Abstract

Human phonation is a complex physiological process involving flow-induced oscillations of the vocal folds and aeroacoustic sound generation. The flow fields encountered in phonation are highly unsteady, feature massive flow separation and recirculation in the supraglottal spaces and generation of coherent vortex structures from the shear layer of the jet. For the sake of computational aeroacoustic modeling of human voice generation, an accurate resolution of the airflow through the vocal folds is essential. The Reynolds-averaged Navier-Stokes turbulence modeling is inappropriate, since it provides only the averaged flow field. The paper presents the first results obtained with a large-eddy simulation of flow through a model of human vocal folds using a second-order finite volume discretization of incompressible Navier-Stokes equations. In the first step, the flow field was resolved on a fine 2D mesh covering a short subglottal region, the glottis and a part of the supraglottal channel. The simulation was parallelized using domain decomposition method and run in parallel on a shared-memory supercomputer. The results compare two large eddy simulations using the algebraic Smagorinsky and one-equation sub-grid scale models against a simulation without turbulence model.

Keywords: human phonation, numerical simulation, large eddy simulations

1 Introduction

Human voice is created by passage of the airflow between vocal folds, which are located in the upper part of larynx (see Fig. 1, left). The vocal folds (also called vocal cords) are two symmetric soft tissue structures fixed between the thyroid cartilage and arytenoid cartilages; basically they are composed of the thyroarytenoid muscle (TA) and ligament covered by mucosa (see Fig. 1, right). More detailed information on vocal fold physiology can be found in the classical publication [1] or in the monographs [2, 3].

When air is expired from the lungs, the constriction formed by the vocal folds (which is called the glottis) induces acceleration of the flow and creates underpressure. Under certain circumstances (subglottal pressure, glottal width, longitudinal tension in the TA and ligament), vocal fold oscillations may occur due to fluid-structure interaction. Creation of voice by vocal fold vibration is usually referred to as phonation. In regular loud phonation, the vocal folds collide and close the laryngeal channel completely, and the duration of the glottal closure may span a considerable part of the vibration period. When whispering or in breathy phonation, the vocal folds may vibrate without collisions.

The frequency of vibration is influenced by many factors, primarily by the longitudinal tension in the TA muscle and in the ligament. The periodical glottal closure modulates the airflow and generates a sound with the fundamental frequency denoted usually F0. The spectrum of the acoustic signal also contains harmonic frequencies \( f_k = k \cdot F0 \). Due to the turbulence, generated mainly in the shear layer of the jet, natural healthy voice always contains broadband noise of certain level. The source sound produced by the vocal folds with vocal tract detached, which can be observed for example in experiments on excised larynges, does not resemble human voice. Human voice results from the acoustic filtering of the source signal by the vocal tract. Based on the actual geometry of the vocal tract, which changes mainly by the posture of the tongue, certain frequencies in the spectrum are amplified and other suppressed: in this way, different vowels are generated from the same source signal.

In the available literature, numerous mathematical models of phonation can be found. An extensive overview is given in a review paper [5]. Basically, the models can be divided into two
large families. First, the low-order models, where the complex interaction between the airflow and
the nonlinear tissue structures is modeled by a small number of discrete masses supported by linear
springs and dampers, coupled to simplified and often linearized flow models. Although this may
seem a serious oversimplification of the real physiological situation, such models proved very useful
especially for modeling of phonation onset mechanisms. The main advantage of these models is that
the equations may be solved either analytically, or using simple and very fast numerical methods
for ordinary differential equations, making it possible to perform nearly real-time simulations on
current computers.

However, since the low-order models resolve accurately neither the glottal airflow nor the visco-
elastic deformation of the vocal folds, increasing effort has been devoted to numerical solution of
the 2D or 3D Navier-Stokes equations on computational domains approximating the glottal chan-
nel, and to proper modeling of the structural mechanics. These high-order models are described by
partial differential equations. The equations contain important nonlinearities, coming from three
major sources: nonlinearity due to the convective term in the Navier-Stokes equations, geometric
nonlinearity in the elastomechanics due to large deformations, and material nonlinearities of the
living tissues. In the case of loud phonation, yet another nonlinearity appears due to vocal fold
collisions. The nonlinear partial differential equations have to be solved numerically, leading to
systems with large number of unknowns.

