Numerical study of transonic flow in a channel at various back pressures

A.G. Kuzmin[a]

[a]*St. Petersburg State University, St. Petersburg, Russia*
(e-mail: a.kuzmin@spbu.ru)

Studies of high-speed airflow in convergent-divergent or curved channels are of practical interest, e.g, for the advanced design of supersonic aircraft engines. The efficiency of engine depends crucially on the intake that decelerates the incoming atmospheric airflow to velocities suitable for fuel burning in the combustor. In on-design conditions, the deceleration of supersonic flow is accomplished through several oblique shock waves located inside the intake. Such a flow regime is known as started intake [1]. Meanwhile, if the inflow Mach number M_∞ decreases or the pressure imposed at the exit p_{back} increases, then the shock waves shift upstream and eventually are expelled from the intake. The latter regime, which is called an unstarted intake, produces considerable losses in the engine thrust.

A number of works addressed the flow behavior in convergent-divergent intakes and bent channels under changes in freestream Mach number M_∞ or angle of attack [2]. Also, a point of importance is the flow behavior under changes in the back pressure p_{back}. Zhao *et al.* [3] carried out an experimental study of the effect of back pressure on the flow in a channel whose lower wall involves a bend of 8°; a noticeable hysteresis was detected during the unstart/restart of the intake.

In this paper, we consider the same channel as in [3] and perform a numerical study of the flow. The flow is governed by the system of Reynolds-averaged Navier–Stokes differential equations [4] with respect to static pressure $p(x, y, t)$, temperature $T(x, y, t)$, and velocity components $U(x, y, t), V(x, y, t)$, where (x, y) are the Cartesian coordinates and t is time.

Figure 1 illustrates a geometry of the channel and outer boundaries of the computational domain. Details of the geometry are available in [3]. On the inflow boundary of computational domain, we impose velocity components $U_\infty = 592.08$, $V_\infty = 83.21$ m/s, temperature $T_\infty = 122.1$ K, and pressure $p_\infty = 4,300$ Pa, which correspond to $M_\infty = 2.7$, as in [3]. At the exit of channel, we prescribe the pressure p_{back}. On the walls of channel, the flow velocity vanishes. Initial conditions are either the uniform freestream or non-uniform flow field obtained for another value of p_{back}. The stated initial-boundary value problem was solved numerically with an ANSYS CFX finite-volume solver on an unstructured mesh constituted by 948,119 cells. At steady values of p_{back}, transient solutions of the problem typically showed a fast convergence to steady states. The obtained steady solutions $p(x, y), T(x, y), U(x, y), V(x, y)$ made it possible to identify locations of shock waves in the channel, as well as unstart/restart conditions.

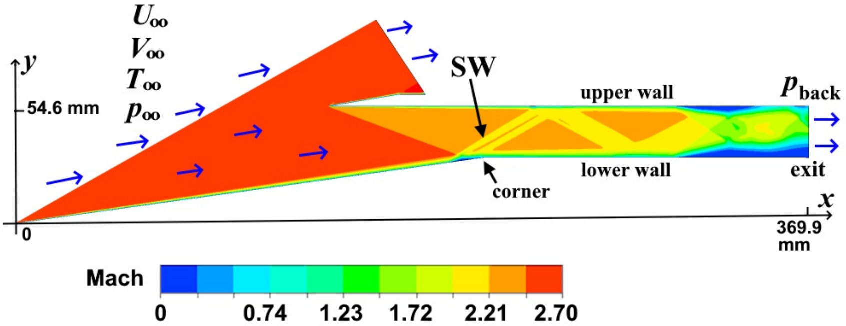

Figure 1: Mach number contours in the channel at $M_\infty = 2.7$, p_{back}=24,000 Pa

In particular, the solutions revealed a shock wave SW induced by the boundary-layer separation ahead of the lower wall corner, see Fig. 1. Changes in the pressure p_{back} showed that the shock SW persists when p_{back} increases step-by-step from 1,000 Pa to 32,300 Pa, whereas the system of oblique shocks gradually shifts upstream in the region between the corner and exit. If p_{back} further increases step-by-step to 34,100 Pa, then the shock SW shifts upstream towards the entrance of channel and exhibits noticeable oscillations. After that, a decrease in p_{back} from 34,100 Pa to 32,200 Pa results in a gradual swallowing of SW. Further decrease in p_{back} does not change the location of SW, though shifts the oblique shocks downstream of the bend towards the exit.

If p_{back} rises to values larger than 34,100 Pa, then the shock SW jumps upstream at a distance from the entrance; therefore, SW becomes expelled from the channel. In this case, numerical solutions show that any subsequent decrease in p_{back} cannot produce the swallowing of SW, i.e., do not provide a restart.

We notice that the abrupt transitions between different flow regimes are caused by the instability of the boundary-layer separation from the lower wall in contrast to the case of smaller freestream Mach number M_∞ [2] when instability is caused by instability of the shock wave interaction with the flow expansion region over the corner.

The results of the numerical study are in good agreement with experimental data documented in [3]. Minor discrepancies in oblique shock locations in the region between the corner and exit of channel at decreasing p_{back} are explained by different ways of imposing the back pressure, since in [3] a transverse jet was used to rise the pressure near the exit, whereas in our simulations the average pressure over the exit section is set directly.

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

References

[1] D. Panchal, D. Chayani, *Review paper on the unstarting of the supersonic air intake*, International Journal of Engineering Research and Technology, 2021, 10(12), 11-15.

[2] A. Kuzmin, *Non-uniqueness of transonic flow in an intake-type channel*, Journal of Physics: Conference Series, 2019, 1392, Article ID 012012, 1-6.

[3] Y.L. Zhao, Y.Y. Zhou, Y.X. Zhao, *Experimental study of the unstart/restart process of a two-dimensional supersonic inlet induced by backpressure*, Journal of Applied Fluid Mechanics, 2022, 15(2), 415-426.

[4] H. Tennekes, J.L. Lumley, *A first course in turbulence* (14th ed.), MIT Press, Cambridge, 1992, 360 p.