J. ŠIMIČEK, O. HUBOVÁ

NUMERICAL ANALYSIS OF THE EFFECTS OF WIND ON BUILDING STRUCTURES

Jozef ŠIMIČEK

email: jozef.simicek@stuba.sk

Research field: Statics and Dynamics, Fluids mechanics

Olga HUBOVÁ

email: olga.hubova@stuba.sk

Research field: Statics and Dynamics, Aero-elasticity

Address:

Department of Structural Mechanics,
Faculty of Civil Engineering,
Slovak University of Technology,
Radlinského 11,
813 68 Bratislava

ABSTRACT

The paper deals with the theoretical foundations of laminar and turbulent fluid flows in a boundary layer and the effects of wind on constructions. The application section describes the flow of compressible Newtonian fluid as a system of partial differential equations (the Navier-Stokes equation) and solves these equations using FEM at each time step. Simplifying a 3D task to a plane problem and ignoring vorticity allows for the formulation of a wind flow on an open terrain and the effects of wind on a barrier. Two approaches to the numerical solution of wind flows are presented: one involves the Fluent commercial program to obtain the velocity field and pressure at a constant wind speed in an inlet region, and the other approach involves the development of the SIXIS specific software program for the purpose of this thesis, which modifies the wind velocity profile according to EN 1994-1-4. The results obtained - velocity fields around obstacles and the wind pressure coefficients on a barrier are processed and compared in a table.

1 INTRODUCTION

In the aerodynamics of structures it is necessary to determine the wind load, which is represented by a simplified set of pressures or forces for the whole structure or for the structural parts. Newly developed numerical Computation Fluid Dynamics (CFD) methods for the solution of flow problems help us to simulate and study characteristics of the wind flow, pressure and wind velocity distributions around an object. Due to the complex topography of surrounding buildings which modifies wind flow, it is necessary to use both experimental and numerical solutions for the definition of boundary conditions.

2 BASIC PHYSICAL PROPERTIES OF FLUIDS

In order to derive equations governing the motion of fluids, the following properties are taken into account:

- compressibility,
- viscosity,
- transmission properties.

The following concepts are introduced to describe fluid properties: the ideal, perfectly liquid, viscous, Newtonian and real fluid. An ideal fluid is understood as a fluid, which is perfectly incompressible with no viscous friction. A perfectly liquid fluid is understood as a fluid which can only be subjected to normal stresses. The flow of such fluids can be described by the so-called Euler equations. For viscous fluids both normal and shear stresses are applied.

Real fluids, however, are compressible and exhibit viscous friction. The flow of real fluids is described by the so-called Navier-Stokes equations. Unlike incompressible fluids, the density of flow in compressible fluids can vary from place to place. Given by a fluid's equation of state, density is correlated with the pressure and possibly with the temperature; therefore, any change in pressure

KEY WORDS

- Laminar and turbulent flow,
- Newtonian fluid,
- Finite element method,
- Numerical simulation of flows,
- Streamlines,
- Wind speed,
- Wind pressure coefficient.

corresponds to a change in density. The effect of compressibility can be determined and quantified by the coefficient of fluid compressibility:

$$\delta = \frac{1}{\rho} \cdot \frac{\partial \rho}{\partial p} = -\frac{1}{V} \cdot \frac{\partial V}{\partial p} \quad (1)$$

There is a distinction between an isothermal and an isentropic compression. If the derivative $\frac{\partial \rho}{\partial p}$ in the expression for the fluid compressibility factor under consideration assumes a constant temperature, then the coefficient δ describes so-called isothermal compressibility. In the case of $\frac{\partial \rho}{\partial p}$, where it assumes a constant entropy S_0 , then the coefficient $\frac{\partial \rho}{\partial p}$ describes so-called isentropic compressibility. The derivative, which determines the value of this coefficient, is a function of the speed of sound c_z [1].

$$c_z^2 = \left(\frac{\partial p}{\partial \rho} \right)_{S_0 = \text{const}} \quad (2)$$

Dynamic viscosity η generally depends on the pressure, temperature and physical properties of the Newtonian fluid and results from the linear relationship between the velocity gradient and shear stress.

$$\tau = \eta \cdot \frac{du}{dy} \cdot \vec{n}, \quad (3)$$

where $\vec{n}$ is the surface normal vector, i.e., a unit vector that is perpendicular to a surface.

