Combining methods for end-to-end calculation of gas flow in a Laval nozzle

R E Dushkov and R I Dzerzhinsky
MIREA - Russian Technological University, Prospekt Vernadskogo, 78, Moscow, 119454, Russia

E-mail: roman.dushkov@gmail.com

Abstract. The paper solves the problem of selecting optimal methods and combining them when performing a complex calculation of all sections of the Laval nozzle. The problem is significant, since different flow modes are realized in different areas of the nozzle and it is impossible to use a single method for the full calculation. Solving the direct problem, parameters in the flow of gas are determined. Solving the reverse problem determine the profile of the Laval nozzle. Profiling is an important applied task for calculating the nozzles of rocket and aircraft engines, or designing experimental turbines. In this paper, we consider a two-dimensional plane axisymmetric flow of an inviscid perfect gas (on the example of dry air). The problem is solved starting with an accelerating subsonic flow via using methods of establishing and finite elements. For subsonic flow found an exact solution (Prandtl-Mayer flow). Solving the problem in a supersonic flow via using proximate methods (the method of characteristics and interpolation for the grid normalization). The solution is supplemented by a method that allows taking into account the features of axisymmetric flow in the nozzle to expand the applicability of the solution for real nozzles. The results obtained – the gas velocity field and the coordinates of nodes. Results allows estimation of the optimal size of the Laval nozzle for the given conditions, its shape, and obtain a flow profile depending on its initial characteristics defined in the previous section of the nozzle. The results of flow simulation can be applied to calculate the flow in circular sections, which are the most common in the current application. The results are confirmed by the solution using the finite element method, and the convergence of the solution in the diffuser is described in the current work. Theoretically, the solution can be applied to a mixture of gases, if peculiarities of physical and chemical transformations and the participation of combustion products are taken into consideration. In this case instead of parameters in the pre-chamber, the parameters of the combustion chamber and the products in it should be used as initial conditions. This calculation will be carried out in future studies of the authors.

1. Introduction

Laval nozzle is a technical device that serves to accelerate the gas flow passing through it to speeds exceeding the sound speed. The converging part of the nozzle is called a confuser, and the expanding part is called a diffuser (see figure 1).

Due to the pressure difference between the prechamber and the environment, the gas begins to move along the longitudinal axis in various modes described below. As the gas moves along the nozzle, its absolute temperature and pressure decrease, and its speed increases. The gas internal energy is converted into its directed movement kinetic energy.
Such structures use is not limited to devices for conducting aerodynamic experiments at supersonic and hypersonic speeds: the Laval nozzle is rocket engines important part.

Two problems in gas dynamics are solved for the flow in the Laval nozzle.

1.1. Direct task
The direct task is to study the gas parameters in the Laval nozzle sections with its known shape. It can be solved by a methods' variety used in commercial programs and described in various works.

1.1.1. Finite element method. Allows to calculate the direct problem in finished software products (FlowVison, ANSYS, SolidWorks, etc.) and get a high accuracy approximate solution.

1.1.2. Series expansion method. Allows obtaining a solution for the nozzle transonic part under the boundary conditions specified in the analytical functions' form. Using this method, it was possible to carry out the flow approximate calculation for the case when the contour is given by the third-degree polynomial, the sonic line calculation, and the characteristics concerning the sonic line on the axis.

1.1.3. Mooreman-Cole Method. Uses a difference approximation scheme and solves a mixed-type second-order equation for the velocity coefficient in an orthogonal coordinate system associated with streamlines (problem solution for a flat nozzle).

1.1.4. Establishment method. Uses an implicit difference scheme that allows comparing the result with a known solution to the inverse problem.

