INSTITUTTET FOR BYGNINGSTEKNIK

DEPT. OF BUILDING TECHNOLOGY AND STRUCTURAL ENGINEERING

AALBORG UNIVERSITETSCENTER • AUC • AALBORG • DANMARK

Lars Davidson and Peter Nielsen

Calculation of the Two-Dimensional Airflow in Facial Regions and Nasal Cavity Using an Unstructured Finite Volume

Dec 1995 **ISSN 1395-7953 R9539**

Calculation of the Two-Dimensional Airflow in Facial Regions and Nasal Cavity Using an Unstructured Finite Volume Solver

Lars Davidson
Thermo and Fluid Dynamics
Chalmers University of Technology
S-412 96 Gothenburg, Sweden
E-mail: lada@tfd.chalmers.se

Peter V. Nielsen
Dep. of Building Technology and Structural Engineering
Aalborg University
Sohngaardsholmsvej 57
DK-9000 Aalborg, Denmark
E-mail: i6pvn@civil.auc.dk

Abstract

In this short report we demonstrate the feasibility of using Computational Fluid Dynamics (CFD) for studying the flow in facial regions and nasal cavity. A two-dimensional unstructured finite volume flow solver [1] is used. For modelling the turbulence we use a standard $k-\varepsilon$ model.

Acknowledgments
This work was carried out during the first author's stay at Dep. of Building Technology and Structural Engineering, Aalborg University in Autumn 1995.

1. The Numerical Method

1.1. The Finite Volume Code
The code is described in detail in Ref. [1] and the its characteristics are:

- two-dimensional;
- based on SIMPLEC;
- control volumes can have arbitrary number of edges, i.e the control volumes can be triangles, quadrilaterals etc;
- Rhie-Chow interpolation [2] for face velocities;
- $k-\varepsilon$ model.

1.2. Equations
The Navier-Stokes equations and the continuity equation are solved

$$\begin{aligned} \frac{\partial}{\partial x_j}\left(\varrho U_i U_j\right) &= -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left((\mu+\mu_t)\frac{\partial U_i}{\partial x_j}\right) \\ \frac{\partial}{\partial x_i}\left(\varrho U_i\right) &= 0 \end{aligned} \tag{1}$$

where the turbulent viscosity is obtained from

$$\mu_t = c_\mu \varrho \frac{k^2}{\varepsilon} \tag{2}$$

The transport equations for the turbulent kinetic energy k and its dissipation have the form

$$\begin{aligned}
\frac{\partial}{\partial x_j}\left(\varrho U_j k\right) &= \frac{\partial}{\partial x_j}\left(\frac{\mu_t}{\sigma_k}\frac{\partial k}{\partial x_j}\right) + P_k - \varrho\varepsilon \\
\frac{\partial}{\partial x_j}\left(\varrho U_j \varepsilon\right) &= \frac{\partial}{\partial x_j}\left(\frac{\mu_t}{\sigma_\varepsilon}\frac{\partial \varepsilon}{\partial x_j}\right) + \frac{\varepsilon}{k}\left(c_1 P_k - c_2 \varrho\varepsilon\right) \\
P_k &= \mu_t \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right)\frac{\partial U_i}{\partial x_j}
\end{aligned}$$

1.3. Boundary Conditions

The configuration is shown in Fig. 1. The left boundary is inlet where

$$U_{in} = 0.1,\ V_{in} = 0$$

$$k_{in} = \left(0.1 U_{in}\right)^2,\ \varepsilon_{in} = \frac{0.16 k^{1.5}}{0.1 H}$$

where H denotes the height of the domain.

The top and bottom boundaries are symmetry boundaries where

$$\frac{\partial U}{\partial y} = \frac{\partial k}{\partial y} = \frac{\partial \varepsilon}{\partial y} = 0,\ V = 0$$

The outlets are located at the upper right corner, and lower right corner of the domain, see Fig. 1. Here constant mass flux is prescribed, and

$$\frac{\partial U}{\partial n} = \frac{\partial V}{\partial n} = \frac{\partial k}{\partial n} = \frac{\partial \varepsilon}{\partial n} = 0$$

where n is the coordinate direction normal to the outlet.

