STEADY FLOW THROUGH MODELED GLOTTAL CONSTRICTION

F. Alipour

Department of Speech Pathology and Audiology

V. C. Patel

Department of Mechanical Engineering &
Iowa Institute of Hydraulic Research

The University of Iowa
Iowa City, Iowa
USA

Abstract The airflow in the modeled glottal constriction was simulated by the solutions of the Navier-Stokes equations for laminar flow, and the corresponding Reynolds equations for turbulent flow in generalized, nonorthogonal coordinates using a numerical method. A two-dimensional model of laryngeal flow is considered and aerodynamic properties are calculated for both laminar and turbulent steady flows. Three configurations of vocal folds, corresponding to convergent, parallel or rectangular, and divergent glottis are used to study the effects of glottal shape on the airflow. Results are reported for several types of calculations. Results are obtained for the velocity and pressure distributions, and friction coefficient and turbulent kinetic energy in the constriction. Flow separation was found to exist even at low Reynolds numbers, and the separation region grew in size with increasing Reynolds number. The glottal shape has a marked influence on the flow pattern and the pressure drop.

Key Words Airflow, Glottal Constriction, Computer Simulation, Voice Production, Navier-Stokes Equation, Aerodynamics, Laryngeal Flow.

چکیده جریان هوا در مدل دهانه حنجره بوسیله حل معادلات نویر-استوکس برای جریان آرام و معادلات رینولد برای جریان مغشوش در مختصات عمومی غیر متعامد بوسیله یک روش عددی حل شده است. یک مدل دوبعدی جریان حنجره درنظر گرفته شده و خواص آئرودینامیکی برای جریان های پایدار آرام و مغشوش محاسبه گردیده اند. سه شکل تارهای صوتی مطابق با دهانه حنجره همگرا، موازی و واگرا برای مطالعه اثرات دهانه حنجره برروی جریان هوا مطالعه گردیده اند. نتایج برای چندین محاسبه مختلف گزارش شده اند. نتایج توزیع سرعت و فشار، ضریب اصطکاک و انرژی سینتکی جریان مغشوش در مجرا بدست آمده اند. جدایش در جریان حتی در محدوده اعداد رینولد پائین مشاهده گردیده است و اندازه ناحیه جدایش با عدد رینولد افزایش می یابد. شکل دهانه حنجره اثر قابل توجهی برساختار جریان و افت فشار دارد.

INTRODUCTION

The study of airflow through glottal constriction has received a lot of attention in recent years due to its influence on the vocal-folds oscillations and voice production. This kind of flow causes a large pressure drop due to the acceleration of the airflow in the constriction, pulls the folds together and initiates the pulsatile flow and vocal fold oscillations. This oscillation which is coupled with airflow is the major source of sound in the human speech production system. The glottal constriction has a complicated three dimensional geometry which is not accessible to instrumentation. Thus a computational method is the approach used in this study of airflow in the glottis.

Previous studies of the airflow in the larynx have been limited to one-dimensional flow models with the assumption of uniform velocity profile across the glottis. Most of the experimental work was aimed at measurement of pressure drops and volume flow rates on static models of a larynx [1-6]. The control parameters in these static models were mostly the glottal configurations (converging, parallel, or diverging glottis) and volume flow rate. These investigators measured pressure drops along the glottal airway and developed empirical formulas for the pressure drop as a function of volume flow rate for various configurations. These empirical relations have been used for computer simulation of speech using modified Bernoulli equations.

Theoretical calculations of the airflow in the larynx have been based on the assumption of potential flow or fully-developed laminar flow [2]. Only in the last few years, have numerical studies with methods of computational fluid dynamics emerged [7,8]. Lijima *et al.* [7] solved the two-dimensional Navier-Stokes equations in primitive variables with a finite-element method and reported velocity and pressure profiles for steady and unsteady laminar flow at Reynolds numbers in the range 2 to 64. Liljencrants [8] also used a finite-element method but with formulation of vorticity transport equation, and reported velocity contours and pressure profiles for unstable flow through a converging-diverging glottis. These recent numerical studies, although limited in scope, are most encouraging because they demonstrate the potential of emerging methods of computational fluid dynamics to resolve details of the flow that are not accessible to measurement.

In this study we have used a state-of-the-art numerical method to investigate laminar and turbulent flows of air in two-dimensional models of the glottal constriction. The geometry of the glottis and the airflow rate (or Reynolds number) were varied to study their effect on the pressure and velocity distributions.

