Computational studies to determine the design of prototype heater for a hypersonic wind tunnel

A S Rtishcheva

Central Aerohydrodynamic Institute named after Professor N. E. Zhukovsky (TsAGI) Russia, 140180, Zhukovsky, 1 Zhykovsky Street

Abstract. During the modernization of the TsAGI hypersonic wind tunnel, which implies an increase in the limiting values of the flow stagnation parameters, and in the thermal load, consequently, computational studies were realized for the existing design of the electric arc heater and new geometry options for the central electrode and the cooling duct of the external electrode and nozzle. The distribution of the heat flux density to the surfaces of the main elements of the wind tunnel was obtained based on the calculations for the air duct of wind tunnel. The qualitative data of the flow formation both in the pre-heater and the pre-chamber served as the basis for changing the geometry of the central electrode. In this study, a numerical modelling approach was implemented for a through calculation of the flow and heat transfer in all areas of the wind tunnel circuit (heater, pre-chamber, nozzle, test section, diffuser). The ANSYS FLUENT software package was used to solve the axisymmetric Navier-Stokes equations for a five-component chemically reacting gas mixture: O₂; N₂; O; N; NO using the Spalart-Allmaras turbulence model. To study the hydrodynamics and thermal state of the structure of the external and central electrodes, the possibility of the ANSYS FLUENT complex for solving conjugate problems was used (the complete Navier-Stokes equations and the energy balance equation were solved in a fluid, and the heat conduction equation was solved in a solid). In this case, at the interface between the media, the conditions for the continuity of temperature and heat flux were satisfied.

1. Introduction

Improvement of hypersonic aircraft depends on capabilities of experimental facilities. TsAGI has a hypersonic wind tunnel designed to conduct research on heat resistance of thermal protection for rockets and spaceships; the flow parameters of its test section are as follows: Mach number 5<\textit{M}<8, stagnation temperature \( p_0 \leq 10 \text{ atm.} \), stagnation pressure \( T_0 \approx 1500-4600 \text{ K} \), and electric power of electric arc heater \( N_{el} \leq 0.8 \text{ MW} \). Expanding the range of test parameters to \( p_0 = 12 \text{ atm.} \) and \( T_0 = 5000 \text{ K} \) requires an increase in the electric power of the heater to \( N_{el} \approx 1.2 \text{ MW} \). In addition, computational studies are needed for the existing design electric arc heater and its possible modifications, which would provide more efficient cooling of electrode surfaces.

It should be noted that dissociation reaction of \( \text{O}_2/\text{N}_2 \) with NO formation begins at temperatures \( T_0 = 5000 \text{ K} \) in the heater and in the pre-chamber of the nozzle. A part of the thermal energy is spent on chemical reactions, which must be taken into account when designing prototypes of heater elements.

This paper presents the results of calculations for the flow and heat transfer in the wind tunnel circuit (including the existing heater), obtained using the ANSYS FLUENT software package (license No. 501024).

Thus, the axisymmetric Navier-Stokes equations for a five-component mixture of a chemically reacting gas were solved using the Spalart-Allmaras turbulence model under conditions of volumetric...
heat supply to the electric arc zone and attaining the maximum test parameters in the pre-chamber: $p_0=12$ atm; $T_0=5000$ K.

Based on the obtained distributions of the heat flux density to the surfaces of the electrodes and other elements of the wind tunnel, the central and external electrodes of the heater together with the cooling duct were investigated. The calculations used the ANSYS FLUENT for solving conjugate problems (the full Navier-Stokes equations and the energy balance equation in a fluid, and the heat conduction equation in a solid).

The obtained results suggest new modifications of the geometry of the central electrode and significant improvement of the cooling duct of the external electrode and the nozzle.

2. Calculation for the flow and heat transfer in the hypersonic wind tunnel circuit

Figure 1 shows the computational grid for modeling the flow in the air duct of hypersonic wind tunnel with the existing geometry of the inner electrode.

The study implements a debugged and verified approach to modelling the flow and heat transfer in a wind tunnel, according to which, in all areas (heater, pre-chamber, nozzle, test section, diffuser), the ANSYS FLUENT software package solves axisymmetric Navier-Stokes equations for viscous and heat-conducting gas using the Spalart-Allmaras turbulence model [1-3].

