Numerical simulations of glottal flow

Liljencrans, J.

journal: STL-QPSR
volume: 30
number: 1
year: 1989
pages: 069-074

http://www.speech.kth.se/qpsr
NUMERICAL SIMULATIONS OF GLOTTAL FLOW

Johan Liljencrants
Dept. of Speech Communication and Music Acoustics, KTH

Abstract

Features of glottal flow are visualized by numerical simulation, using a Navier-Stokes solver for two-dimensional, incompressible flow. Four different shapes of glottal passages are examined, for comparison to stationary model measurements. Results include pressure distributions leading to the forces acting on the vocal folds and illustrate entry pressure loss and exit recovery. Flow field images also suggest where the commonly used discrete elements of resistance and inductance are geometrically located.

INTRODUCTION

Classical work like van den Berg, Zantema, & Doornenbal (1957) describes the glottal pressure drop using empirical terms like entry and exit recovery coefficients. These are difficult to predict since they vary with the channel shape. The details of glottal airflow are also hard to access experimentally due to the small dimensions.

This study illustrates features of the glottal flow by numerical simulation of the fluid motion, an alternate experimental method. Viscosity is accounted for and is primarily important to the flow phenomena. For practical reasons the present scope is otherwise limited but still covers many important effects: The channel is given a static shape, in reality the air velocity is generally a few orders of magnitude higher than that of the walls, so dynamic behaviour can largely be inferred from static data sequences. The flow is taken as two-dimensional, axial and at right angle to the vocal passage, assuming it to be invariant in the third dimension along the slit. The medium is taken to be incompressible since the differential pressures are two orders of magnitude below the static atmospheric pressure.

SIMULATION METHOD

The simulations have been carried out as outlined by Roache (1971) using the Navier-Stokes’ equations cast as a transport equation for vorticity ω

\[
\frac{\partial \omega}{\partial t} = - u \frac{\partial \omega}{\partial x} - v \frac{\partial \omega}{\partial y} + \frac{\mu}{\rho} \left( \frac{\partial^2 \omega}{\partial x^2} + \frac{\partial^2 \omega}{\partial y^2} \right)
\]

This is the central vehicle of the simulation as it shows the development in time of the vorticity (which is a scalar property measuring the local shear rate) under influence of the velocity components u, v in the x, y directions, in turn found from space differentiation of the stream function Ψ. The first two right-hand terms are called the advection terms and show how the vorticity pattern is carried along by the flow. The last term is the diffusion term and shows how inhomogenieties in vorticity will spread out into their vicinity. The relative importance of the terms depends on the Reynolds’ number.

A geometrical mesh for the bounding walls and the enclosed simulation area is laid out on a 65 * 49 point matrix in the general axial and cross directions. The mesh is elastically distorted so that the upper and lower rows of nodes become located on the
boundary. This curved mesh enhances resolution at critical locations and is computationally economic, but causes technical difficulty in evaluating differences. Each node entry in the matrix specifies the x and y coordinates and some reasonable vorticity \( \omega \) and stream function \( \Psi \). In the present program, coded in C language for the Domain Apollo DN 10000 computer, all simulation procedures can be geared to use a resolution reduced by the factors 2, 4, or 8. This is useful for debugging and test runs since each doubling of resolution will increase computation time 16-fold. Gearing is also used to quickly arrive at an approximate solution which is subsequently interpolated and refined at higher resolution.

Once the mesh has been settled the actual simulation is done in the three following basic steps:

1. **Establish boundary values** of \( \Psi \) and \( \omega \). Along the walls the stream function \( \Psi \) has a constant value determined by the total flow. Initial and forthcoming conditions for the inflow boundary are taken to be a Poiseuille flow (the classical flow in a straight channel with a parabolic velocity profile over the cross section) while \( \Psi \) at the outflow boundary is determined by extrapolation from the interior. Vorticity \( \omega \) is generated due to viscous shear at the walls and is evaluated from the stream function and the condition of zero velocity at the wall. This vorticity generation actually drives the simulation problem.

2. **Transport the vorticity**. At all interior nodes the velocities \( u \) and \( v \) are evaluated from \( \Psi \) and the transport equation is applied to find \( \omega \) at a later time \( t + \Delta t \). Initially vorticity will spread to the interior nodes by diffusion from the walls. Interior vorticity will diffuse further and is also advected downstream with the flow.

3. **Modify the stream function**. With the new vorticity distribution \( \omega \) the stream function has to be modified at all internal nodes so that the Poisson equation \( \Delta^2 \Psi = \omega \) (from the definition of vorticity) is satisfied. This is done with an iterative ‘successive over relaxation’ process that consumes most of the total computing time.

