Numerical simulation of stationary flow around a wing with a subsonic leading edge at M = 2 and 2.5

A N Semenov¹, ², V L Kocharin ¹ and N V Semionov¹

¹Khristianovich Institute of Theoretical and Applied Mechanics 630090 Novosibirsk, Institutskaya 4/1, Russia
²Institute of Geology and Mineralogy, SB RAS, 630090, Novosibirsk, Ac. Koptyuga ave. 3, Russia

semenov@itam.nsc.ru

Abstract. The results of direct numerical simulation of the stationary flow above the wing with a swept angle of 72° and a subsonic leading edge are presented. The simulation was carried out at Mach numbers 2 and 2.5 and unit Reynolds numbers 9e+6 m⁻¹ and 11e+6 m⁻¹.

1. Introduction

The occurrence of turbulence on the wing at supersonic flow speeds is of great interest to researchers in many countries. This is due to a practical interest since the transitional boundary layers occur on the wings of high-speed aircraft. At present, all works on numerical modeling or experiments on wings with subsonic leading edges [1-3] at supersonic speeds are related to the issue of stability and transition, and there is still no direct comparison of calculated and experimental data. One of the culprits is that the experimental data obtained at different facilities can radically differ [4]. Our work is focused on the numerical and experimental study of the development of disturbances in the boundary layer on the wing with a subsonic leading edge. Calculations are performed for a specific experimental model and completely replicate the flow parameters of the corresponding experiments. Here we present the first stage of this work, namely, the results of the numerical simulation of a stationary flow and the investigation of the parametric dependences of the flow.

2. Formulation of the problem

The equations of gas motion known as the Navier–Stokes equations, continuity, energy and state were solved using the ANSYS Fluent software package [5]:

$$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho v_i)}{\partial x_i} = 0$$
$$\frac{\partial (\rho v_i)}{\partial t} + \frac{\partial (\rho v_i v_j)}{\partial x_j} = \frac{\partial p}{\partial x_i} - \tau_{ij}$$
$$\frac{\partial (\rho E)}{\partial t} + \frac{\partial (\rho v_i E)}{\partial x_j} = \frac{\partial}{\partial x_j} \left( k \frac{\partial T}{\partial x_j} \right) + \frac{\partial}{\partial x_j} \left( \tau_{ij} v_i \right), \quad \tau_{ij} = \mu \left( \frac{\partial v_i}{\partial x_j} + \frac{\partial v_j}{\partial x_i} - \frac{2}{3} \frac{\partial v_k}{\partial x_k} \right)$$

Here $v$ is velocity vector with components $(u,v,w)$ in $x,y,z$ direction, $p, \rho, T$ are the pressure, density and temperature, $\mu = \mu_i \left( \frac{T}{T_i} \right)^{3/2}$ is the dynamic viscosity. Air is taken as ideal gas with

Content from this work may be used under the terms of the Creative Commons Attribution 3.0 licence. Any further distribution of this work must maintain attribution to the author(s) and the title of the work, journal citation and DOI.

Published under licence by IOP Publishing Ltd.
properties: $c_p = 1006.43 \text{ J/(kg} \cdot \text{K)}$ and thermal conductivity according to the kinetic theory. Density-based solver and explicit second-order schemes were used. Flux splitting was performed using the AUSM scheme.

The experimental model of the swept wing used for numerical modeling has a lenticular profile, the sweep angle along the leading edge is 72°, the sweep angle along the trailing edge is 58°, the sweep angle along the ¼ chord is 69.5°, the central chord is 500 mm, the end chord is 200 mm, wing half-span is 200 mm, and relative thickness along the central chord is 2.8%, relative thickness along the terminal chord is 1.33%. The model is shown in figure 1. The coordinate system was set as follows: $x$ is the streamwise coordinate, $z$ is the spanwise coordinate, and $y$ is the normal coordinate.