NUMERICAL MODEL

The starting point for the present research was provided by the work of Patel, Chon and Yoon [9], who calculated the flow in channels with wavy walls. Their numerical method, adapted for the present study, solves the Navier-Stokes equations for laminar flow, and the corresponding Reynolds equations (sometimes referred to as the Reynolds-averaged Navier-Stokes, or RANS equations) for turbulent flow in generalized, nonorthogonal coordinates, namely,

$$V^i_{,i} = 0 \tag{1}$$

$$\partial V^i/\partial t + V^m V^i_{,m} = F^i - \rho^{-1} g^{ij} P_{,j} - (v^m v^i)_{,m} + \nu g^{mn} V^i_{,mn} \tag{2}$$

where V^i is the mean velocity, v^i is the velocity fluctuation, F^i is a body force, p is pressure, ρ is density, ν is kinematic viscosity, and g^{ij} is the conjugate metric tensor. These equations have to be expressed in a coordinate (grid) system that is appropriate for the particular problem to be solved. Here, there are two options, and both have been followed in CFD. In one, only the coordinates are transformed, leaving the velocity components in a convenient orthogonal system in the physical domain. Such a partial transformation has the advantages that fewer geometrical terms arise in the transformed equations, reducing computing times and storage, and the results of the calculations are easily interpreted. The disadvantage is that numerical diffusion may become significant in regions where the angle between the velocity component and the coordinate lines becomes large. This effect, however, depends upon the numerical scheme that is adopted for the solution of the flow equations and cannot be separated from it.

In laminar flow, the third term on the right hand side of Equation 2 is absent and Equations 1 and 2 can be sloved. For turbulent flows, these equations have to be supplemented by a turbulence model which relates the six Reynolds stresses ($v^m v^i$) to the mean flow V^i. Insofar as applications to three-dimensional flows are concerned, very few new developments have taken place in this area. In the present approach, we use the well-known k-ε turbulence model. This model introduces an eddy-viscosity

$$v^m v^i = (-2k/3)\, g^{ij} + \nu_t (g^{im} V^j_{,m} + g^{jm} V^i_{,m}) \tag{3}$$

which relates the turbulent stresses to the corresponding mean rates of strain. Here, ν_t is the eddy visocosity, and k is the turbulent kinetic energy given by

$$2k = v^i v_i = g_{ij} v^i v^j \tag{4}$$

In the k-ε model, the eddy viscosity is related to the turbulent kinetic energy k and its rate of dissipation ε by

$$\nu_t = C_\mu k^2/\varepsilon \tag{5}$$

where C_μ is a constant, and k and ε are determined from two modelled transport equations, which contain four additional constants. Equation 5 can be expressed as

$$\nu_t/\nu = C_\mu \sqrt{k}\lambda/\nu = C_\mu R_T \tag{6}$$

where R_T is a turbulence Reynolds number based on the velocity and length scales, $\sqrt{k}$ and $\lambda = k^{3/2}/\varepsilon$, respectively. Thus, the k and ε equations provide the velocity and length scales at each point in the flow. One of the advantages of this model then resides in the fact that it does not require the specification of a length scale. This is one of the reasons for the widespread use of the model in diverse applications. A rather serious drawback of the standard k-ε model is that it applies only in regions where the flow is fully turbulent, i.e., where R_T is large. This excludes the near-wall region, occupied by the sublayer and the buffer layer. This difficulty is usually overcome by applying what are called "wall functions" in place of the no-slip condition; these involve the use of the law-of-the-wall and other relations deduced from equilibrium assumptions in two-dimensional flows at some point in the logarithmic layer. The use of wall functions in complex three-dimensional flows remains a source of uncertainty.

The coordinates are generated by numerical solution of a pair of Poissons equations in the computational domain of interest. The momentum equations are discretized using analytic solutions of the linearized equations, according to the finite-analytic method of Chen [10], and the pressure-velocity coupling is made through the continuity equation using a modified form of the SIMPLER algorithm of Patankar [11] which requires the use of a staggered grid arrangement. The time-marching capability of the method makes it suitable for the solution of unsteady flows, but here it has been used only to obtain steady-state solutions, using time as an iteration parameter.

Since the major shapes that are observed during the vibratory motion of vocal folds are convergent, rectangular, and divergent glottis, laryngeal configurations with various angles of convergence and divergence angles were used to study the effect of glottal shape on the flow variables. The glottal shape was approximated by two sinusoidal curves and a straight line with smoothing at their boundaries. The flow was assumed to be symmetric with respect to the midsagittal plane (centerline in 2-D flow) and, therefore, only half of the flow domain was calculated.