1.2. Inverse problem
The inverse problem is to determine the nozzle shape with the gas known parameters flowing in it and the environment parameters. The modern rocket engines efficient nozzles are profiled on the special gas-dynamic calculations basis. The basic equation for calculating the geometry (1) relates the cross-sectional area gradient, the velocity gradient and the Mach number:

\[
(M^2 - 1) \frac{\partial V}{\partial x} = \frac{V}{S} \frac{\partial S}{\partial x},
\]

where:
M - Mach number;
S - nozzle section area;
V - gas velocity; 
\( x \) is the longitudinal coordinate.

1.2.1. There is no single method for solving the inverse problem. Since gas flow different regimes can be realized in the Laval nozzle, which is characterized by different physical models. Therefore, to carry out the end-to-end calculation, it is necessary to determine methods set for calculating in nozzle individual characteristic zones, the conditions for transition between zones (boundary conditions), and then “stitch” the solutions on their boundaries.

2. Model limitations
We have considered an inviscid perfect gas two-dimensional plane axisymmetric flow.

The Laval nozzles used in rocketry and wind tunnels have different shapes: sections can be round, elliptical, and square. In this case, in the general case, a three-dimensional flow can be considered, which, however, causes difficulties. It is possible to restrict to a two-dimensional plane flow, which will simplify the mathematical calculations, but it is necessary to take into account the axisymmetric flow.

Taking into account the viscosity allows taking into account the features in the boundary layer, namely, the gas deceleration at the nozzle walls. In this case, a negative velocity gradient near the walls causes a special shape shock wave appearance, which causes a separated flow [1]. When solving the direct problem, taking into account such phenomena can play an important role, since the nozzle walls in its supersonic part can introduce additional disturbances. When solving the inverse problem, you can set the final condition for the velocity and not consider the viscous effects.

A continuous medium can be described by different models: liquid (gas), which has fluidity, viscosity and compressibility. A model in which gas compressibility under conditions close to normal conditions is significantly affected by the interaction forces between molecules is called a real gas and is difficult to calculate within the even particular problems’ framework. The perfect gas model describes a gas that has heat capacity constants \( C_p, C_V \) and satisfies the Clapeyron state equation (2).

\[
p = \rho R T ,
\]

where:

- \( p \) - pressure
- \( \rho \) - density
- \( T \) - temperature
- \( R \) - universal gas constant

In this model, you can calculate not only the kinematic parameters, but also the state parameters, and later even a several gases mixture flow physicochemical transformations characteristic [2] and [3].

3. Flow modes
Before choosing a method suitable for calculating gas parameters, or the corresponding profile, it is necessary to determine a physical model that describes the flow.

The equation analysis (1) allows determining three flow regimes: subsonic, transonic, or supersonic flows. Which one is implemented in practice depends on the pressure difference between the nozzle inlet and the environment.

3.1. \( M < 1 \) - subsonic flow
If \( \frac{\partial s}{\partial x} < 0 \), then \( \frac{\partial V}{\partial x} > 0 \), the subsonic flow in the converging channel is accelerated. And vice versa, if \( \frac{\partial s}{\partial x} > 0 \) \( \left( \frac{\partial V}{\partial x} < 0 \right) \) - the subsonic flow in the expanding channel is decelerated.

3.2. \( M = 1 \) - sonic flow
If $\frac{\partial S}{\partial x} = 0$, then the flow in the Laval nozzle minimum (critical) section passes through the sonic velocity, and if $\frac{\partial v}{\partial x} = 0$, a velocity extremum is observed: if the pressure reached in the critical section exceeds the external pressure, then the flow at the exit from the nozzle will be supersonic. Otherwise, it remains subsonic.

3.3. $M>1$ - supersonic flow

If $\frac{\partial S}{\partial x} < 0$ then $\frac{\partial v}{\partial x} > 0$, the supersonic flow in the converging channel is decelerated, and if $\frac{\partial S}{\partial x} > 0$ ($\frac{\partial v}{\partial x} < 0$) - the supersonic flow in the expanding channel is accelerated [4].

