Effect of the wall properties on the shock wave / laminar boundary layer interaction

A I Kutepova1, P A Polivanov1 and A A Sidorenko1

1Khristianovich Institute of Theoretical and Applied Mechanics SB RAS, 630090, Institutskaya, 4/1, Novosibirsk, Russia

E-mail: a.kutepova@g.nsu.ru

Abstract. The CFD calculations have been performed based on RANS approach with the Menter transition model [1]. The boundary layer developing on a flat plate was considered. To generate the incident shock wave, a wedge was installed over the plate. The wall was represented by one cm thick solid flat plate made from aluminum and polyacetal. Flow parameters were $M_\infty=1.41$, $P_0=0.7$ bar, $T_0=293K$. As a result of solving the conjugate heat transfer problem, the SWBLI flow was studied. The temperature and pressure distributions on the wall for different surface materials were obtained. It was found that the streamwise heat fluxes inside the wall can lead to a change of the separation zone length.

1. Introduction

Transonic aircraft of the next generation should have significantly lower CO$_2$ emission and have higher fuel efficiency [2]. These can be achieved by improving the engines, aerodynamics of the aircraft and using lighter materials. One of the means of drag reduction of the aircraft is to provide the laminar flow in the boundary layer on the wing and engine nacelle. The adverse pressure gradient generated by a shock wave can lead to separation of the boundary layer [3]. The separation of the laminar boundary layer results in significant instability leading to earlier laminar-turbulent transition. For correct calculations of the aircraft aerodynamic characteristics, an exact definition of the transition position is mandatory. The wall properties, particularly temperature distribution can significantly affect the flow characteristics at Shock Wave/Boundary layer interaction (SWBLI). In CFD calculations it is commonly assumed that the wall boundary conditions are either adiabatic or isothermal. In reality, heat fluxes inside a solid wall can lead to a redistribution of wall temperature.

In the case of a cold wall (the temperature of the wall is less than the recovery temperature), the separation zone decreases, while for the hot wall the opposite effect is observed. An example of the influence of wall temperature can be found in [4], in which the SWBLI zone was studied by direct numerical simulation for the case of an incoming turbulent boundary layer. This work demonstrates a significant change not only of a stationary flow in the interaction zone but also a significant change in non-stationary characteristics in the shear layer. Therefore, when modeling SWBLI, it is important to set the temperature of the model wall correctly.
Adiabatic conditions on the model wall are often assumed when simulating experimental test cases performed in blow-down wind tunnels. But along the SWBLI zone, the recovery temperature changes significantly. This is due to a change in the Mach number at the boundary of the shear layer and heat fluxes in the shear layer. The emergence of the longitudinal temperature gradient on the surface of the model should lead to the appearance of heat fluxes within the experimental model, which will lead to a redistribution of temperature on the surface of the model. The influence of this phenomenon can be estimated by solving the conjugate problem.

An example of such calculations for high-speed flows can be found in papers [5, 6]. In the work [5] it was shown that even for unseparated flows in high Mach number flight, for a good comparison with the experimental results it is necessary to calculate the heat fluxes in the model. A good agreement between the numerical results and the experiments was achieved in the work [6], in which the cooling efficiency of the turbine blades was evaluated. However, this phenomenon is often not taken into account when modeling SWBLI.

Heat flows generated inside the model can be used to control the flow. In the work [7] a method of flow control based on the creation of transverse jets is proposed. The pressure for generating a jet is created by the evaporation of a liquid, which is heated by taking heat from an external stream with a high stagnation temperature.

The temperature effect is important not only for improving the accuracy of the calculation. Another important point is to improve the accuracy of the experimental methods. For example, when using in the experiment the method of measuring the surface temperature with a thermal imaging camera, incorrect interpretation of the data is possible since the heat fluxes inside the model can lead to a significant change in temperature fields. In the work [7] a complex combination of materials was used, which significantly reduced the effect of heat fluxes from the lower and fore part of the model on the flow field under study.