RESULTS

Calculations were performed for three glottal configurations, namely, convergent, rectangular and divergent glottis. Typically, a 150 x 50 grid as shown in Figure 1 was employed in the longitudinal and transverse directions. The grid was refined near the wall to resolve the large velocity gradients there. Also, a finer horizontal grid was used around the constriction where variations in pressure and velocities are expected to be rapid. The discretized equations were solved with iterative methods with convergence criteria based on the accuracies of the results. The inlet velocity profile and exit pressure were considered as known boundary conditions. In the laminar flow calculations, a parabolic profile corresponding to fully developed flow was used as an inlet condition. In the turbulent flow calculations, the fully developed profile was calculated in a straight channel and stored for each Reynolds number in a data file. This data was read in as inlet turbulent velocity profile for glottal constriction. Another possibility is to model the inlet profile with logarithmic law of the wall.

Results on Laminar Flow

Figure 2 shows the flow pattern in a convergent glottis at a Reynolds number of 500. The flow separates at the rear of the constriction and an eddy has formed in each case. Similar patterns exist for various constriction sizes, but as the opening decreases, the size of the eddy increases, the point of reattachment of the flow moves further downstream, and the point of separation moves further up-

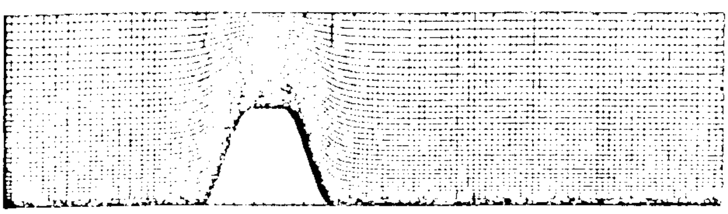

Figure 1. Solution domain and grid pattern in rectangular constriction.

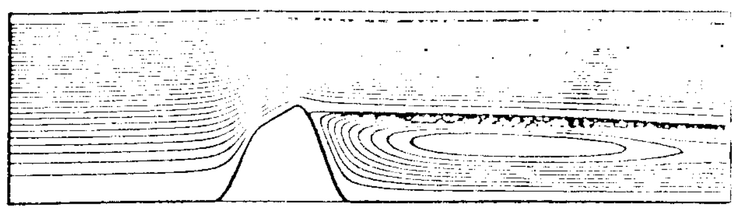

Figure 2. Laminar flow pattern in a convergent constriction.

stream. This results in more horizontal streamlines after the constriction in the case of large blockage. The overall flow in rectangular and divergent constriction is similar to those of convergent constriction and the location of flow separation and the size of the eddies are influenced by the glottal shape.

Figure 3 shows the velocity profiles at a few selected stations. As the flow approaches the constriction, a strong shear layer (region of steep velocity gradients) develops over the tope of the constriction. This shear layer is evident between the reverse flow and jet issuing from the constriction.

In all constrictions, the pressure is almost uniform across the flow until some distance ahead of the constriction. A variation across the flow takes place as the streamlines begin to curve up the constriction. As the flow accelerates, there is a commensurate fall in pressure. Following the constriction, the pressure is negative, and almost uniform across the region of separation as well as the jet issuing from the constriction. This sharp pressure drop in the constriction is accompanied by an increase of friction coefficient due to the flattening of velocity profiles near the constriction. The pressure variation across the flow near the constriction was enhanced as the size of constriction became smaller. It is also observed that the divergent glottis leads to a larger pressure drop than a convergent glottis.

Figure 4 shows the effect of Reynolds number on the wall friction for the rectangular constricion. As the Reynolds number increases, the friction coefficient decreases. This result also shows that point of separation (zero wall friction) moves towards the upstream as the Reynolds number is increased.

Results on Turbulent Flow

Figure 5 shows a typical turbulent mean flow pattern in a divergent constriction. While there is some similarity between this pattern and that of laminar flow, there are major differences such as: the existence of an eddy before the constriction. The size of the eddies are much smaller than laminar flow and are not affected very much by Reynolds number. Finally, the friction coefficient is negligible everywhere except within the constriction.