The dynamic viscosity to the density ratio of a fluid is called kinematic viscosity $\nu = \frac{\eta}{\rho}$

The coefficient of the viscosity of liquids decreases with increasing temperatures, while that of gases increases. This different character of the dependence of dynamic viscosity on the temperature indicates differences in the mechanism of the internal friction of liquids and gases. The individual particles of a fluid in its chaotic movement

Fig. 1 Vectors of velocities and viscous stresses.

move from place to place; thus by changing their position; they change the momentum as well and may cause an exchange of thermal energy [3].

2.1 Classification of a flow

While a flow is being monitored, one can observe:

- the flow's dependence on time
- the nature of the kinematic motion of particles
- internal mixing motion of a real viscous fluid
- the ratio of the velocity to the speed of sound.

In terms of time, we classify fluid flows into a steady flow, known as “stationary flow”, and an unsteady flow, also known as “non-stationary flow”.

For a steady flow we assume that the state characteristics of pressure, density and temperature will not change over time. For non-stationary flow we assume the state characteristics will change over time.

In terms of the kinematic motion of particles, we classify flows into a “streamline flow”, when we assume that the fluid particles will only slide along, and a “vortex flow”, which has additional rotational movements of the particles.

Laminar flow (Fig. 2) occurs when a fluid flows in parallel layers with no disruption between the layers. At low velocities the fluid tends to flow without lateral mixing. **Turbulent flow** (Fig. 3), which occurs at higher velocities where eddies or small packets of fluid particles form, leads to lateral mixing of a random character. The mixing happens due to the transfer of momentum across the streamlines.

Fig. 2 Streamlines typical of laminar flow.

Fig. 3 Streamlines typical of turbulent flow.

3 FLOW EQUATIONS

The essence of fluid flow is a change in the position of fluid particles in space. The main task of kinematics is to describe the fluid motion in time and space and describe the velocity field and the acceleration field of particles at a specific point on the jet space examined. The instantaneous state of the flow on each point of the space is uniquely determined by the particle velocity vector at any given location in a given time. The valid conditions for the fluid flow can be formulated by:

- the balance of the mass,
- the balance of the momentum
- the balance of the energy.

A case of compressible Newtonian fluid flow is assumed. The equation of continuity for such a case is determined in the form of a vector:

$$\int_V \left(\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \cdot \vec{c}) \right) dV = 0. \quad (4)$$

From the balance of the momentum follows:

$$\int_V \rho \cdot \left(\frac{\partial \vec{c}}{\partial t} + (\vec{c} \cdot \nabla) \vec{c} + \nabla p - \vec{g} \right) dV = \int_V (\eta \cdot \Delta \vec{c} + \eta \cdot \delta_{ij} \cdot \nabla \cdot (\nabla \cdot \vec{c}) + \lambda \cdot \delta_{ij} \cdot \nabla \cdot (\nabla \cdot \vec{c})) \cdot dV \quad (5)$$

From the balance of the energy follows:

$$\frac{\partial}{\partial t} \int_V e \cdot dV + \int_V \nabla e \cdot \vec{c} \cdot dV + \int_V \nabla p \cdot \vec{c} \cdot dV - \int_V \rho \cdot \vec{g} \cdot \vec{c} \cdot dV + W_Q = \eta \cdot \int_V \left[\Delta \vec{c} + \left(1 + \frac{\lambda}{\eta} \right) \cdot (\delta_{ij} \cdot \nabla) \cdot (\nabla \cdot \vec{c}) \right] \cdot \vec{c} \cdot dV, \quad (6)$$

where:

$\rho = \rho(x, y, z)$ is the density of the mass,
 $\vec{c} = \dot{u}(x, y, z, t) \cdot \vec{i} + \dot{v}(x, y, z, t) \cdot \vec{j} + \dot{w}(x, y, z, t) \cdot \vec{k}$ is the velocity vector,
 $p = p(x, y, z, t)$ is the pressure field,
 $\vec{g} = g_x(x, y, z, t) \cdot \vec{i} + g_y(x, y, z, t) \cdot \vec{j} + g_z(x, y, z, t) \cdot \vec{k}$ is the force field,
 $\vec{q} = q_x(x, y, z, t) \cdot \vec{i} + q_y(x, y, z, t) \cdot \vec{j} + q_z(x, y, z, t) \cdot \vec{k}$ is the heat flow across a border inside a closed volume,
 W_Q is the power of thermal energy,
 E is the conserved energy,
 η is the dynamic viscosity of the fluid,
 λ is the second coefficient of the viscosity,
 $\delta_{ij} = 0$ for $i \neq j$, $\delta_{ij} = 1$ for $i = j$, $i, j = 1, 2, 3$.

