delta} \tilde{u}_i \tilde{u}_k), \]  

Where \( \beta_y = \frac{1}{\Delta} \tilde{\Delta} S_y \) and \( \gamma_y = \tilde{\Delta} u_j \). The symbol \( \tilde{\Delta} \) represents a test-filtering operation, \( \langle \langle \cdot \rangle \rangle \) represents an averaging operation, and \( \tilde{\Delta} \) is the size of the test filtered width. A fully implicit Crank-Nicolson scheme (CN) and explicit time streamline upwind scheme were employed for integrating the momentum equation. The spatial discretization is based on the Finite Element Method (FEM). The schemes have a second order accuracy in time and space.

### 2.2 Numerical domain and inflow

The three geometries taken into account in our study were created by using a commercial mesh generator developed by [4].

The following parameters were chosen constant for the different geometries: the dimensional height \( H \) of the rectangular pipe is equal to 0.025 m, the aspect ratio \( l/H \), where \( l \) represents the width of the rectangular pipe is equal to \( l/H = 4 \). The uniform dimensional inlet velocity is constant and equal to \( U_0 = 2.5 \text{ m.s}^{-1} \). Therefore, the Reynolds number based on the inlet velocity and the height \( H \) is equal to \( Re = 4000 \). The nondimensional time increment corresponds to \( \Delta t = 10^{-3} = \frac{U_0}{H} \Delta t \).

In the case of a rectangular straight pipe, the computational domain is \([-2H, 2H] \times [0.2H, 1.2H] \times [0, 4H] \) respectively in \( y_1, y_2, y_3 \) direction. The vertical straight pipe has a computational domain \([0, 2.5H] \times [0.2H, 1.2H] \times [0, 4H] \) respectively in \( y_1, y_2 \) and \( y_3 \) direction. The vertical straight pipe has a computational domain \([-1.2H, -2H] \times [0H, -2.5H] \times [0H, 4H] \) in \( y_1, y_2 \) and \( y_3 \) direction. The origin is located on a bottom side of the horizontal rectangular straight pipe. The rectangular curved pipe is a quarter ring with an inner radius of 0.2H and an external radius of 1.2H linking the pipes. 249 elements are used along the streamwise direction, 74 elements along the spanwise direction and 100 along the lateral direction. In this case, the origin is located at the bottom side of the horizontal rectangular straight pipe. The total number of computational elements is approximately 750 000. As for previous cases, the same inlet and boundaries conditions are used.

For the following case, an obstacle representing a schematic tooth is imposed. The dimensions of the computational domain in which the tooth is inserted is \([-2H, 2H] \times [0, H] \times [0, 5H] \) respectively in \( y_1, y_2, y_3 \) direction, where \( y_1, y_2, y_3 \) represent respectively the lateral, the spanwise, and the streamwise coordinates. Structured meshes are used. The origin is located on the inlet face, at the centre of the lateral plan and the bottom side of the rectangular pipe, as shown in Fig. 2 (a). 74 elements are used in the streamwise direction and are uniformly distributed. 74 elements are used in the spanwise direction towards a symmetric tangent hyperbolic distribution. \( \Delta y_2 \), which represents the size of the first element near the wall is equal to \( \Delta y_2 = 0.005H \). 50 elements are used in the spanwise direction. The total number of computational elements is approximately 280 000. The inlet velocity is imposed to be uniform. Non-slip condition was used on the wall. Non-reflective conditions are used as exit conditions.

For the following case, an obstacle representing a schematic tooth is imposed. The dimensions of the computational domain in which the tooth is inserted is \([-2H, 2H] \times [0, H] \times [0, 4.05H] \). The tooth is located at a distance equal to \( y_1 = 16/5H \). The geometry of the tooth is a trapezoid for which the base is \( 1.25/5H \) and its opposite side has a length of \( 0.25/5H \); the upstream and downstream sides of the tooth make respectively an angle of 105° and 90° with the bottom wall. Its height is equal to \( 7/10H \). Approximately 1 880 000 elements were used to compute the flow evolution. Structured meshes are still used. The computational domain is divided into 5 blocks as seen in Fig. 2 (b). Close to the obstacle the grid meshes are refined in order to be small enough to resolve the boundary layer and bound vortices around the obstacle. The origin is located at the same position as for the previous case. The same inlet and boundaries conditions imposed for case 1 are used.

The third geometry is a rectangular curved straight pipes linked with a curved pipe. The horizontal straight pipe has a computational domain \([0, 2.5H] \times [0.2H, 1.2H] \times [0, 4H] \) respectively in \( y_1, y_2 \) and \( y_3 \) direction. The vertical straight pipe has a computational domain \([-1.2H, -2H] \times [0H, -2.5H] \times [0H, 4H] \) in \( y_1, y_2 \) and \( y_3 \) direction. The origin is located on a bottom side of the horizontal rectangular straight pipe. The rectangular curved pipe is a quarter ring with an inner radius of 0.2H and an external radius of 1.2H linking the pipes. 249 elements are used along the streamwise direction, 74 elements along the spanwise direction and 100 along the lateral direction. In this case, the origin is located at the bottom side of the horizontal rectangular straight pipe. The total number of computational elements is approximately 750 000. As for previous cases, the same inlet and boundaries conditions are used.

\[ \begin{align*}
\text{Fig. 2 Typical pipe geometries, a) rectangular straight b) rectangular straight + obstacle, c) rectangular curved.}
\end{align*} \]
3 Results and discussion

3.1 Rectangular straight pipe

The first results concern the basic representation of a rectangular straight pipe. This representation is viewed as the simplest simplification of the vocal tract. No perturbation due to upstream obstacle is assessed; therefore the flow is laminar. The main interest of such geometry resides in the certification of the code under use and as a reference case.