The remaining boundaries are walls, which are identical to those used in Ref. [3]. For convenience the wall functions are summarized below.

I For $y^+ \geq 11.63$ where $\mu_t/\mu \gg 1$, $\tau \simeq \tau_w$

1. The wall shear stress τ_w is obtained by calculating the viscosity at the node adjacent to the wall from the log-law. The viscosity used in momentum equations is prescribed at the nodes adjacent to the wall (index P) as follows. The shear stress at the wall can be expressed as

$$\tau_w = \mu_t \frac{\partial U}{\partial \eta} \approx \mu_t \frac{U_{||,P}}{\eta}$$

where $U_{||,P}$ denotes the velocity parallel to the wall and η is the normal distance to the wall. Using the definition of the friction velocity u_*

$$\tau_w = \varrho u_*^2$$

we obtain

$$\mu_t \frac{U_{||,P}}{\eta} = \varrho u_*^2 \rightarrow \mu_t = \frac{u_*}{U_{||,P}} \varrho u_* \eta$$

Substituting $u_*/U_{||,P}$ with the log-law

$$\frac{U_{||,P}}{u_*} = \frac{1}{\kappa} ln(E\eta^+)$$

we finally can write

$$\mu_t = \frac{\varrho u_* \eta \kappa}{ln(E\eta^+)}$$

where $\eta^+ = u_*\eta/\nu$.

2. The turbulent kinetic energy is set as

$$k_P = C_\mu^{-0.5} u_*^2$$

3. The energy dissipation rate is set as

$$\varepsilon_P = \frac{u_*^3}{\kappa y}$$

4. The shear stress is obtained by

$$\tau_w = \varrho u_*^2$$

II For $y^+ \leq 11.63$ where $\mu_t/\mu \ll 1$, $\tau \simeq \tau_w$

1. calculate u_* as follow

$$\frac{U_{||,P}}{u_*} = \frac{u_* y}{\nu}$$

2. follow the procedure 2 - 4 as explained above where $E = 9$ and von Karman constant, $\kappa = 0.41$

2. Results

The grid is shown in Fig. 2. It was generated using the mesh generator in FEMLAB [4]. The contours of the domain is given as input and then the mesh is generated automatically.

The vector field is presented in Fig. 3. A small recirculation is seen in the lower part of the facial cavity near the face.

3. Conclusions

The present report demonstrates that CFD can be used for studying the flow in facial regions and nasal cavity. A two-dimensional simulation has been used, which is a dramatic simplification of the reality. In the near future this method will be extended to three dimensions. Thermal boundary layers around the body as well as the unsteady breathing function will be considered. This will make it possible to study physical processes such as particle transport, particle deposition, heat transfer etc.

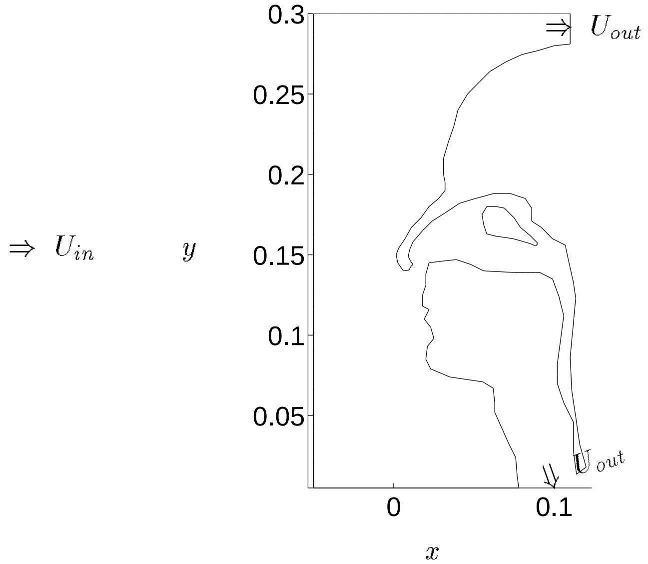

Figure 1. Configuration.

