Numerical investigation of air flow in a supersonic wind tunnel

S M Drozdov and A S Rtishcheva

Central Aero Hydrodynamic Institute named after Professor N. E. Zhukovsky (TsAGI)
Russia, 140180 Zhukovsky, Zhykovsky st., 1

Abstract. In the framework of TsAGI’s supersonic wind tunnel modernization program aimed at improving flow quality and extending the range of test regimes it was required to design and numerically validate a new test section and a set of shaped nozzles: two flat nozzles with flow Mach number at nozzle exit M=4 and M=5 and two axisymmetric nozzles with M=5 and M=6. Geometric configuration of the nozzles, the test section (an Eiffel chamber) and the diffuser was chosen according to the results of preliminary calculations of two-dimensional air flow in the wind tunnel circuit. The most important part of the work are three-dimensional flow simulation results obtained using ANSYS Fluent software. The following flow properties were investigated: Mach number, total and static pressure, total and static temperature and turbulent viscosity ratio distribution, heat flux density at wind tunnel walls (for high-temperature flow regimes). It is demonstrated that flow perturbations emerging from the junction of the nozzle with the test section and spreading down the test section behind the boundaries of characteristic rhomb’s reverse wedge are nearly impossible to eliminate. Therefore, in order to perform tests under most uniform flow conditions, the model’s center of rotation and optical window axis should be placed as close to the center of the characteristic rhomb as possible. The obtained results became part of scientific and technical basis of supersonic wind tunnel design process and were applied to a generalized class of similar wind tunnels.

1. Introduction

Modern wind tunnels are expected to provide a highly uniform flow in the test section where objects under study are placed. Therefore a part of TsAGI’s research activities is devoted to flow simulation in wind tunnel circuits during the design process of their elements. In the framework of TsAGI’s supersonic wind tunnel modernization program aimed at improving flow quality and extending the range of test regimes it was required to design and numerically validate a new test section and a set of shaped nozzles: two flat nozzles with flow Mach number at nozzle exit M = 4 and M = 5 and two axisymmetric nozzles with M = 5 and M = 6. The diameter of axisymmetric nozzle inlet section is 140 mm. Flat nozzle inlet section dimensions are 140×200 mm. The new nozzles must integrate with the existing wind tunnel circuit: a heater, a prechamber, a supersonic diffuser and a gas outlet system.

The importance of present investigation stems from the problem of choosing an optimal geometric configuration of each element of a wind tunnel and integrating them in a circuit that provides better flow quality in the test section. Numerical flow investigation methods allow accurate detection of flow non-uniformities, analysis of the causes of flow perturbations, determination of heat flux at wind tunnel walls.
The obtained results became part of scientific and technical basis of supersonic wind tunnel design process and were applied to a generalized class of similar wind tunnels.

2. Problem formulation and brief description of calculation methods

At the initial stage of designing the new shaped nozzles, their contours were determined using the method of characteristics \([1–3]\) for a uniform flow with Mach number \(M = 4; 5; 6\) at nozzle exit. The obtained geometry was then applied for meshing and numerical simulation of viscous heat-conducting air flow in the new nozzles using ANSYS Fluent software. The parameters specified at inlet sections of simulated nozzles (total temperature \(T_0\) and total pressure \(p_0\)), as well as air mass flow \(G\) through the nozzles, are presented in table 1. In all calculations, a constant temperature of 300 K was imposed at nozzle walls.

| Nozzle type     | M     | \(T_0\), K | \(p_0\), atm | \(P_0\), Pa | \(G\), kg/s |
|-----------------|-------|------------|--------------|-------------|-------------|
| Flat            | 4.0   | 300        | 15           | 1519875     | 23.828      |
| Flat            | 5.0   | 425        | 30           | 3039750     | 21.445      |
| Axysimmetric    | 5.0   | 425        | 30           | 3039750     | 6.756       |
| Axysimmetric    | 6.0   | 530        | 70           | 7092750     | 6.507       |

In the numerical investigation, two formulations were considered: 1) two-dimensional flat and axysimmetric problems with a large set of geometric configurations used for finding the optimal one; 2) three-dimensional (final) simulations of the entire wind tunnel circuit with a new Eiffel test section and nozzles with a rectangular cross-section for Mach numbers \(M = 4\) and \(M = 5\).