Even the high-order models usually do not try to model the phonation in all its complexity, and
contain numerous simplifications and reductions. With respect to the fluid-structure interaction,
the problem is sometimes simplified to static vocal fold case or forced vocal fold oscillation. The
latter approach was adopted within this paper, too.

The relation between the flow field and the aerodynamically generated sound is complex, and
still subject to ongoing research. A hybrid aeroacoustic approach to the computational modeling of
human voice generation was published by the author recently in [6]. In this work, the aeroacoustic
sound sources are computed using the Lighthill’s analogy or a perturbation approach, based on a
flow field calculated separately by a dedicated flow solver. The problem is that the glottal airflow,
with Reynolds numbers ranging typically between 1000 – 10000, lies in the transitional regime. In
the subglottal ducts (trachea and bronchial bifurcations, the flow is most likely laminar. Above
glottis, where a pulsating air jet enters the supraglottal volume, considerable turbulence is gener-
ated in the shear layer of the jet. For the sake of aeroacoustic modeling, using Reynolds-averaged
turbulence models providing an ensemble-averaged flow field is inappropriate. This is why the flow

Figure 1: Scheme of the vocal tract in sagittal section (left) – the vocal folds are located in the
region of the cricoid and thyroid cartilages. Detailed view of the larynx in coronal section (right)
[4].
model in [6] included no turbulence model. However, the effect of the sub-grid scale turbulence on the flow field may be important. The direct numerical simulation being unfeasible, the most reasonable approach for aeroacoustic modeling of human voice generation seems the Large Eddy Simulation (LES). The first simulations of the glottal flow field obtained using LES are presented in this paper, and compared to the previous numerical simulations without turbulence model. The CFD simulations were implemented using the open-source software package OpenFOAM version 2.3.0 and run in parallel on a shared-memory supercomputer SGI Altix UV-100.

2 Mathematical model

The Large Eddy Simulation approach can be regarded as a compromise between Direct Numerical Simulations (DNS) and Reynolds-Averaged Navier Stokes equations (RANS). It relies on the fact that most energy is carried by the large flow scales, and that the large scales are most influenced by the geometry and boundary conditions. Thus, they are most anisotropic, case-specific and most difficult to capture by a universal turbulence model. On the contrary, the small scales of turbulence are believed to be isotropic and universal, at least far enough from the channel walls. Unlike RANS, the large scales are fully resolved in LES, thus providing much more detail in the flow field. Only the small scales are averaged and their effect on the flow field modeled.

In the LES concept, any flow variable \( f \) may be decomposed into a into a large scale \( \bar{f} \) and small scale \( f' \) contribution:

\[
f = \bar{f} + f'
\]  

The large scales of any flow variable are resolved, the small scales (also called sub-grid scales, SGS) need to be modeled. If we denote by \( \Delta \) the characteristic length scale separating the large and sub-grid scales, the large scales can be extracted from the actual flow variable by a filtering operation

\[
\bar{f}(x,t) = \int G_{\Delta}(x-y) f(y,t) \, dy ,
\]

where \( G_{\Delta}(x-y) \) is the filter kernel, which is a function with compact support satisfying

\[
\int G_{\Delta}(x-y) \, dy = 1
\]

and where \( \Delta \) acts as the filter width. The current simulation uses a top-hat filter, which is (in 1D) defined as

\[
G_{\Delta}(x) = \begin{cases} \frac{1}{\Delta} & |x| \leq \frac{\Delta}{2} \\ 0 & |x| > \frac{\Delta}{2} \end{cases}
\]

This is a common choice for finite volume methods where the flow variable is assumed to vary linearly over the finite volume. The top-hat filter being a simple average, the averaged (cell-centered) value and the local value of \( \bar{f} \) are then equal for a suitably chosen filter width.

2.1 Incompressible Navier-Stokes equations in the LES concept

The Mach numbers in typical phonation being well below 0.2, the airflow is can be regarded as incompressible. Applying the filtering (2) to the incompressible Navier-Stokes equations

\[
\frac{\partial \mathbf{u}}{\partial t} + \nabla \cdot (\mathbf{u} \otimes \mathbf{u}) = -\frac{1}{\rho} \nabla p + \nabla \cdot \nu (\nabla \mathbf{u} + \nabla \mathbf{u}^T) \\
\nabla \cdot \mathbf{u} = 0 ,
\]

