European Journal of Mechanics B/Fluids 63 (2017) 1–8

Contents lists available at ScienceDirect

European Journal of Mechanics B/Fluids

journal homepage: www.elsevier.com/locate/ejmflu

Constricted channel flow with different cross-section shapes

A. Van Hirtum [a,*], B. Wu [a], H. Gao [b], X.Y. Luo [b]

[a] GIPSA-lab, CNRS UMR 5216, Grenoble University, France

[b] School of Mathematics and Statistics, University of Glasgow, Glasgow G12 8QW, United Kingdom

ARTICLE INFO

Article history:
Received 6 November 2015
Received in revised form
24 November 2016
Accepted 22 December 2016
Available online 12 January 2017

Keywords:
Analytical model
Numerical simulation
Experimental validation
Moderate Reynolds number flow

ABSTRACT

Pressure driven steady flow through a uniform circular channel containing a constricted portion is a common problem considering physiological flows such as underlying human speech sound production. The influence of the constriction's cross-section shape (circle, ellipse, circular sector) on the flow within and downstream from the constriction is experimentally quantified. An analytical boundary layer flow model is proposed which takes into account the hydraulic diameter of the cross-section shape. Comparison of the model outcome with experimental and three-dimensional numerically simulated flow data shows that the pressure distribution within the constriction can be modeled accurately so that the model is of interest for analytical models of fluid–structure interaction without the assumption of two-dimensional flow.

1. Introduction

Pressure driven channel flow is associated with physiological flows for which constricted channel portions occur either naturally or due to a pathology. Well-known examples are airflow through the human airways (speech production, asthma, obstructive sleep apnea) or blood flow through a stenosis. Physical studies of these flows often rely on mechanical replicas aiming to reproduce phenomena experimentally in a repeatable and controllable way. Such replicas allow a systematic investigation of the potential effect of geometrical parameters on the flow. In the case of human speech production, a partly constricted channel is used as mechanical replica representing a severe simplification of portions of the larynx and vocal tract, see *e.g.* [1–6]. The potential impact of the constriction degree and streamwise position of the constricted channel portion is evaluated whereas the impact of the cross-section shape is not considered. Indeed, in [1–6] a rectangular cross-section shape is used for which the spanwise dimension is not varied. Data obtained when mounting mechanical replicas to a suitable experimental setup are than used in order to validate flow models. In the case of speech production and even more general in the case of physiological flows [7,8], simplified analytical flow models are often sought since they favor to assess the impact of well defined and physiologically meaningful input parameters. In the following we consider flow and geometrical configurations relevant to physical studies of human speech production.

Simplifications of the flow model through the constricted channel geometry are based on a non-dimensional analysis of the governing Navier–Stokes equations [9]. Accounting for typical values of physiological, geometrical and flow characteristics observed on human speakers [10,11,3] result in non-dimensional numbers which allows one to assume the flow through the human upper airways during speech production as incompressible [Mach number, $Ma^2 \ll 0.1$], laminar [Reynolds number $Re \approx O(10^3)$], quasi-steady [Strouhal number $Sr \ll 1$] and two-dimensional (2D) given the channel's mean aspect ratio ($Ar \geq 4$) corresponding to the width-to-height ratio of the rectangular cross-section [2,12,6]. Based on these assumptions, quasi-2D or 2D flow models [13–15] derived from boundary layer theory [9] have proven to be extremely useful to capture the underlying physics and are applied to mimic and predict ongoing phenomena using few computational resources while allowing experimental validation on mechanical replicas. Nevertheless, the assumption of a 2D geometry implies that details of the cross-section shape perpendicular to the streamwise flow direction (x) are neglected whereas medical imaging studies of the human upper airway during speech production reveal a large variation of the cross-section shape within the larynx as well as within the vocal tract [10,11].

Previous studies (see *e.g.* [2,13,14]) have shown that in order to represent the flow observed on mechanical replicas of the human upper airway, the contribution of viscous effects to the model outcome is essential. Since viscous effects depend on the cross-section shape [9], adding the constriction's cross-section shape to the set

* Corresponding author.
E-mail address: annemie.vanhirtum@grenoble-inp.fr (A. Van Hirtum).

http://dx.doi.org/10.1016/j.euromechflu.2016.12.009

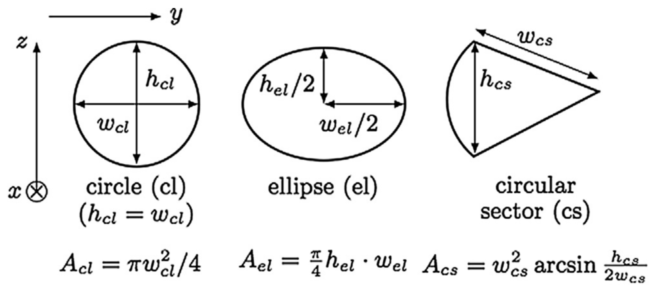

Fig. 1. Schematic front view ((y, z) plane) and area A for circle (cl), ellipse (el) and circular sector (cs). Spanwise extent w (y-direction) and transverse extent h (z-direction) are indicated.

of flow model input parameters is likely to alter the model outcome. Concretely, this implies that the common assumption of 2D flow is not made whereas other flow assumptions, *i.e.* incompressible, laminar and quasi-steady are still valid. Nevertheless, experimental evidence of the influence of the constriction's cross-section shape on mechanical replicas is lacking.