A fundamental innovation in the calculations was the simulation of the flow for a chemically reacting gas mixture: $\text{O}_2; \text{N}_2; \text{O}; \text{N}; \text{NO}$.

The rate constants of chemical reactions were determined with the Arrhenius formula [4]:

$$k_{f(r)} = A_{f(r)} T^{B_{f(r)}} \exp(-C_{f(r)}/T),$$

where subscripts f and r correspond to forward and backward reactions.

The kinetic model and the values of the reaction rate constants are shown in Table 1.

The dependences of gas-dynamic and thermophysical parameters on temperature for each component were specified in accordance with the reference data [5].

The calculation was made in a two-dimensional axisymmetric formulation under conditions of a volumetric supply of thermal power to the electric arc zone upon reaching the test limits in the pre-chamber: $p_0=12$ atm; $T_0=5000$ K.

In addition, at the entrance to the computational domain, the mass fractions of oxygen and nitrogen were specified: $g(\text{O}_2)=0.23$; $g(\text{N}_2)=0.77$.

The walls of the air duct were modelled as catalytic ones with $T_\text{w}=300$ K.

The boundary conditions at the edge of computational domain (pressure level $p_{\text{out}}$) were selected taking into account the ejectors capacity and the implementation of a steady flow regime in the test section and diffuser.
Table 1. Kinetic model.

| Number | Chemical reaction | $A_f$ \([\text{cm}^3/\text{mol}]^{-1}\) | $B_f$ | $C_f$, K | $A_r$ \([\text{cm}^3/\text{mol}]^{-1}\) | $B_r$ | $C_r$, K |
|--------|-------------------|-----------------|------|--------|-----------------|------|--------|
| 1      | $N_2 + M \leftrightarrow N + N + M$ \((M = N; O; NO; N_2; O_2)\) | 7.000E+21 | -1.60 | 113200 | 3.000E+13 | -0.60 | 0      |
| 2      | $O_2 + M \leftrightarrow O + O + M$ \((M = N; O; NO; N_2; O_2)\) | 2.000E+21 | -1.50 | 59750  | 1.000E+13 | -0.50 | 0      |
| 3      | $NO + M \leftrightarrow N + O + M$ \((M = N; O; NO; N_2; O_2)\) | 5.000E+15 | 0     | 75500  | 8.500E+07 | 1.00  | 0      |
| 4      | $NO + O \leftrightarrow N + O_2$ | 3.200E+09 | 1.00  | 20000  | 1.300E+10 | 1.00  | 4000   |
| 5      | $N_2 + O \leftrightarrow NO + N$ | 2.000E+12 | 0.50  | 38000  | 4.400E+11 | 0.50  | 0      |

According to the results of checking the grid convergence, to obtain a qualitative resolution of the viscous sublayer of a turbulent boundary layer in the near-wall region, and an acceptable resolution of shock waves and flow separation zones, a computational grid containing ~0.8 million cells is sufficient for an axisymmetric problem. A grid with so many cells was used as the base. However, to obtain the best quality results, all the final calculations were performed on a more detailed grid containing ~1.5 million cells.

Figures 2–7 show the data of calculations for the flow and heat transfer in the air duct of hypersonic wind tunnel.

![Figure 2. Streamlines in the heater and in the nozzle pre-chamber.](image)

![Figure 3. Mach number field.](image)

![Figure 4. Static temperature field $T$, K.](image)
Figure 5. Field of mass fractions for gas mixture components.
Figure 6. Heat flux density to the walls of the external electrode, pre-chamber and nozzle.

Figure 7. Heat flux density to the walls of the central electrode.

In the heater (near the front wall of the inner electrode), the flow separation from the corner point of the central electrode and the formation of recirculation zones could be observed. Intensive chemical reactions of $\text{N}_2$ and $\text{O}_2$ and formation of $\text{N}$, $\text{O}$, $\text{NO}$ took place. In the supersonic part of the nozzle, the test section and the diffuser, the decomposition of NO and the atom recombination of $\text{N}$, $\text{O}$ began. Full recombination occurred at the walls, since they were catalytic and had a relatively low temperature.