3.1 Time approximation for a fluid flow model

For an ideal fluid flow we would consider at each time point t the validity of relations (4) - (6).

For a linear approximation of the acceleration we can write:

$$t = \langle t_b, t_{b+1} \rangle, \\ \vec{a}(t) = \vec{a}_{b-1} + \frac{\vec{a}_b - \vec{a}_{b-1}}{t_b - t_{b-1}} \cdot t = \vec{a}_{b-1} + \frac{\Delta \vec{a}_b}{\Delta t_b} \cdot t,$$

$$\vec{c}(t) = \int_0^t \left(\vec{a}_{b-1} + \frac{\Delta \vec{a}_b}{\Delta t_b} \cdot t \right) dt + \vec{c}_{b-1}$$

$$\vec{c}_b = \vec{a}_{b-1} \cdot \Delta t_b + \Delta \vec{a}_b \cdot \frac{\Delta t_b}{2} + \vec{c}_{b-1},$$

$$\Delta \vec{c}_b = \vec{a}_{b-1} \cdot \Delta t_b + \Delta \vec{a}_b \cdot \frac{\Delta t_b}{2} \quad \Delta \vec{a}_b = \frac{2}{\Delta t_b} \cdot \Delta \vec{c}_b - 2 \cdot \vec{a}_{b-1},$$

$$[K_c]_{t=b} = \frac{2}{\Delta t_b} \cdot [M] + [L(\{\vec{c}\}_{t=b})] - [K],$$

$$\{R\}_{t=b} = \{\Delta G\}_{t=b} - [B]^T \cdot \{\Delta p\}_{t=b} + 2 \cdot [M] \cdot \{\vec{a}\}_{t=b-1},$$

$$[K_c]_{t=b} \cdot \{\Delta \vec{c}\}_{t=b} = \{R\}_{t=b},$$

$$[B] \cdot \{\Delta \vec{c}\}_{t=b} = \{0\},$$

$$\vec{c}_b = \vec{c}_{b-1} + \Delta \vec{c}_b,$$

$$\Delta \vec{a}_b = \frac{2}{\Delta t_b} \cdot \Delta \vec{c}_b - 2 \cdot \vec{a}_{b-1},$$

$$\vec{a}_b = \vec{a}_{b-1} + \Delta \vec{a}_b. \quad (7)$$

3.2 Laminar and turbulent flow

For the planar along the streamlines of the moving laminar incompressible fluid flow, zero vorticity can be assumed.

$$\omega_z(x, y) = \frac{\partial \dot{v}(x, y)}{\partial x} - \frac{\partial \dot{u}(x, y)}{\partial y} = 0. \quad (8)$$

By introducing the scalar stream function $\Psi(x, y)$, which is based on the idea of streamlines, it is possible to express the wind velocity components:

$$\dot{u}(x, y) = \frac{\partial \Psi(x, y)}{\partial y}, \\ \dot{v}(x, y) = -\frac{\partial \Psi(x, y)}{\partial x}. \quad (9)$$

By substituting the velocity components expressed by the derivative of the stream function (9) in the condition of zero vorticity (8), we

can assume the equation for the laminar flow as follows:

$$\omega_z = \frac{\partial \Psi^2}{\partial x^2} + \frac{\partial \Psi^2}{\partial y^2} = 0. \quad (10)$$

In areas of the extreme gradient of the velocity vector behind a barrier, a backflow due to the vorticity of the wind and turbulent wind flow can be expected (Fig. 4).

Fig. 4 Wind velocity profile of laminar flow, boundary layer and backflow.

