Modeling pictures of the flow past rough and smooth surfaces in the program comsol multiphysics

Win Thu and D V Ilin

National research University Moscow power engineering Institute «MPEI», Krasnokazarmennaya street., 14, Moscow, 111250 Russia,

E-mail: winthu.thu51@gmail.com

Abstract. The aim of this work was to obtain simulated flow patterns of rough surfaces by aerodynamic flow at different speeds for comparison with the results obtained by anemometry from images of particles in a laboratory wind tunnel. The results obtained by the program Comsol Multiphysics, which is used to simulate complex high-turbulent flows characteristic of gas flows in aircraft engines and gas turbine plants.

1. Introduction

When describing the gas flow near a solid surface, the models of interaction of gas particles with the surface play a decisive role. Any real surface for a falling gas particle is rough. Therefore, one of the most important factors that need to be taken into account in the process of interaction is roughness [1, 2]. Roughness as a property of a real non-smooth surface is manifested through a set of individual irregularities that form the microstructure of the surface. It is associated with such surface properties as microhardness, friction, Aero- and hydrodynamic resistance, etc. the Problem of boundary conditions for the gas-dynamic task often depends on the ignorance of the real surface structure, which interact with the gas molecules. At the same time, experiments show that the surface roughness at the atomic level and above has a significant effect on the energy and momentum transfer during the flow of solids by gas flows.

To date, both analytical methods – series expansions, approximation theory, asymptotic expansions – and numerical methods (calculation of multiple integrals) are used to study the interaction of a gas with a rough surface.

In this work, a numerical simulation of the flow pattern of rough surfaces by an aerodynamic flow at different speeds is carried out for comparison with the results obtained by anemometry from images of particles (AIP). Anemometry by images of particles (AIP) enables contactless measuring of the velocity of the motion of gaseous, liquid and solid media containing light-scattering heterogeneities. This method is now widely used in scientific research and technical applications. Anemometry by images of particles (AIP) can solve a wide range of problems: from the study of slow directional movements in capillaries and
living cells to remote measurements of the turbulent velocity of gas flows in supersonic tubes and wind speed in the atmosphere. The measured velocities range from several μm/s to several km/s [3].

In the application of optical methods of flow diagnostics to specific tasks often raises the question of confirmation of the correctness of the results. The solution of the problem is obtained using the comsol Multiphysics software package, which uses the finite element method for the numerical solution of systems of partial differential equations [4].

The program contains various modeling and simulation tools, which include tools for geometric constructions, a finite element mesh generator, various solvers for linear and nonlinear problems, and post-processing tools.

2. The calculation of the modeling pictures of the flow around rough surfaces using the program comsol multiphysics

To solve the problem of air flow along the rough surface (Rz = 160 μm) in Comsol Multiphysics, a physical interface (module) of the turbulent flow was used, taking into account the properties of the viscous boundary layer near the solid wall. Of the many models of turbulent flow offered by Comsol Multiphysics, the v2-f model was chosen. This model is low-Reynolds and can be used to calculate the flow over the entire thickness of the boundary layer, including the viscous sublayer and buffer layer.

The equations solved by the Turbulent Flow interface, v2-f, are the Reynolds-averaged Navier-Stokes (RANS) equations for momentum conservation and the continuity equation for mass conservation [5-7].

The Navier-Stokes Equations – it is a system of differential equations that describes the motion of a viscous Newtonian fluid or gas (1 and 2):

\[
\begin{align*}
\frac{d\mathbf{u}}{dt} + \rho (\mathbf{u} \cdot \nabla) \mathbf{u} &= \nabla \left[-p \mathbf{I} + \mathbf{K}\right] + \mathbf{F}, \\
\rho \nabla \cdot (\mathbf{u}) &= 0,
\end{align*}
\]

where: \(\rho\) – fluid density, \(t\) – time, \(p\) – pressure, \(\mathbf{u}\) – fluid velocity, \(\nabla\) – Nabla operator, \(\mathbf{F}\) – vector field of mass forces, \(\mathbf{I}\) – intensity of turbulent flow and vector \(\mathbf{K}\) is calculated as the product of the sum of viscosities (dynamic and turbulent) and velocity operators:

\[
\mathbf{K} = (\mu + \mu_t)(\nabla \mathbf{u} + (\nabla \mathbf{u})^T),
\]

where \(\mu\) – dynamic viscosity, \(\mu_t\) – turbulent dynamic viscosity.

The first equation (1) in the system is the equation of motion itself. In the left part are the product of the density of the corresponding acceleration. In the right part – the product of density on the forces of pressure and internal friction. The second equation (2) is the continuity equation. Its physical meaning is the preservation of mass for the flow of liquid.