The static temperature field in the pre-heater, the pre-chamber and the nozzle was non-uniform. Areas with high temperatures were concentrated on the periphery of the main stream.

An increase in the density of the heat flux to the walls of the air duct was observed in the area of the electric arc, the pre-chamber and the throat of the nozzle. The maximum value of the heat flux density in the region of the nozzle throat was $q_{\text{w,max}}=1641.8 \text{ W/cm}^2$. The heat flux density to the surface of the central electrode reached $q_{\text{w}}=350 \text{ W/cm}^2$ at the corner points.

Thermal power, required for the regime $M=7$; $p_0=12 \text{ atm}$; $T_0=5000 \text{ K}$, was $Q=0.658 \text{ MW}$.

Taking into account the fact that the coil accounted for $N_{\text{coil}}=0.1-0.15 \text{ MW}$ of the total electrical power, the maximum surface heat power released in the electric arc zone could be up to $Q_w\approx0.196 \text{ MW}$ (the value obtained indirectly without taking into account electrical and heat losses). However, according to the qualitative analysis of the temperature field of the existing central electrode the surface thermal power did not exceed the value of $Q_w\approx0.1 \text{ MW}$. 

a) Heat flux to a horizontal surface

b) Heat flux to a vertical surface
3. Calculation of the cooling system for the external electrode and nozzle

The purpose of calculating the cooling system of the external electrode and nozzle was to determine the most heat-loaded structural elements and analyze its efficiency.

The calculation was made in a two-dimensional axisymmetric formulation. In this case, an approach was used for the conjugate solution of the problems of fluid hydrodynamics and thermal conductivity in a solid body. At the interface between the media, the conditions for the continuity of temperature and heat flux were satisfied.

Figure 8 shows the geometry and a fragment of the computational grid in the nozzle throat region. The final calculations were performed on a grid containing 4.1 million cells.

The dependence of the thermophysical properties of copper, stainless steel and water on temperature was set in accordance with the reference data [6].

On the inner surface (wall of the air duct), the distribution of the heat flux density $q_w$, obtained as a result of solving the gas-dynamic problem, was specified. The surface heat power released in the electric arc zone was also set.

On the outer surfaces, the heat transfer coefficient was $\alpha=10$ W/cm², which is characteristic for natural convection, and the air temperature in a distance from the wall was $T=300$ K=27°C. The inlet water temperature was $T_in=288$ and K=15 °C. The water rate in the cooling system was $G_{water}=8$ kg/s.

Figures 9-10 show the main calculation data.

The maximum temperature of a solid structure corresponds to the electric arc zone: $T_{max}=830.5$ and K=557.5 °C.

Boiling of water was observed in stagnant zones near the divergent section of the nozzle and the throat of the nozzle, as well as the near-wall region of the electric arc zone. Avoiding boiling in stagnant zones was possible by changing the geometry of the cooling path according to Figure 8 (it was necessary to reduce the flow area for water).

In accordance with calculations, no boiling was possible for the existing structure. This would require increasing the water consumption to $G_{water}=30$ kg/s.

As the most promising option, it is recommended to increase the water consumption to $G_{water}=12-16$ kg/s provided that there are no stagnant zones in the cooling duct of the nozzle.
The purpose of calculations for the design of the central electrode was to study its thermal state and cooling efficiency.

The calculation was made in a two-dimensional axisymmetric formulation. The approach was also used for the conjugate solution of hydrodynamics problems and thermal conductivity in a solid body.

Figure 11 shows the geometry and a fragment of the computational grid of the central electrode (3.1 million cells).

On the inner surface (wall of the air duct), the distribution of the heat flux density $q_w$, obtained as a result of solving the gas-dynamic problem, was specified. The surface heat power released in the electric arc zone was also set. The inlet water temperature was $T_{in}=288$ K=15 °C. The water rate in the cooling system was $G_{water}=2.8$ kg/s.

Figures 12-13 show the main calculation data.

The streamline pattern demonstrated the formation of a stagnant zone at the corner point, which is why avoiding boiling was impossible even with a significant increase in water consumption. In this case, the electrode would melt during the wind tunnel run, which makes its reuse impossible. These disadvantages can only be eliminated by changing the geometry.
a) Air duct of the external electrode cooling system.

b) Fragment of computational grid.