For planar stationary incompressible fluid flow equations of motion will take the form:

$$\begin{aligned} \dot{u} \cdot \frac{\partial \dot{u}}{\partial x} + \dot{v} \cdot \frac{\partial \dot{u}}{\partial y} &= -\frac{1}{\rho} \cdot \frac{\partial p}{\partial x} + \nu \cdot \left(\frac{\partial^2 \dot{u}}{\partial x^2} + \frac{\partial^2 \dot{u}}{\partial y^2} \right), \\ \dot{u} \cdot \frac{\partial \dot{v}}{\partial x} + \dot{v} \cdot \frac{\partial \dot{v}}{\partial y} &= -\frac{1}{\rho} \cdot \frac{\partial p}{\partial y} + \nu \cdot \left(\frac{\partial^2 \dot{v}}{\partial x^2} + \frac{\partial^2 \dot{v}}{\partial y^2} \right). \end{aligned} \quad (11)$$

Together with the equation of continuity,

$$\frac{\partial \dot{u}}{\partial x} + \frac{\partial \dot{v}}{\partial y} = 0 \quad (12)$$

we will obtain three differential equations for three unknown variables $\dot{u}$, $\dot{v}$, p .

4. EXPERIMENTAL MEASUREMENTS

To verify the accuracy of the numerical solutions, it is necessary to obtain the values of wind pressure and velocity by other methods. Conducting experimental measurements, either in wind tunnels or directly by measuring in situ using real structures, is the best method to verify them. Extensive experiments were carried out in situ at the Silsoe Research Institute of the University of Auckland [7]. Detailed measurements of the pressure on the surface of a cube at the speed of 5 frames per second were performed on a cube with dimensions of 6 x 6 x 6 m (Fig. 5).

Fig. 5 Cube at the Silsoe Research Institute.

The external pressure coefficient c_p is usually used for a quantitative evaluation of a pressure distribution in the building industry to determine the effect of wind on the external surfaces of a building. It generally depends on the shape of the structures, the wind velocity profile and the position of the structure. The local value of the pressure coefficient for a given velocity profile can be expressed by the relationship:

$$c_p = \frac{p(y)}{\frac{1}{2} \cdot \rho \cdot \dot{u}(y)^2} \quad (13)$$

where: $p(y)$ [Pa] ∞ is the experimentally measured or calculated (using FEM) local wind pressure value at the height y ∞

ρ [kg · m⁻³] is the air density,

$\dot{u}(y)$ is the horizontal wind speed component, corresponding to the measured value of the local pressure, which is not affected by the barrier.

It is possible to see the distribution of the wind pressure coefficients for various surfaces of the Silsoe cube in Figure 6.

The external wind pressure coefficient $c_{p,e}$, as expressed in the form of civil engineering codes, applies only to the velocity profile of a wind resulting from the agreed-upon measurements of its average speed.

Fig. 6 Distribution of the external pressure coefficient.

Fig. 7 Boundary conditions used in the FLUENT program.

5. NUMERICAL SIMULATION OF FLOWS

For the design of structures and the assessment of wind effects on a building structure, it is necessary to obtain information about the wind load and distribution of the load on specific areas of the surface or the whole structure. It is possible to find the wind velocity and pressure by means of numerical simulation. In this article numerical methods for a solution of averaged Navier-Stokes equations are discussed. The FLUENT and SIXIS computer programs based on FEM were used for the numerical simulation of two-dimensional fluid flows. The result of this computer-modeled example is a pressure distribution on saddle-shaped obstacles.

5.1 Calculation using the FLUENT program

The numerical solution to determine the velocity and pressure fields of a wind flow with a saddle-shaped barrier and with a constant wind speed in an inlet region was first calculated with the Fluent commercial program. A planar 2D model for the numerical simulation of turbulent flows with an air density of $\rho = 1,25 \text{ kg} \cdot \text{m}^{-3}$ and a dynamic viscosity of $\eta = 1,8 \cdot 10^{-3} \text{ kg} \cdot \text{m}^{-1} \cdot \text{s}^{-1}$ was applied. A turbulent flow near the walls was assumed, and boundary conditions of the following types were used: inlet region - constant wind velocity; outlet region - the velocity perpendicular to the streamline direction is zero; the wind speed on the terrain is zero (Fig. 7). For the upper border, at a height of 100m, the wind velocity is constant and zero. The wind velocity perpendicular to the surfaces of the barrier is zero. The results obtained from the FLUENT commercial program are illustrated in Figs. 8-10. It is possible to see the streamlines forming the boundary layers on Fig. 8 and 9. The distribution of the wind pressure around the obstacle is shown on Fig. 10. The distance from the barrier, where the velocity profile is not affected by it, is approximately equal to the height of the computer - modeled area.

Fig. 8 Wind field and streamlines from the FLUENT program.