The main tool of numerical investigation was ANSYS Fluent software, in which all procedures of numerical solution of fluid mechanics equations were based on the finite volume method using an implicit scheme with a second-order approximation in space.

Two-dimensional planar, two-dimensional axysimmetric and three-dimensional stationary Navier-Stokes equations for compressible perfect air \(c_p=1006\ J/kg/K, \gamma=c_p/c_v=1.4\) were considered in the calculations.

Viscosity dependence on temperature was defined by the Sutherland formula:

\[
\mu(T) = 1.716 \times 10^{-5} \left( \frac{T}{273.11} \right)^{1.5} \left( \frac{273.11 + 110.56}{T + 110.56} \right). \tag{1}
\]

Heat conduction coefficient as a function of temperature was defined using tabulated data and a polynomial approximation:

\[
\lambda(T) = 7.486 \times 10^{-5} + 9.629 \times 10^{-5} T - 3.077 \times 10^{-8} T^2. \tag{2}
\]

In all calculations a developed turbulent flow was assumed and the Spalart-Allmaras model was used.

3. Results of two-dimensional numerical flow simulation in a supersonic wind tunnel

In order to find an optimal geometric configuration of the Eiffel test section and to estimate the efficiency of a supersonic wind tunnel with the new nozzles and the new test section, multiple variants of 2D planar geometry and mesh for the nozzle, the Eiffel test section and the diffuser were created. Calculations were made for Mach numbers \(M = [4, 5, 6]\) in the test section. As an illustration, results obtained at a regime with \(M=4\), which is the most problematic regime regarding flow core dimensions in the test section, will be discussed. The initial variant of test section geometry was (2D_geom_0).

The following boundary conditions were used at prechamber inlet section: total pressure \(p_0=1519875\ Pa\), static pressure \(p_{st}=10010\ Pa\), total temperature \(T_0=300\ K\). The boundary condition at diffuser outlet \(p_{st}=80000\ Pa\) was chosen to ensure that the flow separation near the wall caused by an unfavorable pressure gradient would form right after the diffuser throat.
The wind tunnel geometry used in calculations consists of three parts (figure 1): 1 is a flat supersonic nozzle; 2 is an Eiffel chamber; 3 is a supersonic diffuser. The main results were obtained using two meshes. The first mesh consisted of 1,404,600 cells; the second mesh was created by increasing cell density at boundary and high-gradient zones in the first mesh and consisted of 2,708,463 cells. Calculations showed that Mach number values in similar points of the two meshes diverged by less than 0.02%. The results presented below were obtained using the second mesh.

Figure 1. Wind tunnel geometry (2D_geom_4) used in calculations.

Numerical investigation of (2D_geom_0) geometry variant demonstrated that a reverse flow in test section periphery collided with the stream flowing out of the nozzle and generated considerable N-wave perturbations. Also, by the center of the test section (x = 234 mm) only 30% of the flow core was preserved. Adding a deflecting baffle oriented along the stream did not reduce the perturbations. The presence of “pockets” in Eiffel test section periphery behind the diffuser inlet caused flow instability and oscillations. Thus, the initial (2D_geom_0) geometry variant did not comply with the required flow quality for a model installed in the center of the test section. Therefore a number of extra variants of wind tunnel geometry were considered:

2D_geom_1: the nozzle is joined with the external wall of the test section. The “pockets” in Eiffel test section periphery behind the diffuser inlet are removed: the flat back panel of the test section is seamlessly joined with the diffuser.

2D_geom_2: the nozzle is joined with the internal wall of the test section. The Eiffel test section has a flat back panel seamlessly joined with the diffuser.

2D_geom_3: the nozzle is moved into the test section by 15 mm. The Eiffel test section has a flat back panel seamlessly joined with the diffuser.