![Figure 1. Wing model.](image)

Two buffer zones were used to construct the computational grid. The first zone was obtained scaling the model to 130% and subtracting 110% from it. The second zone was obtained scaling the model to 110% and subtracting 100% from it. Next, the compound of these two zones was placed in the calculation box and the lower half was cut off, since we are interested in the flow above the wing. Free-stream conditions were set on the upper and front wall of the box, namely: at $M = 2$ $T_s = 164$ K $P_s = 9200$ and 11300 Pa, at $M = 2.5$ $T_s = 130$ K $P_s = 5300$ and 6500 Pa, which for both cases correspond to a unit Reynolds number of $9 \cdot 10^6 \text{ m}^{-1}$ and $11 \cdot 10^6 \text{ m}^{-1}$. The exit conditions were set on the rear wall. In the lower part adiabatic wall conditions were set on the wing and symmetry conditions were set in the zones outside the wing. Zero disturbances were placed on the side walls so that they did not affect the flow above the wing. A tetrahedral mesh was used with slight condensing in the outer buffer zone toward the wing. In the area near the wing, the grid was condensed near the surface and the leading edge at the same time. As a result, we had approximately 40–50 points in the boundary layer along the entire wing along the normal coordinate, while the total number of points did not exceed 80 million. The calculation was carried out in one run. In order to make sure that we reached convergence to the stationary flow, the residual control was used at several random points near the wing surface.

3. Results

The calculation results are given for the profiles located in the middle of the span and at a distance of 80 mm from the leading edge along the streamwise coordinate. Figure 2 shows the non-dimensional streamwise velocity component, which is non-dimensionalized with its external value. It can be noted that a slight curvature of the profile occurs with an increase in the unit Reynolds number, and this effect is most noticeable at $M = 2$ and $Re_1 = 9 \cdot 10^6 \text{ m}^{-1}$. 

Figure 2. Streamwise velocity profiles.

The spanwise velocity component, which is non-dimensionalized with the external velocity magnitude is shown in figure 3 at the same coordinates as the streamwise velocity in figure 2. It can be seen that an increase in the Mach number and the unit Reynolds number leads to an absolute increase of spanwise velocity at the level of the critical layer ($y \approx 0.0003$ m), and to the growth of its negative value beyond the boundary layer, which is typical for three-dimensional swept wings.

Figure 3. Spanwise velocity profiles.
If we look at the distribution of temperature that is non-dimensionalized with its external value (figure 4), then we can say that it is restored to a single value on the adiabatic wall, namely 270 K. In this case, the influence of the Reynolds number on the shape of the profile is clearly visible, the profile becomes more filled with decreasing static pressure.

![Temperature profiles](image)

**Figure 4.** Temperature profiles.

It makes no sense to bring the static pressure, because at this distance from the leading edge it changes by no more than 1% of the value in the external flow, so we can say that it is constant. Therefore, based on the ideal gas equation, we can say that the density reflects the trend of the temperature distribution.

### 4. Conclusion

Thus, we successfully carried out the first stage of numerical simulation, we managed to solve the problems with the grid and calculate the stationary flow. Now we can expand this research by studying how the profiles are distorted when the angle of attack changes, and also try to solve the unsteady problem with the study of the development of disturbances inside the boundary layer and make a direct comparison with experiment.

### Acknowledgments

The study was conducted at the Joint Access Center “Mechanics” of ITAM SB RAS. The work is supported by Russian Science Foundation (grant number №19-08-00772).

### References

[1] Arnal D, Casalis G and Piot F 2008 *Proc AAAF 43ème Coll Aero Appliquée*

[2] Duan L, Choudhari M and Li F 2014 *Proceedings of the Center for Turbulence Research Summer Program*

[3] Hetsch Ti and Rist U 2004 The effect of sweep on laminar separation bubbles *IUTAM Symposium on Laminar-Turbulent Transition*

[4] Semionov N V, Yermolaev Yu G, Kosinov A D, Dryasov A D, Semenov A N and Yatskikh A A 2016 *AIP Conference Proceedings* 177

[5] ANSYS Inc. 2013 *ANSYS Fluent Theory Guide* 275 Technology Drive Canonsburg 15317 USA