BIFURCATION OF TRANSONIC FLOW IN A CHANNEL WITH A CENTRAL BODY

A. Riabinin 1, A. Suleymanov 1

1 Department of hydroaeromechanics, Faculty of mathematics and mechanics, St.Petersburg State University, 28 Universitetsky prospekt, Peterhof, 198504 St.Petersburg, Russia

Abstract

2D transonic flow in a channel of variable cross-section with a central body is studied numerically using a solvers based on the Euler and Reynolds-averaged Navier-Stokes equations. The flow velocity is supersonic at the inlet and outlet of the channel. Between the supersonic regions, there is a local subsonic region. Computations reveal a hysteresis in the shock position versus the inflow Mach number. In the certain range of inlet Mach numbers there are asymmetrical solutions of the equations.

Keywords: transonic flow, shock wave, instability, hysteresis.

1 Introduction

In the 2000s, it was demonstrated that the transonic flow around an airfoil with surface of small curvature is sensitive to the changes in the parameters of the free flow [1]. Sensitivity is caused by the interaction of local supersonic regions near the airfoil. Two supersonic regions are formed near the area of small curvature. The leading supersonic region is terminated by shock wave. Behind the shock, the flow is subsonic. The subsonic region is terminated by a sonic line behind which the flow is supersonic again. The increase in the Mach number leads to arising and expansion of supersonic regions and rapprochement. At the moment of coalescence of two regions the pressure distribution and the lift force abruptly change. This phenomenon takes place for a number of symmetrical airfoils, and for an asymmetrical airfoil whose upper surface is nearly flat in the midchord region [1, 2].

In the paper [3], the instability of the shock wave position is studied in the case of transonic flow in a channel with an expansion corner. In this paper, we consider the flow in the channel with two expansion corners and with a central body. It should be noted that the problems of hysteresis of shock waves were studied in [4, 5].

2 Formulation of the Problem. A Numerical Method

We consider the 2D flow in the channel. Straight segments constitute lower and upper walls of the channel:

lower wall: \[ 0 < x < 2.3, \quad y = -0.88 - 0.0521x \quad \text{and} \]
\[ 2.3 < x < 3.0, \quad y = -1 - 0.229(x - 2.3), \]
upper wall: \[ 0 < x < 2.3, \quad y = 0.88 + 0.0521x \quad \text{and} \]
\[ 2.3 < x < 3.0, \quad y = 1 + 0.229(x - 2.3). \]

The coordinates \((x, y)\) are dimensional. Here and further coordinates are given in meters. There is a central body in the channel. Body’s contour consists of three segments: \[ 1.4 < x < 3.0, \quad y = \pm 0.01 \pm 0.119(x - 1.4) \quad \text{and} \]
\[ x = 1.4, \quad -0.01 < y < 0.01. \] Inlet and outlet boundaries are segments too:

inlet: \[ x = 0, \quad -0.88 < y < 0.88, \]
outlet: \[ x = 3.0, \quad -1.16 < y < -0.2 \quad \text{and} \quad 0.2 < y < 1.16. \]
2D hybrid mesh was generated using the package Gmsh [6]. This mesh was used for calculations in the package SU2 [7]. A program written in Pascal language transformed it into a 3D mesh, whose lateral size was equal to one element. The transformed mesh is in the TGrid / Fluent format [8], which is suitable for the calculation in the commercial package Ansys CFX [9].

Most of the calculations are performed with the mesh that consists of 120786 elements. The mesh was fined near walls for valid simulation of boundary layer in solutions of Reynolds-averaged Navier-Stokes equations. However, a few calculations are carried out with more fined mesh near walls with 125314 elements. Another mesh consists of 109864 elements. Calculations demonstrated the independence of the results on mesh size.

In the solutions of Euler equations unstructured mesh was used. This mesh consists of 75507 elements.

On the inflow boundary temperature $T_{in} = 250$ K, the Mach number $M_{in}$ and Reynolds number $Re$ of the free-stream flow on the base of the length of 1 m are set. The no-slip condition and vanishing flux of heat are used on the wall. Specific heat of air at constant pressure is equal to 1004.4 J / (kg K). Molar mass is equal to 28.96 kg/kmol.

Solutions of Reynolds-averaged Navier-Stokes equations are obtained with SU2 and Ansys CFX - 13 finite volume solvers. The nondimensional thickness of the first mesh layer $\gamma^+$ is approximately equal to 1. The models of turbulence Spalart-Allmaras (SU2) and $k-\omega$ SST (Ansys CFX) are used.

### 3 Results of Calculations

Mach number at the channel inlet $M_{in}$ varies from 1.2 to 1.4. Fig. 1 shows two variants of shock waves in the channel. Both variants are characterized by the presence of subsonic zones. At Mach number $M_{in} = 1.23$ (see Fig. 1 a) there is a large subsonic region between the shock wave and the sonic line. The distance between the inlet boundary and the shock wave is denoted as $x_s$.