2D_geom_4: the nozzle is moved into the test section by 40 mm. The Eiffel test section has a flat back panel seamlessly joined with the diffuser.

Calculations showed that 2D_geom_4 geometry variant provided the best flow quality, therefore the wind tunnel circuit considered in further investigation was based on this geometry. In a flow with 2D_geom_4 wind tunnel geometry, compared to 2D_geom_1, the characteristic rhomb is shifted to the right and occupies a larger part of the test section, which extends model testing capabilities (figure 2). A Mach number value of M = 4.101 is attained at nozzle exit. The stream leaving the test section decelerates due to emergence of a series of oblique shocks and reaches the diffuser throat at a slower speed (M = 3.6). Total pressure of the flow core in the supersonic nozzle and in the test section remains equal to the pressure in the prechamber $p_0 = 15$ atm. A rise of static pressure is observed near diffuser inlet opening zone where static pressure attains a value of $p_{st} = 0.24$ atm. Air temperature near diffuser inlet walls is close to wall temperature $T_w = 300$ K, therefore heat flux rate to the surface is very small.
Figure 2. Results of numerical flow simulation in a wind tunnel (2D_geom_4 geometry) at M=4.

In order to find an optimal geometric configuration of the diffuser inlet, the second series of calculations were made at a regime with a total temperature $T_0=530$ K in the prechamber. Since a rise
of static pressure and temperature takes place at diffuser inlet, the optimal geometry of the diffuser inlet corresponds to minimal heat flux rate values at its walls. Calculation results for three geometric configurations are shown in figures 3-5. One can see that diffuser inlet geometry affects the flow in the test section of the wind tunnel. A clear difference can be observed for the characteristic rhomb at nozzle exit and in the test section, as well as for the series of shocks in the diffuser. A rise of static pressure takes place at diffuser inlet. The average Mach number value in the flow core is \( M = 4.105 \) for G72 geometry, \( M = 4.107 \) for G73 and \( M = 1.407 \) for G74. The shape of diffuser inlet considerably affects static and total temperature distribution. Heat load is the smallest for a surface with G74 geometry. Stagnation temperature near the surface of nozzle throat is \( T_0 \approx 490 \text{ K} \) for G72 geometry, \( T_0 \approx 510 \text{ K} \) for G73 geometry and \( T_0 \approx 440 \text{ K} \) for G74 geometry.

![Figure 3. Diffuser inlet geometry.](image)

![Figure 4. Mach number.](image)
Figure 5. Heat flux density at diffuser inlet surface.

The maximum value of heat flux density at diffuser inlet surface is $q_{w_{\text{max}}} = 10.83 \text{ W/cm}^2$ for G72 geometry, $q_{w_{\text{max}}} = 21.58 \text{ W/cm}^2$ for G73 and $q_{w_{\text{max}}} = 5.73 \text{ W/cm}^2$ for G74. The average values are 3.03 W/cm$^2$ for G74, 4.09 W/cm$^2$ for G72 and 4.13 W/cm$^2$ for G73. Thus, the optimal G74 geometry allows to reduce average heat load on the surface by 35% and to achieve 3.8 times smaller maximum thermal load.

Similar calculations were made for a nozzle with $M=5$.

4. Results of three-dimensional numerical flow and heat transfer simulation in a supersonic wind tunnel

In the present work, final results of three-dimensional calculations are presented for a supersonic wind tunnel with a rectangular nozzle for $M=5$, due to the problem of heat flux minimization at diffuser inlet walls being more relevant at this Mach number. Similar calculations were made for a nozzle with $M=4$ (see table 1).

The aim of numerical investigation was to obtain flow field (and distribution of its main parameters, including heat flux at the walls) in order to estimate flow quality in the new test section (Eiffel chamber) and diffuser efficiency in decelerating the stream generated by a new nozzle.

The wind tunnel geometry used in calculations consists of three parts (figure 6): 1 is a flat supersonic nozzle (with $M = 5$ at nozzle exit); 2 is an Eiffel test section; 3 is a supersonic diffuser.