These steps are reiterated until a sufficient time span has been covered. Sometimes the process converges into a stable solution, in others the solution may vary with time, the flow problem itself is often unstable.

The pressure contour along the walls is finally integrated from \( \delta \psi / \delta s = \mu \delta \omega / \delta n \), \( s \) and \( n \) being the directions along and normal to the wall, and the interior pressure field is found from surface integration of the fundamental N-S equations from the now known \( \Psi \) and \( \omega \). This procedure poses some difficulty and may call for revision in the present pressure data though tests with a straight channel give excellent accordance with Poiseuille flow.

The output is sequences of graphs of field variables, examples are shown in Fig. 1.

**SPATIAL LOCATION OF FLOW RESISTANCE AND INDUCTANCE**

It is illuminating to note how flow variable images can show the effective distribution in space of resistance and inductance, conventionally modelled as discrete circuit elements.

Within a small volume with essentially unidirectional sheared flow we can apply the elementary formula for the force \( F \) between two surfaces of area \( S \) sliding past each other at a distance \( d \) with velocity \( u \) in a medium of viscosity \( \mu \):
Fig. 1. ISO contours of field variables 25 ms after onset of the unstable flow through a 1 mm wide diverging slit. The average velocity is 20 m/s in the constriction, corresponding to Reynolds' number 1300, based on the cross width. From top to bottom: stream function (stream lines), velocity in steps of 2 m/s, vorticity, and pressure in steps of 20 Pa (2 mm H₂O).
to develop an expression for the dissipation in Watts per unit volume of the medium, namely

$$w_d = \mu (du/dn)^2 \quad [ \text{Nm/s}^2 \text{m}^3 = \text{W/m}^3 ]$$

where \( n \) is normal to the flow. But for this laminar flow \( du/dn \) by definition equals the vorticity, so the \textit{vorticity squared} will directly display where flow energy is dissipated, mostly in the boundary layer at the passage walls and in the outskirts of the following jet, see Fig. 1c. This would in a sense display where the flow resistance is located.

The kinetic energy density (formally like the 'dynamic pressure') is

$$p_k = \rho u^2 / 2 \quad [ \text{Ws/m}^3 = \text{Nm/m}^3 = \text{Pa} ]$$

and the \textit{velocity squared} shows this as located in the middle of the passage and downstream the jet, see Fig. 1c. Modelling with an inductance is problematic because of the nonlinearity. A linear inductance would require pressure and velocity to be proportional; here we will have to assume an inductance proportional to \( u \).

RESULTS

In the initial phase a very strong vortex is formed downstream the slit as the jet develops (a 'smoke ring'). In low velocity cases this vortex widens and slows down and the flow becomes stationary. With higher velocities the jet often oscillates and sheds vortices to alternate sides.

Gauffin & Liljencrants (1988) show some measured data for similarly shaped channels, where the drop is not proportional to the ideal dynamic pressure. Drop versus velocity characteristics also differ in slope depending on whether the channel is convergent, straight, or divergent. Preliminary results of the present simulations essentially conform with these measurements even if some question remains on the magnitude of the drop. The principal irregularities appear to come in the pressure recovery, much connected with the point of flow separation.

To illustrate how several effects contribute to the pressure drop, Fig. 2c shows the pressure profile in a straight passage. A sharp drop occurs at the entry where the air is violently accelerated. Inside the entry corner the flow separates from the wall and some pressure is recovered until the flow reattaches after approximately 1 mm. Then pressure gradually drops again due to viscous friction. The exit pressure peak is a common feature but may be exaggerated from a numerical artifact. The pressure variations in the wide outlet area are generally quite small, even with the fairly strong irregular vortices present in this case.

There are several questions left for future study. A technical one is if the jet oscillations are significantly influenced by numerical inaccuracy, for instance in the Poisson equation solution. In relation to speech production continued simulations may provide comprehensive data on the forces acting to move the vocal folds, they may reveal the possible importance of the transient vortices as an excitation of the vocal tract, and shed some light on the notorious problem of source amplitude and location for the generated noise.
Fig. 2. Pressure distributions in four differently shaped slits: convergent, divergent, straight, and combined convergent-divergent. The average velocity $v_0$ is 6 m/s in the constriction, corresponding to Reynolds' number 400. The vertical scale is in units of nominal dynamic pressure $0.5 \rho v_0^2$ and the geometric area is displayed in perspective, from the center line to the wall.
Acknowledgments
This work has been supported in part by grants from the Swedish Board for Technical Development.

References