where \( \mathbf{u} \) is the flow velocity, \( p \) the fluid dynamic pressure, \( \rho \) density and \( \nu \) the kinematic viscosity, yields [7]
\[
\frac{\partial \mathbf{u}}{\partial t} + \nabla \cdot (\mathbf{u} \otimes \mathbf{u}) = -\frac{1}{\rho} \nabla \bar{p} + \nabla \cdot (2 \nu \mathbf{S})
\]

\[
\nabla \cdot \mathbf{u} = 0 ,
\]

where

\[
\mathbf{S} = \frac{1}{2} (\nabla \mathbf{u} + \nabla \mathbf{u}^T)
\]

is the large-scale strain rate tensor. Equations (6) are formally almost equivalent with (5) and represent the Navier-Stokes equations for the large-scale velocity field \( \mathbf{u} \), except for the convective term. Since \( \mathbf{u} \otimes \mathbf{u} \neq \mathbf{u} \otimes \mathbf{u} \), the difference has to be modeled, similarly as in the case of RANS closures. In LES, the term

\[
T = \mathbf{u} \otimes \mathbf{u} - \mathbf{u} \otimes \mathbf{u} = (\mathbf{u} + \mathbf{u}') \otimes (\mathbf{u} + \mathbf{u}') - \mathbf{u} \otimes \mathbf{u}
\]

is called the SGS stress tensor, and is responsible for the momentum exchange between the subgrid and filtered scales. For the filter width approaching zero, the SGS stresses vanish and in the limit a DNS solution is obtained:

\[
\lim_{\Delta \to 0} |T| = 0
\]

Compared to the Reynolds stresses in RANS, the SGS stresses carry much less of the turbulent energy, and so the accuracy of the model may be less crucial. It can be anticipated that even very simple SGS models can provide satisfactory results. Combining (6) and (8) yields the final governing equations for the filtered flow field

\[
\frac{\partial \mathbf{u}}{\partial t} + \nabla \cdot (\mathbf{u} \otimes \mathbf{u}) = -\frac{1}{\rho} \nabla \bar{p} + \nabla \cdot (2 \nu \mathbf{S} - T)
\]

\[
\nabla \cdot \mathbf{u} = 0 ,
\]

with the SGS stress \( T \) left to be modeled to close the set.

### 2.2 Smagorinsky SGS model

One of the first and simplest SGS stress models is the Smagorinsky algebraic model. The primary aim of the SGS model being to simulate the unresolved dissipative scales of the turbulent energy cascade and extract energy from the larger scales, an eddy-viscosity model can be employed. This approach is based on the assumption that the dissipation of the kinetic energy is analogous to molecular diffusion. The deviatoric part of the SGS stress tensor is modeled as

\[
T - \frac{1}{3} \text{tr}(T) I = T - \frac{2}{3} k I = -2 \nu_{SGS} \mathbf{S}
\]

where \( \nu_{SGS} \) is the sub-grid scale eddy viscosity and \( k \) the SGS kinetic energy. The Smagorinsky model assumes that the small scales dissipate instantaneously all energy transferred from the resolved scales. From this, Smagorinsky [8] derived that the SGS viscosity may be estimated as

\[
\nu_{SGS} = (C_S \Delta)^2 \sqrt{\mathbf{S} : \mathbf{S}}
\]

with the Smagorinsky constant \( C_S = 0.18 \ldots 0.23 \). In OpenFOAM, the SGS viscosity is computed as

\[
\nu_{SGS} = C_k \sqrt{k} \Delta
\]

with the SGS kinetic energy solved from the equation

\[
\mathbf{S} : \left( \frac{2}{3} k I - 2 \nu_{SGS} \text{dev}(\mathbf{S}) \right) + C_E \frac{k^{3/2}}{\Delta} = 0 .
\]
Equation (14) is based on the assumption of local equilibrium, the first term following from (11) (for incompressible flow, \( \text{dev}(\mathbf{S}) = \mathbf{S} \)) and the second one representing the SGS dissipation \( \epsilon \). Combining (13) and (14) yields a quadratic equation

\[
\mathbf{S} : \left( \frac{2}{3} \sqrt{k} \mathbf{I} - 2 C_k \Delta \text{dev}(\mathbf{S}) \right) + C_E \frac{k}{\Delta} = 0
\]  

from which the SGS kinetic energy is evaluated as

\[
k = \sqrt{-b \pm \sqrt{b^2 + 4ac}} / 2a ,
\]

where

\[
a = \frac{C_E}{\Delta} \\
b = \frac{2}{3} \text{tr}(\mathbf{S}) \\
c = 2 C_k \Delta (\text{dev}(\mathbf{S}) : \mathbf{S}) .
\]

The constants are set to \( C_k = 0.094 \) and \( C_E = 1.048 \).