![Fig. 3](image1)

**Fig. 3 Mean velocity evolution along the spanwise direction for different streamwise localisation.** a) streamlines dimensional velocities $U_3$, b) mean velocity normalised with centreline velocity $U_{c3}$.

Fig. 3(a) and 3(b) represent the mean velocity along the spanwise direction. Fig. 3(b) shows the evolution of the streamwise component of the velocity normalised by its centreline velocity $U_{c3}$. As the downstream localisation increases, the rectangular velocity profile imposed at the inlet tends to be transformed into the parabolic Poiseuille profile. In order to compare with the Poiseuille profile the origin of the axis where shifted to the centre of the rectangular pipe.

\[
U_3 = \frac{3}{2} U_0 \left(1 - \frac{y_3^2}{H^2} \right) \quad (9)
\]

This evolution confirms boundary layer grows on both wall, due to viscosity, and mass-conservation.

![Fig. 4](image2)

**Fig. 4 turbulent intensity $u_{tr}/U_3$ for different streamwise localisation.**

As a further analysis, Fig. 4 presents the turbulence intensity $u_{tr}/U_3$, where $u_{tr}$ represents the rms value of the streamwise component of the velocity along the rectangular straight pipe. The turbulence level is increased with the downstream localisation. The maximum values are observed close to the wall and comprised between $5\% < u_{tr}/U_3 < 30\%$.

3.2 Rectangular straight pipe with an obstacle

This subsection presents the impact of an obstacle on the laminar inflow. Fig. 5 shows as the downstream location increases the flow becomes asymmetric. From $y_3/H = 1.8$, the position of the velocity maximum $U$ is shifted to the flat wall. From Fig. 5 the flow maximum of the velocity is localised in an area which starts downstream the end of the obstacle and is equal to about 4 times the inlet velocity.

![Fig. 5](image3)

**Fig. 5 Contourlines of the dimensional mean velocity $U$ [m.s$^{-1}$].**

Fig. 6, for which the axis are similar to Fig. 3, shows the flow evolution along the obstacle and reveals the existence of a recirculation zone, by the presence of negative velocities. The velocities in the recirculation zone are comprised between $-0.75 \text{ m.s}^{-1} < U < 0 \text{ m.s}^{-1}$. 

3964
Fig. 6: Dimensional mean velocities for different streamwise localization.

Fig. 7 presents the mean velocities profiles along the obstacle at different distance $\Delta E_1$, where $\Delta E_1$ represents the distance between the location of the velocity measurements and the edge $E_1$. The flow is asymmetric, compared with the velocity profile before and after the obstacle, the maximum velocity is close to the obstacle and accelerated along the singularity.

Fig. 8: Turbulent intensity profiles at different location $\Delta E_1$.

For which $Re = 9600$ and $b = 9.6 \times 10^{-3}$ m [1]. Although, the profiles are close to the obstacle, it seems the flow tends to present a self-similar evolution comparable to a wall jet.

Fig. 9: Mean velocity profile at different downstream location $\Delta E_2/b$ 1 from the edge ($E_2$).

3.3 Rectangular curved pipe

The study of a curve, is motivated by the geometry of the vocal tract. In this section, we briefly give a qualitative presentation of the flow behaviour imposed by the curvature. From Fig. 10 a) the curve implies an increase of the velocity from the beginning of the curvature and a stall of the boundary layer leading to a recirculation zone which is zoomed in on Fig. 10 b). In Fig 11 it appears the curvature generates turbulent velocities area which propagates downstream the geometry. The magnitude turbulent velocities are comprised between $0 \text{ m.s}^{-1} < u_{tur} < 1.861 \text{ m.s}^{-1}$ which represent about 20% of the Mean velocity.
and lead to the creation of a recirculation zone. Therefore, further investigations are needed concerning the degree of constriction, the shape of the obstacle, the distance between the obstacle and the outlet.

**Acknowledgments**

The support of the French Rhones-Alpes region (CIBLE 2006), Agence National de la Recherche (ANR-07-JJCJC-005), College Doctoral Franco-Japonais (X. Grandchamp), and Japanese Society for the Promotion of Science (JSPS fellowship A. Van Hirtum) are gratefully acknowledged.

**References**


[3] *FrontFlow/Blue version 4.0 User Manual*. Collaborative Research Center for Computational science and Technology, Institute of Industrial Science, the University of Tokyo, JAPAN. software@rss21.iis.u-tokyo.ac.jp

[4] *Gridgen Tutorial workbook*. version 15 volume3 gridgen@pointwise.com