The purpose of this study is to estimate the influence of the wall material on the development of separated flow and the position of the laminar-turbulent transition in the shock wave/boundary layer interaction.

2. Problem formulation

The flow parameters and geometry of the computational domain correspond to the experiment described in [3]. The computational domain was as close as possible to the real geometry, which consisted of a plate, a wedge and two bypass channels located in the lower and upper parts of the domain. The presence of sharp steps and caverns in the bypass channels was due to the design considerations. In the experiment, the region of interaction of the shock wave generated by the wedge with the boundary layer developing on a flat plate was studied. Therefore, when constructing a computational grid in a given area, the grid refinement was used. All calculations were performed at a wedge angle of attack of 4°. The main material of the plate was polyacetal. To ensure the requirements of rigidity the nose section was made of steel. The heat fluxes inside the solid body were calculated only for the plate. The boundary conditions for the remaining walls were the condition of zero heat flux.

The computational domain was covered by a regular hexahedral computational grid. The total number of the computational domain cells finally amounted to 1.1 mil. The problem was simulated numerically with the aid of the software package ANSYS Fluent. The steady Navier–Stokes equations were solved with the aid of a density-based solver. An implicit second-order scheme in time and space
was used together with the AUSM method of splitting the convective fluxes. The calculations were performed using the Menter transition model $\kappa-\omega-\gamma Re_\theta$ and the turbulent model $\kappa-\omega$ SST. The freestream parameters were specified at the inlet boundary of the computational domain. At the outlet boundary, the flow expiration condition was set. The capability of conjugate heat transfer calculation is accomplished by integrating a heat conduction procedure in a solid model (plate).

In total, three calculation cases with different plate material were considered. The base case was the calculation, where the main material was polyacetal, and the nose part was steel. In the second case, the entire plate was steel. In the third configuration, adiabatic boundary conditions were set on the plate wall.

![Figure 1. Velocity distribution in the computational domain](image)

### 3. Calculation results

Figure 1 shows the velocity field for the base case. On the left input boundary, the flow parameters were specified, which corresponded to the values calculated for the nozzle under the required conditions. It is seen that when the incident flow interacts with the wedge, a shock wave is formed, which interacts with the boundary layer developing on the upper part of the plate. The deflection angle of the lower nose of the plate is 9°, which leads to the formation of a powerful shock wave. In the absence of the bypass channel, the interaction of this shock wave with the lower wall could lead to the choking of the wind tunnel. The presence of a sudden expansion formed by the bypass channel leads to the formation of rarefaction waves weakening the shock wave formed by the nose of the plate. Further downstream, the flow accelerates in the fan of the rarefaction waves generated in the zone of the end of the wedge-shaped part of the plate. This leads to a significant decrease in pressure, which becomes less than the static pressure of the bypass channel. As a result, an oblique shock wave is formed with the formation of the developed separation zone in the lower part of the plate. The main difference between the flow formed in the upper bypass channel is the small geometrical dimensions of the wedge. This leads to the formation of a weaker shock wave and rarefaction waves on its upper part. The absence of a significant circulation zone in the upper bypass channel is due to the small geometrical dimensions of the wedge.
Figure 2. Temperature distribution for the base case in the computational domain

Figure 2 shows the temperature field, from which the formation of the region with the minimum temperature in the acceleration section under the plate is seen. This is accompanied by an increase in the Mach number, which should lead to a decrease in the recovery temperature on the bottom wall of the plate. Since the difference in the Mach number on the lower and upper walls of the nose part of the plate is not large (at the top $M = 1.45$, at the bottom $M = 3$), there is no significant difference in the recovery temperature. Therefore, the nose of the plate has a uniform temperature distribution throughout the area. Since the flow velocity is subsonic in the separation zone under the plate, the temperature of the bottom wall coincides with the flow temperature in the separation zone. It can be seen in Fig. 3 that in the separation zone the stagnation temperature is about 230K, which is significantly lower than the total temperature of the free stream 293K. As a result, heat flux is generated, directed from the upper wall to the lower one, which reduces the temperature on the upper wall of the plate in the SWBLI zone.