The flow regimes are determined by the kinematic parameters, which are described by the gas dynamics basic kinematic equation (3).

$$(V_x^2 - a^2) \frac{\partial v_x}{\partial x} + (V_y^2 - a^2) \frac{\partial v_y}{\partial y} + V_x V_y \left( \frac{\partial v_x}{\partial x} + \frac{\partial v_y}{\partial y} \right) = 0,$$

(3)

where:

- $a$ is the sound velocity, $M = \frac{V}{a}$, which can also be found from the state equation (4):

$$a^2 = kRT,$$

(4)

where:

- $k = \frac{C_p}{C_v}$ adiabatic indicator

Let be a potential function, then we get expression (5).

$$d\varphi = V_x dx + V_y dy = \frac{\partial \varphi}{\partial x} dx + \frac{\partial \varphi}{\partial y} dy.$$  

(5)

Then equation (3) can be represented as expression (6).

$$(V_x^2 - a^2) \frac{\partial^2 \varphi}{\partial x^2} + (V_y^2 - a^2) \frac{\partial^2 \varphi}{\partial y^2} + 2 \cdot V_x V_y \frac{\partial^2 \varphi}{\partial x \partial y} = 0.$$  

(6)

4. Simulation results

As described in pp. 3.1-3.3, 3 flow regimes are realized in the Laval nozzle.

Gas acceleration in the nozzle is accompanied by a decrease in static pressure. Therefore, the gas pressure in the nozzle front should be higher than the environment pressure into which it flows out.

In the confuser, gas motion is realized, in which, in the entire considered region, the medium $V$ particles motion velocity is less than a sound propagation local speed (in the same medium). When the particle velocity is much less than the sound velocity (up to 100 m/s), then when describing the flow, the medium compressibility can be neglected, that is, a change in its density, for example, can be considered incompressible. Also, one of the fundamental problems is the flows' study without taking into account resistances in the heat transfer absence. Such flows are called adiabatic.

Transonic gas flows in Laval nozzles with logarithmic singularities at the limiting characteristics have been investigated in various works, for example, in [5]. However, if this zone is calculated separately, given only the critical section dimensions and the initial gas parameters, an exact solution can be obtained (see section 4.2).

The supersonic flow in the diffuser can be conditionally divided into several zones since at different velocities one should take into account mesh construction different features [6] and changes in physical properties [The previously unmixed components homogeneous combustion processes study]. More details in section 4.3.
4.1. Confuser calculation

Gas parameters at rest are energy indicators and do not change with time – pressure $p$, density $\rho$ and temperature $T$ are related to the corresponding braking parameters through gas-dynamic functions (7) as follows:

$$\begin{align*}
\frac{p}{p_0} &= \left(1 - \frac{k-1}{k+1} \lambda^2 \right) \Lambda \left(\frac{k}{k-1}\right), \\
\frac{\rho}{\rho_0} &= \left(1 - \frac{k-1}{k+1} \lambda^2 \right) \Lambda \left(\frac{k}{k-1}\right), \\
\frac{T}{T_0} &= \left(1 - \frac{k-1}{k+1} \lambda^2 \right)
\end{align*}$$

(7)

The Laplace equation is a partial differential equation. In three-dimensional space, the Laplace equation is written as equation (8) and is a Helmholtz equation special case:

$$\nabla^2 V = \frac{\partial^2 V}{\partial x^2} + \frac{\partial^2 V}{\partial y^2} + \frac{\partial^2 V}{\partial z^2} = 0. \tag{8}$$

The equation is also considered in two-dimensional and one-dimensional space. In two-dimensional space, the equation for the potential gas flow using equation (8) can solve the Dirichlet problem - determine the velocity field if the Stokes equation is considered and expression (9) is obtained:

$$\frac{\partial^2 \psi}{\partial x^2} + \frac{\partial^2 \psi}{\partial y^2} = \varphi(x, y). \tag{9}$$

On a plane in the form of second-order difference formulas on a given size grid with a constant step, the grid equation is solved by the cross method. The potential value at each point is found by the difference formula (10) [7].

$$\varphi(i, j) = \frac{\varphi(i+1, j) + \varphi(i-1, j) + \varphi(i, j+1) + \varphi(i, j-1)}{4}. \tag{10}$$

We determined convergence in itself of the problem to be solved for potential values at the computational domain boundaries: at the left, right, upper borders $\varphi = 1$, at the lower border $\varphi = 0$.