Therefore, in the current work it is aimed firstly to provide experimental evidence of the impact of the cross-section shape on main flow quantities for a mechanical replica pertinent to represent an upper airway constriction during speech sound production [1,6,14]. The mechanical replica has a uniform constriction of constant area but variable cross-section as depicted in Fig. 1. Secondly, it is aimed to propose and validate an analytical laminar boundary layer flow model against measured flow quantities. Furthermore, the outcome of the laminar flow model is compared to numerically simulated data describing laminar three-dimensional flow. Simulated data are obtained using the immersed boundary (IB) method [16] since it is commonly used to describe fluid–structure interactions such as occurring during human speech sound production [17]. Consequently, modeled flow data are compared with measured as well as simulated flow data. Note that for fluid–structure interaction applications such as vocal fold auto-oscillations during voiced speech production, accurate estimation of the pressure within the constriction ($x_1 \le x \le x_2$) is most important since it determines the forces exerted by the fluid on the enveloping walls.

In the following, a simplified analytical boundary layer flow model is presented accounting for the cross-section shape by means of its hydraulic diameter. Next, the immersed boundary method to obtain 3D flow simulations is detailed. After that, the experimental setup used to characterize the flow field (velocity and pressure) of the mechanical replica is detailed. Then, results are presented. At first, the impact of the cross-section on measured flow data is shown and the quasi-3D flow model is discussed with respect to experimental observations. Subsequently, a quantitative comparison between measured, modeled and simulated flow data is presented. Finally, a conclusion is formulated.

2. Laminar boundary layer flow model

Flow through a uniform circular flow channel of area A_0 containing a constriction of constant length L_c and with minimum area A_c is considered as illustrated in Fig. 2. All sharp edges are rounded (measured radius $r = 0.05$ cm). Rounding outlet edges with such a small curvature radius will avoid occurrence of the so-called Coanda effect which is not the case when the constriction outlet is more divergent [9]. Rounding leading edges at the inlet of the constriction aims to reduce a potential vena contracta effect in comparison with sharp inlet edges. Consequently, rounded edges allow to focus on the impact of the cross-section area by reducing other effects related to the constriction geometry. Finally, it is noted that with respect to physiological flow applications rounded edges are more pertinent.

Flow through the constricted channel is then generated by imposing upstream pressure P_0 and hence the total driving pressure difference $\Delta P = P_0 - P_d$ where downstream pressure $P_d = 0$. Downstream from the constriction jet formation occurs at the downstream end of the constricted region ($x = x_2 - r$) so that the jet has a finite potential core x_{pc} due to flow mixing and subsequent pressure recovery as depicted in Fig. 2. The impact of the wall curvature of the trailing portion of the constriction on flow separation is neglected since the wall curvature radius of the assessed cross-section shapes (Fig. 1) is much smaller (factor 5 or more) than the overall spanwise or transverse extent (see *e.g.* $w_.$ and $h_.$ in Table 1). The pressure distribution $P(x, t)$ along the channel ($x_0 \le x$) is sought.

Boundary layer development influences the flow of a viscous fluid such as air (kinematic viscosity $\nu = 1.5 \times 10^{-1}$ cm^2/s and density $\rho = 1.2 \times 10^{-3}$ g/cm^3). The constriction length L_c is assumed much shorter than the entrance length necessary to obtain fully viscous flow (developed boundary layers) for all assessed Reynolds numbers Re [9] so that thin boundary layers envelop the core region of the flow. Therefore, a boundary layer flow model is proposed.

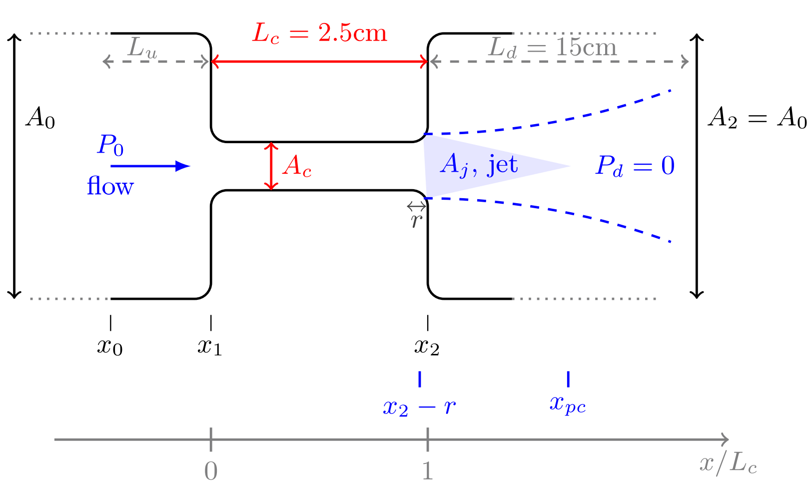

Fig. 2. Illustration of pressure driven flow through a uniform circular channel (area $A_0 = 4.9$ cm^2) enveloping a constricted portion (area $A_c = 0.79$ cm^2 and length $L_c = 2.5$ cm) for which the cross-section shape can be varied (Fig. 1). All sharp edges are rounded (measured radius $r = 0.05$ cm). Main streamwise direction x, pressure upstream from the constriction P_0, pressure downstream from the constriction P_d, upstream unconstricted channel portion length (L_u) and downstream unconstricted channel portion length ($L_d = 15$ cm) are indicated. A developing jet (dashed curved lines) of area A_j with finite potential core extent x_{pc} (shaded area) is depicted.