Typic horizontal velocity profiles for a divergent constriction are shown in Figure 6. The profile becomes flat

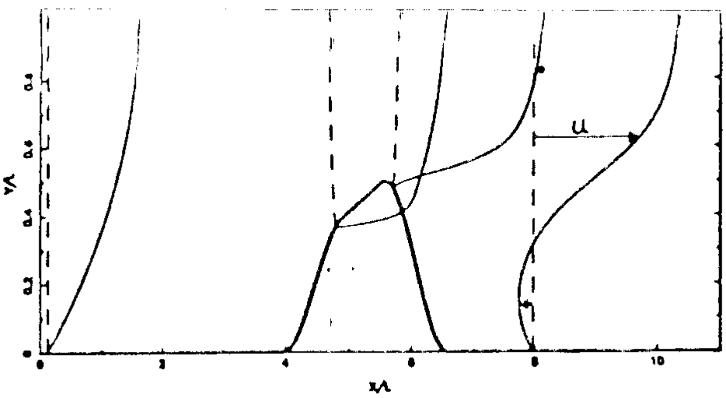

Figure 3. Velocity profiles in a laminar flow at Re = 500.

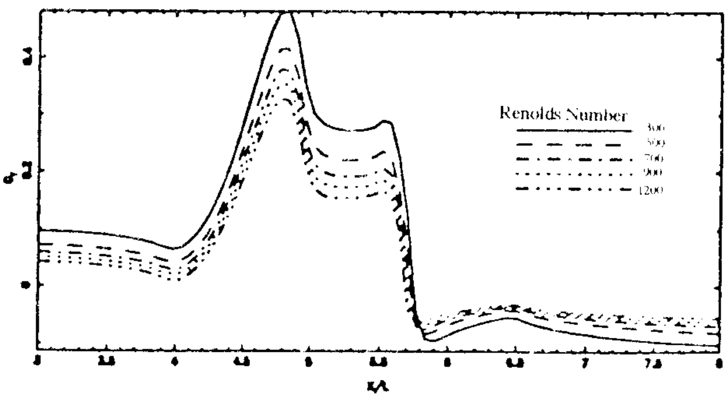

Figure 4. Effect of Reynolds number on wall friction.

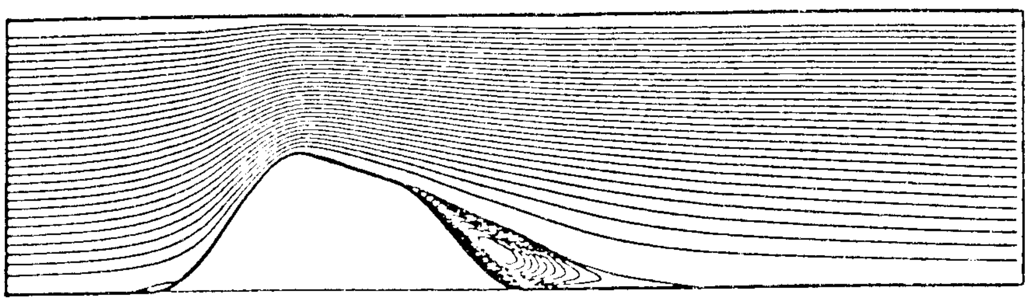

Figure 5. Turbulent flow pattern in a divergent constriction at Re = 5000

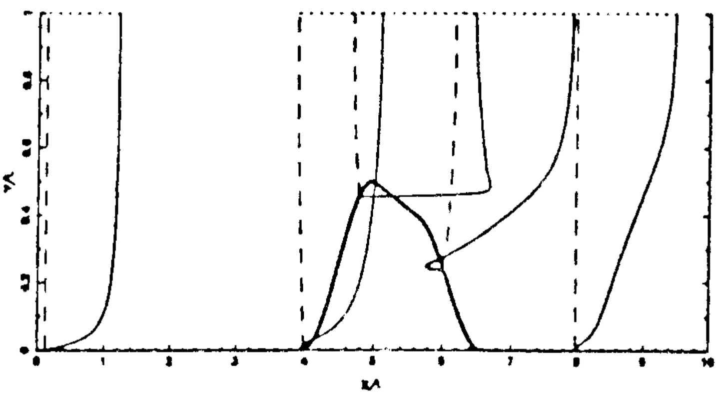

Figure 6. Velocity profiles in a divergent constriction.

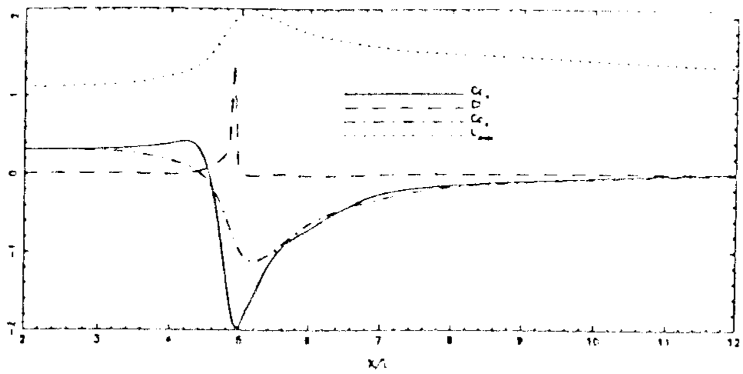

Figure 7. Pressure and friction coefficients in divergent constriction.

and very steep in the constriction's minimum area. Similar pattern and velocity profiles obtained for other configurations at Reynolds numbers ranged 3000 to 9000. Similarity exist between turbulent and laminar results on pressure drops and friction coefficient variations along the constriction and across the flow.