As can be seen from figure 2, with an increase in the nodes number the physical picture remains, it is possible to calculate the flow potential and determine the velocities according to equation (5). The gas flow is accelerating. Such a calculation makes it possible to determine the kinetic parameters and find the value at each new point and proceed to solve the problem in transonic flow (in the nozzle critical section area).

A detailed solution can also be carried out for a gases' mixture, taking into account physical and chemical transformations, which results are given in [8].

4.2. The nozzle critical section calculation

The nozzle geometry, defined in p. 4.1 and presented in Figure 1, determine the conditions for solving the problem in the transonic flows field.

We calculate the output parameters according to the system equations (7) and determine the gas-dynamic functions' values for pressure, density and temperature, respectively (11):

$$\begin{align*}
\frac{p}{p_0} &= \pi \approx 0.0142, \\
\frac{\rho}{\rho_0} &= \varepsilon \approx 0.0288, \\
\frac{T}{T_0} &= \tau \approx 0.4919
\end{align*}$$

(11)
Figure 2. The Dirichlet problem solution for the Laplace equation in a two-dimensional gas flow.

Let us calculate the given velocity value $\lambda \approx 2.3641$ according to the formula (12):

$$\lambda(\pi) = \sqrt{1 - \left(\frac{k+1}{k-1}\right)^k \pi^{k-1}}.$$  

The nozzle size is also influenced by the medium into which the outflow occurs - into a void or stationary/moving air. For an exact definition, you can set the thrust using the expression (13):

$$F_x = \varepsilon(\lambda) \cdot p_0 \cdot z(\lambda) \cdot S^* - p_H S_{crit},$$

where:

- $z(\lambda) = \lambda + \frac{1}{\lambda}$ - gas dynamic function;
- $S^*$ is the last section area, determined from the equation solution (9);
- $S_{crit}$ is the critical section area (the required value).

Then it is possible to determine the critical section area - $S^* \approx 0.0170m^2$ and, accordingly, its diameter - $d^* \approx \sqrt{\frac{4S^*}{\pi}} \approx 0.1472m$.

Further, for a certain size, an exact problem was solved, describing the flow turn through the rarefaction waves fan until the moment it reaches a speed exceeding the sound velocity by a given value, or when turning at a given angle. Let's designate the angle $\phi$ and get an expression for determining the velocity (14):

$$\lambda = 1 + \frac{2}{k-1} \left(\phi \sqrt[2]{\frac{k-1}{k+1}}\right)^2,$$
To calculate the Mach number, we use the nonlinear dependence (15):

$$\sqrt{\frac{M^2 - 1}{\lambda}} = \sqrt{\frac{\lambda^2 - 1}{k - 1}} \frac{1}{k + 1}.$$  \hspace{1cm} (15)

Then the new value for the rotation angle can be obtained according to the geometric expression (16), which describes the discharge waves lines position:

$$\phi_{i+1} = \phi + \arcsin\left(\frac{1}{M}\right) - \frac{\pi}{2}.$$  \hspace{1cm} (16)

Let’s set the conditions for the end of the calculation: the Mach number $M = 1.1$ and the u-turn angle $\phi = 45$ degrees. Received the exact value for the velocity projections on the rarefaction waves lines, the results are shown in table 1.