Figure 11. Geometry and a fragment of the computational grid.

Figure 12. Streamlines.

Figure 13. Temperature field, °C.
Based on the analysis of the obtained data, two options for the geometry of the central electrode were proposed with rounding of the corner points, both from the side of the air duct and from the side of the water-cooling duct (Figure 14).

![Figure 14. New geometry options for the center electrode.](image)

Rounding of corners is necessary for a better gas flow around the central electrode and for a more efficient cooling.

5. Calculation for new design options of the central electrode

To investigate new design options for the central electrode, additional studies of the air duct were realized with the purpose to obtain the distribution of the heat flux density to the walls (including the central electrode). In this case, the smallest heat loads were seen for the variant with full wall rounding (the heat flux density to the surface did not exceed $q_w=250 \text{ W/cm}^2$). This geometry provided the smoothest flow around the electrode, although the formation of recirculation zones near the front wall was observed in all the considered cases.

The rounding of the inside corners also contributed to an increase in cooling efficiency. However, at the existing water flow rate in the cooling system $G_{\text{water}}=2.8 \text{ kg/s}$, the central electrode would also start melting during the wind tunnel run (Figure 15). Boiling would occur in the near-wall region (electric arc zone). For the option of the electrode with full rounding of the inner and outer surfaces, damage would be minimal, and with 1.5-2 times increase in water consumption, water boiling and melting were almost completely excluded.

![Figure 15. Temperature field of new central electrode, °C](image)

Thus, the performed calculations confirmed the efficiency of rounding the front surface of the central electrode. The most expedient was the option with full rounding of the inner and outer surfaces, which ensured a better flow around the electrode both from the gas flow side and from the cooler side.
Conclusions
In this paper, the design studies for the external and central electrodes of the electric arc heater in a hypersonic wind tunnel were carried out on the basis of full-scale studies of the flow and heat transfer in the air duct, taking into account the physicochemical processes occurring in the gas.

It has been found that the thermal power required to implement the maximum test parameters is significantly lower than the electrical one and is $N_{\text{heat}} = 0.658 \text{ MW} \ (N_{\text{el}} \approx 1.2 \text{ MW})$. For the existing design of the heater, the most heat-loaded are the corner points of the central electrode. Therefore, alternative designs are proposed with a slight and full rounding of the front surface of the central electrode.

Full rounding of the front surface facilitates a better flow around it, both from the gas flow side and from the water flow side, providing more efficient cooling of the entire structure. With the existing water flow rate $G_{\text{water}} \approx 2.8 \text{ kg/s}$ this geometry option will be least of all subjected to destruction during the run of the wind tunnel, which makes its reuse possible.

At water flow rate $G_{\text{water}} \approx 8 \text{ kg/s}$ for cooling the external electrode and the nozzle, the maximum temperature of the solid structure corresponds to the electric arc zone – $T_{\text{max}} = 557.5 \degree \text{C}$. The study of the cooling duct shows that near the edge of the divergent section of the nozzle and the throat of the nozzle, stagnant zones and recirculation zones appear where water boiling occurs. For more effective surface cooling, the flow area for water should be changed (decreased) in the indicated areas.

It has been also found that water boiling (both for the design of the external and internal electrodes) may be avoided with the help of an almost twofold increase in water consumption.

The obtained results of the study form the scientific and technical basis for the design of a prototype of an electric arc heater for an operating hypersonic wind tunnel.

References
[1] Drozdov S M, Davletkildeev R A and Rtischeva A S 2019 TsAGI Scientific Notes vol L 2 24–34
[2] Drozdov S M and Rtishcheva A S 2017 Journal of Physics: Conference Series 891 doi:10.1088/1742-6596/891/1/012043
[3] Drozdov S M, Davletkildeev R A and Rtischeva A S 2018 Publ. of HiSST: International Conference on High-Speed Vehicle Science Technology (Moscow)
[4] Roop N Gupta, Yos J, Thompson R and Kam-Pui Lee 1990 NASA RP-1232
[5] McBride B, Gordon S and Reno M 1993 NASA Technical Memorandum 4513
[6] Chirkin V S 1959 State publishing house of physical and mathematical literature (Moscow) [in Russian]