Fig. 9 Wind velocity from the FLUENT program in [m/s].

5.2 Calculation using the SIXIS program and comparison

A Visual Basic program was developed for the numerical method to solve the problem of planar fluid flows. This program is based on relationships (8) - (10) of an ideal fluid without any heat exchange and deals with horizontal undisturbed wind flows in an open terrain (Terrain cat. II).

Fig. 10 Wind pressure distribution from the FLUENT program in [Pa].

Boundary condition:

Inlet region: wind velocity profile according to EN 1991-1-4 (see expression (12)) for the different terrain categories,

$$\dot{u}(y) = 0,19 \cdot \left(\frac{z_0}{0,05} \right)^{0,07} \cdot \ln \left(\frac{y}{z_0} \right) \cdot v_{b,0} \quad (14)$$

$$y \in \langle y_{\min}, y_{\max} \rangle$$

Where:

z_0 is the roughness length, for open terrain Terrain category II $z_0 = 0,05\text{m}$, Boundary interval $y_{\min} = 2\text{m}$, $y_{\max} = 100\text{m}$

For the interval $y \in \langle 0, y_{\min} \rangle$, the wind velocity profile is constant with the value $\dot{u}(y_{\min})$.

The wind profiles used, which are plotted in Fig. 11, assume different fundamental values of the basic wind speed: for the saddle-shaped obstacle, the fundamental values of the wind velocity are: $v_{b,0} = 24 \text{ m} \cdot \text{s}^{-1}$, $v_{b,0} = 26 \text{ m} \cdot \text{s}^{-1}$.

The wind velocity perpendicular to the surfaces of the barrier is zero.

Outlet region: parallel streamlines; the wind speed on the terrain is zero.

It is possible to see the streamlines and the calculation model's area with the obstacle in Fig. 12.

A comparison of the results – the external pressure coefficients for a windward area obtained by different methods - is given in Tab. 1.

Tab. 1 Comparison of the external wind pressure coefficients.

STN EN 1991-1-4		Measurements at Silsoe RI	SIXIS		FLUENT
$c_{pe,10}$	$c_{pe,1}$		$v_{\text{ref}} = 26 \text{ m/s}$	$v_{\text{ref}} = 24 \text{ m/s}$	$\dot{u}(y) = 25 \text{ m/s}$
0.8	1.0	c_{pe}	c_{pe}	c_{pe}	c_{pe}
		0,5 ÷ 0.8	0.64	0.76	0.91

Fig. 11 Wind profiles due to STN EN 1991-1-4 for different basic wind velocities.

Fig. 12 Streamlines from the SIXIS program.

6. CONCLUSION

The numerical methods presented for the solution to the flow problem and the numerical simulation of the fluid flows, together with the analysis and parametric studies, will allow engineers and designers to get a good picture of the distribution of the wind pressure around obstacles. The SIXIS program calculates the wind pressure coefficients on the windward side of structures for different wind speed profiles. This approach represents a better fitting description of a wind's natural behavior.

ACKNOWLEDGEMENTS

The presented results were achieved under the sponsorship of the VEGA Grant Agency of the Slovak Republic (Grant. Reg. No. 1/1119/11).

REFERENCES

- [1] Dvořák, R., Kozel, K.: Matematické modelování v aerodynamice (Mathematical modeling of aerodynamics), Prague, 1996.
- [2] Easom, G.J.: Improved turbulence models for computational wind engineering, PhD. Thesis in Civil Engineering, University of Nottingham, UK, 2000.
- [3] Horák, Z., Krupka, F.: Fyzika -Příručka pro vysoké školy (Physics - Handbook for universities), Prague, 1976.
- [4] Krempaský, J.: Fyzika - Příručka pre vysoké školy technické (Physics - Handbook for Technical Universities), Bratislava, 1982.
- [5] EN 1991-1-4: Eurocode 1: Action on structures, Parts 1- 4 Wind actions, 2005.
- [6] Hubová, O.: The Effect of the Wind on a Structure, Slovak Journal of Civil Engineering 2007/3, Vol. XV, ISSN 1210-3896, 2007.
- [7] Richards, P. J., Hoxey, R. P., Short, L. J.: Wind pressures on a 6 m cube, Journal of Wind Engineering and Industrial Aerodynamics, Vol. 89, Issues 14 – 15, 2001, pp.1553 – 1564, ELSEVIER.