Figure 3. The distribution of the total temperature for the base case in the computational domain
Figure 4. The temperature distribution in the plate for the calculated cases of base (a); second configuration (b)

Figure 4 presents the temperature distribution inside the solid body for the first and second configurations. It can be seen, that for the case of an all-steel plate (Figure 4b), due to the high value of the thermal conductivity coefficient, the plate temperature is equalized along the normal coordinate. Before the SWBLI zone, the plate temperature rises, and in the wake zone, it falls. The reason for this is the redistribution of the stagnation temperature in the separation zone under the plate.

In the case where the main material of the plate was polyacetal, one can see a significant temperature non-uniformity for the plastic part of the model along the normal coordinate. This is due to the low thermal conductivity of the plastic, which makes the temperature distribution closer to the adiabatic one, so the temperature in the SWBLI zone increases in the model due to the rise in the recovery temperature (the Mach number decreases).

Figure 5. Comparison of experimental results with calculated temperatures for the upper part of the plate (a); pressure distribution in the stream (b)
The coordinate $x=130$ corresponds to the intersection of the incident shock wave generated by the wedge with the plate. In this region, an increase in temperature occurs for the adiabatic case. It was noted above that the reason for this is the drop in the Mach number. In the case of polyacetal, the temperature distribution is qualitatively similar to the adiabatic one. However, there is a decrease in temperature gain and overall cooling of the upper wall. The most interesting case is the steel plate. The obtained temperature distribution (Figure 5a) does not allow determining the flow structure on the upper part of the plate. The red curve corresponds to the experimental data obtained on a plastic model for a turbulent boundary layer. Temperature measurement was carried out by thermal imaging method. One can see a good qualitative agreement of the results. The difference obtained may be attributed to the absence of three-dimensional effects in a two-dimensional calculation, as well as the neglect of nonstationary phenomena in separation zones of RANS methods. Despite the significant effect of the plate material on the temperature of the upper wall in the SWBLI zone, this does not have a significant effect on the flow structure, as can be seen from the pressure distribution.

4. Conclusion

As a result of the calculations, it was shown that the material of the experimental model has a significant effect on the temperature distribution on the surface of the model. A significant effect of the flow formed in the lower part of the model on the temperature distribution in the zone of experimental measurements was found. Most likely, the reason for the discrepancy between the calculated and experimental temperature distributions is the limitations of flow modeling under the plate. RANS modeling methods do not allow conducting simulation with sufficient accuracy of the circulation zone found under the bottom surface of the plate.

Acknowledgments

The work was supported by the grant of RSF 18-19-00547.

References

[1] Menter F R, Langtry R B, Likki S R, Suzen Y B, Huang P G and Völker S 2004 J. Turbomach, 128(3), 413-422.
[2] Budd T, Suau-Sanchez P 2016 Research in Transportation Business & Management, 21, 68-75.
[3] Polivanov P A, Sidorenko A A, Maslov A A 2015. 53rd AIAA Aerospace Sciences Meeting 2015 (Kissimmee, Florida, USA, 5-9 Jan., 2015): Conference Paper. 18 No.AIAA 2015-1974.
[4] Bernardini M, Asproulias I, Larsson J, Pirozzoli S and Grasso F 2016 Physical Review Fluids 1(8):1-18.
[5] Chandra Murty, Manna P and Debasis Chakraborty 2012 Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering 227 1672-1681
[6] Peng Wang, Yu Li, Zhengping Zou, Weihao Zhang 2013 Propulsion and Power Research 2(1):56–69.
[7] Liang Zhu, Xiong Chen, Yingkun Li, Omer Musa, Changsheng Zhou 2018 Acta Astronautica 142, 300-313.