### 2.3 One-equation SGS model

The main deficiency of the Smagorinsky algebraic model arises from the assumption of local equilibrium. In many real cases, notably free shear layer flows, separating and reattaching flows and wall-dominated flows, this assumption is not fulfilled [9]. This is precisely the case of airflow past the vocal folds, where a planar jet separates in the divergent part of glottis. The problem may be addressed by solving an additional transport equation for certain SGS turbulent quantity, e.g. the SGS kinetic energy \( k \), and thus taking into account the history effects. The implementation in OpenFOAM is based on a statistically-derived SGS kinetic energy model by Yoshizawa [10], which gives the transport equation for the SGS kinetic energy

\[
\frac{\partial k}{\partial t} + \nabla \cdot (k \mathbf{u}) - \nabla \cdot ((\nu + \nu_{\text{SGS}}) \nabla k) = -\mathbf{T} : \mathbf{S} - C_E \frac{k^{3/2}}{\Delta} .
\]  

The model again relies on the SGS eddy viscosity concept with the SGS viscosity

\[
\nu_{\text{SGS}} = C_k \sqrt{k} \Delta .
\]

The terms in (18) represent (from left to right) the changes of the SGS kinetic energy in time, convection by the large-scale velocity field and diffusion by the effective viscosity. The first source term on the right side models the decay of turbulence from the resolved scales to the SGS scales via the energy cascade and is approximated by

\[
-\mathbf{T} : \mathbf{S} = 2 \nu_{\text{SGS}} \sqrt{\mathbf{S} : \mathbf{S}} ,
\]

the second one is the SGS dissipation. The constants \( C_k \) and \( C_E \) are set identically as in the Smagorinsky algebraic model.

### 3 Geometry, mesh and numerical solution

For the purpose of the LES simulation testing and validation, a simplified 2D geometry was used to spare computational resources. The geometry of the model vocal folds used in this study was specified according to the parametric vocal fold shape description “M5” proposed by [11]. The “M5” model is piecewise linear with rounded corners; as compared to real vocal fold geometry, the model is considerably simplified [12]. However, it is widely used since it provides an easily parameterizable 2D approximation of the vocal fold surface geometry. In the current model, the
FVF s were approximated by a shape described in [13]. More details on the geometry can be found in [6].

The computational domain Ω consists of a short subglottal channel with inlet $G_{in}$, constriction formed by the oscillating bottom and upper vocal fold $G_{bVF}$ and $G_{uVF}$, immobile ventricular folds and a straight supraglottal channel bounded by fixed walls $G_{wall}$ and the outlet boundary $G_{out}$. Fig. 2 also shows the computational mesh in the initial (undeformed) configuration. The mesh is isotropic triangular and consists of about 60 k elements.

\[ \text{Fig. 2: Geometry, mesh, boundary parts and prescribed displacements } w_1, w_2 \]

Instead of a fixed flow velocity at inlet, which is not physiologically correct when the vocal folds oscillate, the flow is driven by a constant pressure difference between inlet and outlet. The boundary conditions are summarized in Tab. 1.

<table>
<thead>
<tr>
<th>$G_{in}$</th>
<th>$G_{out}$</th>
<th>$G_{bVF}$</th>
<th>$G_{uVF}$</th>
<th>$G_{wall}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$u$ [m/s]</td>
<td>$P$ [m$^2$/s$^2$]</td>
<td>$d$ [m]</td>
<td>$k$ [m$^2$/s$^2$] (s198 only)</td>
<td></td>
</tr>
<tr>
<td>eval. from flux</td>
<td>$P + \frac{1}{2}</td>
<td>u</td>
<td>^2 = 300$</td>
<td>$d = 0$</td>
</tr>
<tr>
<td>$u = 0$</td>
<td>$\frac{\partial P}{\partial n} = 0$</td>
<td>$d = 0$</td>
<td>$k = 0$</td>
<td></td>
</tr>
<tr>
<td>$(u \cdot n &lt; 0)$</td>
<td>$(u \cdot n &gt; 0)$</td>
<td>$(u \cdot n &gt; 0)$</td>
<td>$(u \cdot n &lt; 0)$</td>
<td></td>
</tr>
<tr>
<td>$u = 0$</td>
<td>$\frac{\partial u}{\partial t} = 0$</td>
<td>$d = h(x, y)$</td>
<td>$k = 0$</td>
<td></td>
</tr>
<tr>
<td>$G_{bVF}$</td>
<td>$G_{uVF}$</td>
<td>$G_{wall}$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>$u = \frac{\partial d}{\partial t}$</td>
<td>$u = \frac{\partial d}{\partial t}$</td>
<td>$u = 0$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>$\frac{\partial P}{\partial n} = 0$</td>
<td>$\frac{\partial P}{\partial n} = 0$</td>
<td>$\frac{\partial P}{\partial n} = 0$</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