Figure 7 shows the wall and centerline pressure coefficients, also friction and centrerline velocity along the flow for a typical turbulent regime in a divergent constriction for a Reynolds number of 5000. The two pressures exhibit a large difference in the constriction where area variation forces the two dimensional flow. The centerline velocity reaches a maximum in the constriction where the centerline pressure reaches its minumum. The friction coefficient increases sharply in the curved section before the minumum cross-sectional area.

CONCLUSION

This paper reports an attempt at simulating the flow in the larynx using a numerical method for the solution of the Navier-Stokes equations. As the method itself has been tested and employed for other applications, the emphasis here has been on the flow patterns that arise in typical laryngeal passages. The simulation is limited to steady, two-dimensional flow. Results were obtained for various glottal configurations for both laminar and turbulent regimes. These clearly indicated the flow separation and formation of a jet downstream of the glottis. These solutions should enable a study of the influence of the strong shear layer associated with the separation, and the eddies downstream of the glottis, on the acoustic pressure in the vocal tracts. The calculated pressure showed not only severe changes along the glottis, but also some variations across the glottis. This demonstrates the inaccuracy of previous one-dimensional models for the prediction of pressure and flow in the glottis.

The present study represents only a first, and rather simple step in the numerical simulation of the actual flow in the larynx. There are, to be sure, a number of very important flow features that are ignored in the present analysis. Among these are, the unsteadiness of the flow, induced by the separation downstream of the constriction and by the deformation of the walls themselves, the three-dimensional geometry of the prototype. Furthermore, the turbulent flow in the glottal constriction is usually of low Reynolds number type and the k-ε turbulence model which is used in this analysis may need to be refined with experimental data.

ACKNOWLEDGMENTS

The study was supported by the National Institute on Deafness and Other Communication Disorders, Grant No. DC00831-01A2.

REFERENCES

1. Jw. van den Berg, J. T. Zantema and P. Jr. Doornenbal , *J. Acoust. Soc. Am.*, **29** (5), pp. 626-631 (1957).
2. K. Ishizaka and M. Matsudaira: "Fluid mechanical considerations of vocal cord vibration". SCRL-Monograph **8**, Speech Communication Research laboratory, Santa Barbara (1972).

3. R. C. Scherer: "Laryngeal Fluid Mechanics: Steady Flow Considerations Using Static Models". Ph. D. thesis, University of Iowa, Iowa City (1981).

4. R. C. Scherer, I. R. Titze and J. F. Curtis, *J. Acoust. Soc. Am.*, **73** (2), pp. 668-676 (1983).

5. R. C. Scherer and I. R. Titze, Pressure-flow relationships in a model of the laryngeal airway with a diverging glottis. In *"Vocal Fold Physiology: Contemporary research and Clinical Issues,"* pp. 179-193, edited by D. M. Bless and J. H. Abbs, College-Hill Press, San Diego (1983).

6. N. Binh, and J. Gauffin, Aerodynamic measurements in an enlarged static laryngeal model. *Technical Phoniatrics, STL-QPSR*, Department of Gas Dynamics, Royal institute of Technology, Stockholm (1983).

7. H. Lijima, N. Miki and N. Nagai, "Viscous flow analyses of the glottal model using a finite element method". Presented at the Second Joint Meeting of ASA and ASJ. Honolulu, Hawaii (1988).

8. J. Liljencrants, "Numerical simulations of glottal flow". Presented in Vocal Fold Physiology Conference. Stockholm, Sweden (1989).

9. V. C. Patel, J. T. Chon and J.Y. Yoon, "Laminar Flow Over Wavy Walls," submitted to *ASME, J. Fluids Engineering* (1991).

10. C. J. Chen, "Finite Analytic Method, Handbook of Numerical Heat Transfer" Edited by W. J. Minkowycz, E. M. Sparrow, R.H. Pletcher and G. E. Schneider (pp. 723-746), John Wiley & Sons (1988).

11. S. V. Patankar, "Numerical Heat Transfer and Fluid Flow," McGraw-Hill, New York, N. Y. (1980).