The expression for \(\mathbf{K}\) is nothing but a substantial derivative (also called a complete derivative). It shows how the acceleration of a material point that moves in a stationary gas medium changes. It shows the change in the properties of a point over time, as if it were stationary. A term \(\nabla \mathbf{u}\) is called a convective derivative describing the evolution of properties at a fixed point due to the fact that a gas medium flows through it at a speed.

It should be noted that the system of Navier-Stokes equations gives very accurate solutions if the laminar gas flow is considered, but in turbulent flow the equations are very sensitive to the values of the coefficients: a change in the Reynolds number by 0.05% can lead to a radically different result.

For simplicity, consider an example in which the gas flow rate is considered constant, etc. \(du/dt=0\). In this case, the system of Navier Stokes equations takes the form (3):
\[
\begin{align*}
\rho (u \cdot \nabla) u &= \nabla \left[ -\rho I + K \right] + F, \\
\rho \nabla \cdot (u) &= 0.
\end{align*}
\]

For the v2-f model, the turbulent viscosity is based on velocity fluctuations normal to the current lines. This makes it possible to model the anisotropy of turbulence and to separate the effects with a low Reynolds number and take into account the effect of the walls. The nonlocal influence of pressure fluctuations on turbulent fields is taken into account by applying elliptic relaxation to the pressure deformation term \[8\]. In COMSOL Multiphysics, this effect is formulated for normalized velocity oscillations \( \xi = \frac{u^2}{k} \) and the elliptic mixing function \( \alpha \), which is used to combine the effects associated with the walls in the far field. Thus, the complete model for compressible gas flow has for our case the form (4):

\[
\begin{align*}
\rho (u \cdot \nabla) k &= \nabla \left[ \left( \mu + \frac{\mu_T}{\sigma_k} \right) \nabla k \right] + P_k - \rho \varepsilon, \\
\rho (u \cdot \nabla) \varepsilon &= \nabla \left[ \left( \mu + \frac{\mu_T}{\sigma_k} \right) \nabla \varepsilon \right] + \frac{1}{\tau} \left( C_{T1} (\xi, \alpha) \right) P_k - C_{T2} (k, \varepsilon, \alpha) \rho \varepsilon, \\
\rho (u \cdot \nabla) \xi &= \nabla \left[ \left( \mu + \frac{\mu_T}{\sigma_k} \right) \nabla \xi \right] + \frac{2}{k} \left( \alpha^3 \mu + \frac{\mu_T}{\sigma_k} \right) \nabla k \cdot \nabla \xi + \left( 1 + \alpha^3 \right) f_w + \alpha^3 f_h - \frac{\xi}{k} P_k,
\end{align*}
\]

where \( k \) – turbulent kinetic energy, \( \sigma \) – stresses in fluid flow, \( \varepsilon \) – turbulent dispersion rate. Since the equation of turbulent kinetic energy \( k \) and turbulent scattering velocity \( \varepsilon \) is derived directly analytically, it uses a number of auxiliary coefficients that provide the required dimension of the input quantities, including these coefficients \( C_{T1} = 1.5, C_{T2} = 1.9, C_\mu = 0.09, \sigma_\varepsilon = 1.4 \). This model introduces two important concepts – generation \( P_k = \mu_T \left[ \nabla u \cdot \left( \nabla u + (\nabla u)^T \right) \right] \) and dissipation \( \varepsilon \). The physical meaning of generation of turbulence \( P_k \) is in the generation of new vortices and ripples that make up the turbulence. Dissipation \( \varepsilon \), on the contrary, is the dispersion of large vortices into smaller ones, which leads to averaging of the flow and reducing turbulence. The elliptic mixing function \( \alpha \) is related to the turbulence length \( L = C_L \max \left[ \frac{k^{3/2}}{\varepsilon}, C_p \left( \frac{v^3}{\varepsilon} \right)^{1/4} \right] \), according to the expression 5:

\[
\alpha - L^2 \nabla^2 \alpha = 1,
\]

where \( v = \mu / \rho \) - kinematic viscosity of the gas.

Turbulent dynamic viscosity \( \mu_T \) in equation (4) and parameters and parameters \( f_w \) and \( f_h \) in equation (4) are defined according to expressions (6-8):

\[
\mu_T = \rho C_p k \varepsilon \tau,
\]

\[
f_w = -\frac{\varepsilon}{\xi} \frac{k}{\xi},
\]
The stresses in the gas flow in the wall area \( \sigma_w \) and distance \( l_w \) are found from the solution of boundary conditions for the flow (9-10):

\[
\nabla G \cdot \nabla G + \sigma_w G(\nabla \cdot \nabla G) = (1 + 2\sigma_w)G^4,
\]

\[
l_w = \frac{1}{G} - \frac{l_{ref}}{2},
\]

where \( G \) – inverse distance to the wall, \( l_{ref} \) – the distance from the wall at which there is a linear change in the air velocity (viscous sublayer).