During the CFD simulation, the vocal folds move, thus deforming the computational domain. The kinematics of the vocal folds were programmed to allow for two-degrees-of-freedom, convergent-divergent motion of each of the vocal folds, with prescribed sinusoidal displacement of the inferior and superior vocal fold margins in the medial-lateral direction (see Fig. 2)

\[
\begin{align*}
    w_1(t) &= A_1 \sin(2\pi ft + \xi) \\
    w_2(t) &= A_2 \sin(2\pi ft),
\end{align*}
\]

(21)

where $f = 100$ Hz is the frequency of vibration, $A_1 = A_2 = 0.3$ mm are the amplitudes and $\xi = \pi/2$ the phase difference. The bottom and upper vocal folds vibrate symmetrically. The coordinates (21) determine uniquely the glottal half-gap $g$ and the medial surface convergence angle. The function $h(x, y)$ is calculated from $w_1$ and $w_2$ in such a way that during oscillation, the inferior, medial and superior vocal fold surfaces remain straight, and the connecting radii correspond exactly to the “M5” shape definition from [11]. In this way, the shape is smooth
without sharp edges and the basic convergent-divergent motion of the real vocal folds is mimicked, while keeping the kinematic model simple.

The numerical solution was implemented in OpenFOAM v.2.3.0, an object-oriented open-source set of libraries programmed in the C++ language. OpenFOAM is based on the finite volume method in collocated cell-centered approach on arbitrarily unstructured meshes. The incompressible Navier-Stokes equations on a moving computational mesh were solved numerically using a modified PISO algorithm [14]. In contrast to the standard PISO algorithm, it has a substep iteration loop: multiple cycles over the same timestep with the last iteration results used as an initial guess for the next substep iteration. The current simulation used two substep iterations.

In general, at least second-order discretization schemes are appropriate for LES. The discretization schemes were as follows: second-order backward implicit Euler for the time derivative, total variation diminishing (TVD) scheme with a limiter function

\[ \Phi(r) = \max(0, \min(2r, 1)) \]  

for the convective terms, and central differencing scheme (CDS) with nonorthogonal correction for the diffusion term. The timestep \( \Delta t \) is adjusted automatically during the transient solution so that the maximum local Courant number is kept below a predefined limit. The Courant number on unstructured 3D meshes is calculated as

\[ Co = \frac{1}{2} \sum_f |\Phi_f| \Delta V \Delta t, \]  

where \( \Phi_f = A_f (u_f \cdot n_f) \) is the velocity flux normal to face \( f \) with surface \( A_f \) of the cell with volume \( \Delta V \).

The computational domain changes in time due to vocal fold oscillations. Since the vocal folds do not collide and close the channel completely in current simulations, it is not necessary to introduce topological changes to the mesh, instead it is simply deformed. The coordinates of the element vertices in a new timestep are found by solving an auxiliary Laplace equation

\[ \nabla \cdot (\gamma \nabla d) = 0 \]  

for the mesh displacement \( d \) with a (constant) diffusivity \( \gamma \). Due to significant element distortion and consequent loss of element orthogonality between the moving vocal folds, one outer loop of nonorthogonal correctors was utilized within the modified PISO algorithm to guarantee stability of the computation. The nonorthogonal correction strategy within the current finite volume discretization is described in detail by [15]. Unlike the discretization of the temporal, convective and diffusive terms, the nonorthogonal correctors are treated explicitly.

The resulting linear system for momentum and SGS kinetic energy were solved using a Gauss-Seidel smoother. For the pressure predictor and corrector steps, faster convergence was obtained using a geometric multigrid solver, using an OpenFOAM-specific cell agglomeration algorithm and a conjugate gradient-type method for solution of the coarsest level matrix. The Laplace equation (24) for the mesh motion was discretized using central differences and solved using the the multigrid solver.