Figure 6. Three-dimensional wind tunnel geometry used in calculations.
At prechamber inlet section, a total pressure of $p_0=3039750$ Pa and a total temperature of $T_0=425$ K were imposed. Static pressure at diffuser outlet was $p_{st}=50000$ Pa. A constant temperature of $T_w=300$ K was imposed at all walls.

The initial mesh consisted of 29 245 205 cells. The results presented below were obtained using a mesh of 40 230 783 cells. A grid convergence check showed that the average Mach number value at a distance of $\Delta x=100$ mm from the nozzle exit section inside the characteristic rhomb in the test section (the suggested location of the model) is $M=4.9528$ for the first mesh and $M=4.9494$ for the second one. The divergence of calculated Mach numbers in similar points did not exceed 0.04%. For both static and total pressure the difference was less than 0.1%.

Figure 7 shows that the Mach number at nozzle exit has a uniform distribution and attains a value of $M=4.99$. The stream leaving the test section decelerates at the oblique shocks that emerge as a consequence of interaction between the central stream and a secondary flow in the periphery of the Eiffel chamber. Flow speed in the second diffuser throat (M=$\approx4$) is smaller than in the test section. A rise of static pressure is observed near diffuser inlet opening zone where static pressure attains a value of $p_{st}=0.1$ atm.

Figure 8 shows Mach number distribution in a cross-section at a distance of 100 mm from the nozzle exit (the suggested location of the model). One can see that the flow core is at least 136 mm in size in both directions. The deviation of Mach number from its average value does not exceed 0.3%. Thus, the model should be placed closer to the nozzle exit, at a distance of $\Delta x=100$ mm, rather than in test section center.
Figure 8. Mach number distribution in a cross-section at a distance of $x=100$ mm from the nozzle exit.

Calculated static temperature distribution is shown in figure 9. With the given conditions in the prechamber ($T_0=425$ K, $T_w=300$ K), static temperature in the flow core in the supersonic part of the nozzle and in the test section is about 71 K, and does not exceed 380 K near diffuser walls, which means that dedicated cooling of the walls is not required.

Figure 9. Static temperature, K.

5. Conclusions
Based on the results of numerical investigation of air flow in a supersonic wind tunnel, the following conclusions and recommendations were made.

Flow non-uniformity in the test section is mainly caused by sharp intersection of flat nozzle contour with test section front panel and by collision of reverse flow in test section periphery with the stream flowing out of the nozzle. Therefore this non-uniformity can only be substantially reduced by moving the nozzle into the test section. It is recommended to move the nozzle into the test section by 40 mm.
Perturbations emerging from the nozzle exit and spreading down the test section behind the boundaries of characteristic rhomb’s reverse wedge are nearly impossible to eliminate (even in a closed test section). Therefore the model’s center of rotation and optical window axis should be placed as close to the center of the characteristic rhomb as possible. It is recommended that the model’s center of rotation (and optical window axis) are placed at a distance of 100 mm from the nozzle exit. The suggested diameter of the window is 150-180 mm.

Numerical investigation of three-dimensional air flow in the final geometry configuration of the supersonic wind tunnel showed that flow core size in the model’s location is about 130-150 mm in both lateral dimensions, and that Mach number fluctuations in the core do not exceed 0.5%.

The obtained results became part of scientific and technical basis of supersonic wind tunnel design process and were applied to a generalized class of similar wind tunnels.

References
[1] Katskova A.N., Naumova I.N., Shmiglevsky Y.D., Shulishnina N.P. *An experience of calculation of plane and axysymmetric supersonic flows by the method of characteristics.* Precedence of SU AS Calculation Center, 1961.
[2] Verhovsky V.P. Calculation technique of wind tunnel shaped nozzles. Paper in the book “TsAGI – main milestones of scientific work 1993 - 2003”, Mos. Fizmatlit, 2003.
[3] Verhovsky V.P. Nosov B.B. Flow field investigation in the axysymmetric shaped nozzle at Mach numbers M=7 ÷ 10. Scientific papers of TsAGI, , 2004, т.XXXV, № 1-2.