3. **Computational models**

In figure 1 shows a General view of the geometry of the computational domain. The air flow is moving from left to right. The flow rate varies in steps from 0.1 and 0.5 m/s to 5.5 m/s in increments of 0.5 m/s. the lower boundary of the computational domain is represented as a curved periodic structure, repeating the irregularities of the test sample of the grinding skin.
4. The results are simulated pictures of the flow around rough surfaces and smooth surfaces by an aerodynamic flow with different speeds.

As a result of the calculation, the distribution pattern of the air flow velocity along the curved surface is obtained. In figures 4-7 shows the final characteristic velocity field for different air flow rates at the input boundary of the model.

**Figure 3.** Finite element mesh in boundary layer.

![Finite element mesh in boundary layer](image)

**Figure 4.** (a) Flow velocity field at the input flow velocity of 1.0 m/s in rough surface. (b) The input flow velocity field with a flow velocity of 1.0 m/s brings the viewer closer to the rough surface for comparison with 1.0 m/s smooth surface.
Figure 5. (c) Flow velocity field at the input flow velocity of 5.0 m/s in rough surface. (d) The input flow velocity field with a flow velocity of 1.0 m/s brings the viewer closer to the rough surface for comparison with 5.0 m/s smooth surface.
Figure 6. (e) Flow velocity field at the input flow velocity of 1.0 m/s in smooth surface. (f) The input flow velocity field with a flow velocity of 1.0 m/s brings the viewer closer to the smooth surface for comparison with 1.0 m/s rough surface.

Figure 7. (g) Flow velocity field at the input flow velocity of 5.0 m/s in smooth surface. (h) The input flow velocity field with a flow velocity of 5.0 m/s brings the viewer closer to the smooth surface for comparison with 1.0 m/s rough surface.
With an increase in the input flow rate, a decrease in the transition zone width between the lower region with a minimum flow rate near the curved surface and the upper region with a maximum flow rate and the developed turbulent flow region is observed. It is obvious that this effect is due to the increase in pressure in the upper region of the flow.

For the convenience of assessing the impact of the input flow rate on the picture of the air flow near the solid curved wall, a graph was constructed showing the value of the flow rate at a height of 0.25 mm (figure 8 (i)) above the lowest point of the surface under study and a smooth surface (figure 8 (j)).

![Graph (i)](image1)

![Graph (j)](image2)

**Figure 8.** (i) Influence of the input flow rate on the nature of the envelope flow in rough surface. (j) Influence of the input flow rate on the nature of the envelope flow in smooth surface.
The result is shown in the figure 8. As noted earlier, with the increase in the flow rate, the pressure in the near-wall lower region increases and the flow of irregularities becomes noticeable with the repetition of the nature of the relief. When comparing the results for smooth and rough surfaces it can be seen that the flow of rough surfaces velocity is very much reduced.

5. Conclusion
The paper presents the calculation and modeling of flow patterns of rough surfaces in the program Comsol multiphysics at different speeds from 0.1 and 0.5 m/s to 5.5 m/s in increments of 0.5 m/s. As a result of the calculation, the distribution pattern of the airflow velocity along the curved surface under study is obtained.

With an increase in the input flow rate, a decrease in the transition zone width between the lower region with a minimum flow rate near the surface and the upper region with a maximum flow rate and the developed turbulent flow region is observed.

References
[1] Miroshin R N 1981 The intersection curves of the Gauss processes (publishing House Leningrad state University named after Zhdanov A A) 212
[2] Khalidov I A 2014 The use of poly-Gaussian stochastic processes to the modeling of flow over rough surfaces flow of rarefied gas Vestnik St. Petersburg University 1(59) 3 428-37
[3] Rinkevichius B S 1978 Laser anemometry (Moscow: Energy) 160
[4] Krasnikov G E, Nagornov O V and Starostin N V 2012 Modeling of physical processes using COMSOL Multiphysics (Moscow: national research nuclear University MEPhI) p 184
[5] Kharitonov V P 2012 Fundamental equations of fluid mechanics (MGTU named. N. Eh. Bauman) 65
[6] Durbin P A 1993 Application of a near-wall turbulence model to boundary layers and heat transfer Int. J. Heat and Fluid Flow 14(4) 316-23
[7] Frick P G 1998 Turbulence: models and approaches (Course of lectures Part 1 Perm State technical University Perm) 108
[8] Smirnov E M and Habakuk A V 2010 The flow of viscous fluid and turbulence model: the methods of calculation of turbulent flows (lectures Saint-Petersburg state Polytechnical University ) 127