For parallelization of the CFD simulations, OpenFOAM employs the domain decomposition method. For the current 2D test simulations, it was sufficient to decompose the domain into four subdomains and solve the equations on four cores of the SGI Altix UV 100 shared-memory parallel supercomputer.

### 4 Results

Fig. 3 compares the velocity fields at the end of the second vibration period calculated in three numerical simulations: s197 (no turbulence model), s198 (LES simulation with one-equation SGS model described in sec. 2.3) and s199 (LES simulation with Smagorinsky SGS model described in sec. 2.2). The velocity fields show an air jet formed by the glottal constriction, which separates close to the superior vocal fold margin and enters the supraglottal volume. Large coherent vortex structures in the supraglottal region are convected slowly towards the outlet of the domain, interacting with the pulsating jet and slowly disintegrating. The fact that the large vortices persist for
a long time is caused by the (unphysical) 2D setup, which largely influences the vortex dynamics. In 3D simulations, the coherent vortex structures disappear much earlier.

Figure 3: Velocity field at the end of the second period of vocal fold vibration for the three cases s197, s198 and s199

The major difference between cases s198-s199 and s197 is the velocity magnitude in the glottal constriction. In both LES simulations, the maximum flow velocity in the narrowest point of glottis is significantly lower than in s197 – 47.2 % for the one-equation model and 46.8 % for the Smagorinsky model. This is caused by the high effective viscosity $\nu_{\text{eff}} = \nu + \nu_{\text{SGS}}$. Fig. 4 demonstrates that the SGS viscosity in the high shear regions, i.e. the glottal constriction, shear layer of the jet and vortex boundaries and places where the jet and vortices interact with channel walls, exceeds the molecular viscosity $\nu = 1.58 \times 10^{-5}$ m$^2$s$^{-1}$ by up to almost one order of magnitude.
Discussion and conclusions

The test 2D LES simulations show promising results. The SGS stresses in the LES simulations influence the resolved flow field in a reasonable way, while not destroying the flow details important e.g. for aeroacoustic computations. It is known [7, 9] that the classical Smagorinsky SGS model overpredicts the dissipation rate and thus the SGS viscosity in free shear layers and near the walls. One possibility is to use the van Driest damping function, which reduces the sub-grid viscosity artificially in the near-wall regions. The second option is to continue with the one-equation SGS model or to use the dynamic Smagorinsky model. Theoretically, the one-equation model introduces additional computational cost and disk storage requirements due to the solution of the additional transport equation for the SGS kinetic energy. However, in current tests the s198 simulation required, surprisingly, even slightly lower total CPU time than s199. This behavior can be caused by a lot of factors and cannot be generalized.

For further work, several improvements are necessary: First, turbulence is from principle a 3D phenomenon and for correct modeling of the real 3D flow field, 3D simulations are inevitable. This means dramatic increase of demands on computational resources. However, the parallelization of the numerical simulation is functional and proved reasonable parallel scaling up to about 50 CPU cores [6]. A first 3D LES simulation was already successfully run on a mesh consisting of 2.4 M elements, but is not reported within this paper.

Second, the computational mesh will still require certain refinement near the channel walls. For wall-resolved LES simulations, wall-normal coordinate of the first gridpoint should be $y^+ < 1$ [16]. In the current simulations, the first mesh point is at $y^+ = 2$ (with wall units based on the freestream velocity in the wide channel and boundary layer length 0.01 m). However, in the narrowest glottal section, the $y^+$ coordinate is significantly higher.

The third important point for correct Large Eddy Simulations is the treatment of the inlet boundary, especially initialization of turbulence. This is arguably one of the most complicated issues in LES. Multiple approaches, such as synthetic turbulence generation, precursor simulations or mapping of the resolved turbulence inside the computational domain back to the inlet exist, and can have significant effect on the results. However, all of these approaches have their drawbacks, should be used with caution and deep understanding of the underlying physics.

Figure 4: Sub-grid-scale viscosity predicted by two SGS models (one-equation eddy viscosity model, Smagorinsky model)
Acknowledgment

The research has been supported by the Czech Science Foundation, project P101/11/0207 Coupled problems of fluid and solid mechanics - nonlinear aerelasticity. The CFD simulations were run on the facilities of the Supercomputing center of the CTU in Prague.

References