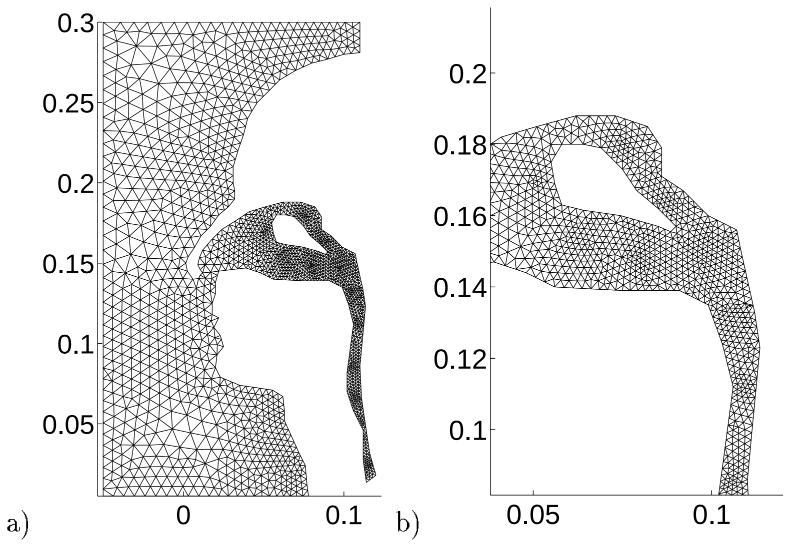

Figure 2. a) Global grid. b) Zoom of grid.

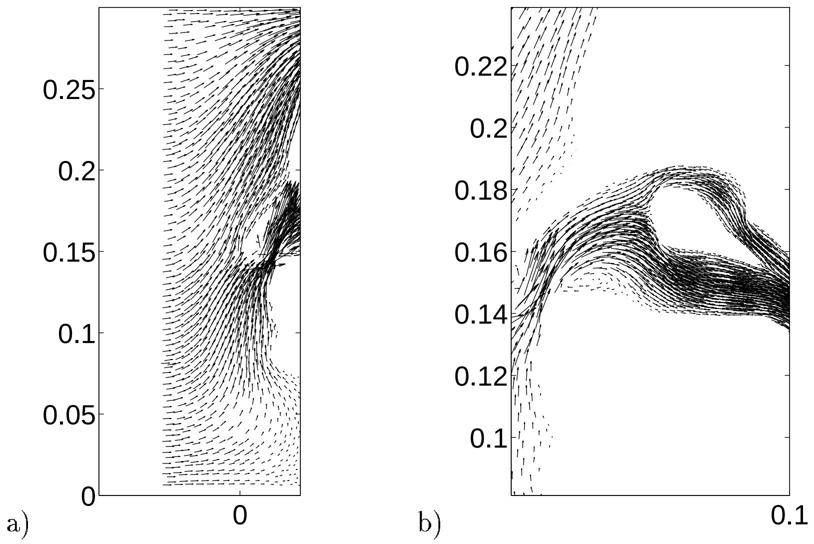

Figure 3. Vector plot. a) In front of the face. b) Inside the nasal cavity.

REFERENCES

1. DAVIDSON L. A pressure correction method for unstructured meshes with arbitrary control volumes. *Int. J. Numer. Meth. Fluids*, 22:1–17, 1996.
2. RHIE C.M. and CHOW W.L. Numerical study of the turbulent flow past an airfoil with trailing edge separation. *AIAA J.*, 21:1525–1532, 1983.
3. DAVIDSON L. and FARHANIEH B. CALC-BFC: A finite-volume code employing collocated variable arrangement and cartesian velocity components for computation of fluid flow and heat transfer in complex three-dimensional geometries. Rept. 92/4, Thermo and Fluid Dynamics, Chalmers University of Technology, Gothenburg, 1992.
4. FEMLAB. available on internet (`www.math.chalmers.se/research/femlab`), Dep. of Mathematics, Chalmers University of Technology, Göteborg.