![Figure 1: Two variants of shock waves in the channel. Re= 5.6 \cdot 10^6. (a) $M_{in} = 1.23$, (b) $M_{in} = 1.33$. 1 — shock wave, 2 — sonic line, $x_s$ — the distance from the inlet up to shock wave. Solutions are obtained with package SU2.](image)
If $M_{in}$ increases step-by-step from 1.23 to 1.32, then the qualitative pattern of the flow persists (see Fig. 1a) though the shock shifts downstream. Meanwhile if $M_{in}$ is further increased to 1.33, then the supersonic regions coalesce abruptly, and we arrive at the flow pattern shown in Fig. 1b.

At Mach number $M_{in} = 1.33$ (see the Fig. 1b) there is a single supersonic region. In addition, there are three small local subsonic zones. One of them is situated between the bow shock and the central body. Others subsonic small regions adjoin to the side surfaces of the central body. They are between shock wave $I$ and sonic line $2$. The distance between the inlet boundary and the shock wave is denoted by $x_s$ as in the case of large subsonic zone.

Decreasing of inlet Mach number from $M_{in} = 1.33$ down to 1.24 leads to decreasing of $x_s$ and increasing of the size of subsonic regions adjoined to the side surfaces of the central body. At $M_{in} = 1.23$ the single supersonic region splits into two regions. So computations reveal a hysteresis in the shock position $x_s$ versus the inflow Mach number $M_{in}$.

In the range of hysteresis there is a range of Mach numbers $M_{in}$ in which asymmetrical stable solutions exist. Asymmetric solution can be obtained by introducing of asymmetry in the boundary conditions at the initial stage of calculation. For example, we started the calculation at the angle of attack $\alpha \neq 0$. Fig. 2 shows the four flow regimes at Mach number $M_{in} = 1.31$.

![Figure 2: Four flow patterns at $M_{in} = 1.32$. Re= 5.6 \cdot 10^6. Solutions are obtained with package SU2.](image)

Realization of specific mode depends on the flow history. Fig. 2a shows the pattern with a single large subsonic region. This pattern was obtained by increasing the inlet Mach number $M_{in}$ from 1.23 to 1.31. Two asymmetrical regimes in the Fig. 2b and Fig. 2c were obtained from asymmetrical initial conditions. These patterns have the large subsonic region at one side of the central body and the small subsonic region attached to another side of the body. Fig. 2d presents the symmetrical flow with three small subsonic regions. Pattern in the Fig. 2d was obtained by decreasing the inlet Mach number $M_{in}$ from 1.33 to 1.31.

Fig. 3 demonstrates the dependence of the distance $x_s$ on Mach number $M_{in}$ for two values of Reynolds number Re.
Figure 3: The distance from inlet boundary to shock wave $x_s$ as a function of Mach number $M_{in}$, solver SU2. 1-4 — $Re=3.4 \cdot 10^7$, 5-8 — $Re=5.6 \cdot 10^6$, 1, 5 — two supersonic regions, 2, 6 — single supersonic region, 3, 4, 7, 8 — asymmetrical modes.

The graphs corresponded to different Reynolds number are close to one another.

The solutions of the Euler equations for non-viscous fluid are similar to the solutions of the Navier-Stokes equations. However, the boundaries of hysteresis range are shifted. The range of $M_{in}$ in which there are asymmetrical solutions is increased significantly. In the Fig. 4 the dependencies of the $x_s$ on $M_{in}$ for nonviscous and viscous fluids are shown.

Figure 4: The distance from inlet boundary to shock wave $x_s$ as a function of Mach number $M_{in}$. Solver SU2. 1-4 — viscous fluid, Re=5.6 \cdot 10^6, 5-8 — non-viscous fluid.

The influence of solver and the model of turbulence are presented in the Fig. 5.
Figure 5: The distance from inlet boundary to shock wave $x_s$ as a function of Mach number $M_{in}$. 1-4 — solver SU2, turbulence model Spalart-Allmaras, $Re=5.6 \cdot 10^6$, 5-8 — solver Ansys CFX, turbulence model $k-\omega$ SST, $Re=3.4 \cdot 10^7$.

The graphs corresponded to different solvers and models of turbulence slightly differ from one another. The largest difference was observed in the vicinity of the upper boundary of the hysteresis range for the two supersonic regions.

4 Conclusions

The numerical simulations of 2D flow in the channel with the central body revealed a large hysteresis in the shock wave position as a function of the Mach number specified at the inlet. In the certain range of inlet Mach numbers there are asymmetrical solutions of the Euler and Reynolds-averaged Navier-Stokes equations. These results are obtained with solvers SU2 and Ansys CFX and two models of turbulence: Spalart-Allmaras and $k-\omega$ SST.

Acknowledgment

This research was performed using computational resources provided by the Computational Center of St. Petersburg State University (http://cc.spbu.ru).

References