**Table 1.** The velocity exact value determination during the transonic flow turn in the nozzle critical section area.

| $V_y$, m/s | $V_x$, m/s | $V$, m/s | $M$, 1 |
|------------|------------|----------|--------|
| -7.5583    | 325.0321   | 325.12   | 1.016  |
| -7.8534    | 325.6653   | 325.76   | 1.018  |
| -8.7501    | 326.6028   | 326.72   | 1.021  |
| -8.8675    | 327.2399   | 327.36   | 1.023  |
| -9.5457    | 328.1812   | 328.32   | 1.026  |
| -10.1223   | 329.1244   | 329.28   | 1.029  |
| -11.1689   | 330.3713   | 330.56   | 1.033  |
| -11.5611   | 331.3184   | 331.52   | 1.036  |
| -12.4016   | 332.5688   | 332.80   | 1.040  |
| -13.1478   | 333.8212   | 334.08   | 1.044  |
| -13.8111   | 335.0755   | 335.36   | 1.048  |
| -14.4003   | 336.3319   | 336.64   | 1.052  |
| -15.3687   | 337.8907   | 338.24   | 1.057  |
| -16.2505   | 339.4512   | 339.84   | 1.062  |
| -17.0545   | 341.0138   | 341.44   | 1.067  |
| -17.7879   | 342.5785   | 343.04   | 1.072  |
| -18.8548   | 344.4443   | 344.96   | 1.078  |
| -19.8430   | 346.3120   | 346.88   | 1.084  |
| -20.7594   | 348.1817   | 348.80   | 1.090  |
| -21.9800   | 350.3512   | 351.04   | 1.097  |
| -22.7607   | 352.2254   | 352.96   | 1.103  |

The value $M = 1.103$ defines the condition for the calculation end when the flow turn angle is 17 degrees. The obtained velocity values can be used as initial conditions for calculating the diffuser.

Figure 3 shows an obtained rarefaction wave lines geometric representation in the physical plane: a dashed line indicates the Laval nozzle symmetry axis, dashed lines indicate the velocity vector for five nodes of the computational grid, and solid lines indicate rarefaction waves.
4.3. Diffuser’s calculation
The equation solution (6) for supersonic velocities can be found by the characteristics' method. The essence of it is that at each space point you can draw 2 lines (characteristics of the 1st and 2nd families respectively). Their intersection will determine the new calculation node position and the calculation can be repeated layer by layer. At the same time, for an inviscid gas, it is not necessary to calculate separation flows [1].

The method peculiarity is that unlike the cases described in sections 4.1-4.2 are not known in advance to the grid nodes positions, and the interpolation used to identify new points does not provide an exact value, as was possible when calculating the rarefaction waves (section 4.2). However, the solution automatically converges on the supersonic current, which allows you to limit the convergence manifestation on the grid, unlike the case described in the current work section 4.1.

If you set new variables (17-18), the equation (6) can be presented as (19):

\[
\begin{align*}
(V_x^2 - a^2) &= A \\
V_x V_y &= B \\
(V_y^2 - a^2) &= C \\
(V_x^2 - a^2) \frac{\partial^2 \varphi}{\partial x^2} &= \varphi_{xx} \\
(V_x^2 - a^2) \frac{\partial^2 \varphi}{\partial y^2} &= \varphi_{yy} \\
\frac{\partial^2 \varphi}{\partial x \partial y} &= \varphi_{xy} \\
A \cdot \varphi_{xx} + 2B \cdot \varphi_{xy} + C\varphi_{yy} &= 0,
\end{align*}
\]

(17) (18) (19)

Then you can identify the inclination angles tangential to both families characteristics lines (20):

\[
y'_{1,2} = \frac{B \pm \sqrt{B^2 - 4AC}}{2A},
\]

(20)

where:

Because the current is supersonic, there are two solutions to the square equation.

If you imagine that the performance lines are straight (with a fairly shredded grid) you can determine the physical parameters values and velocity projections (21).
\[ V_n + \frac{k-1}{k+1} V_r = \frac{k-1}{k+1} V_{max}^2, \quad (21) \]

where:

- the maximum flow rate determined by the energy conservation equation (22).

\[ V_{max} = a_k \cdot \sqrt{\frac{k-1}{k+1}} = \sqrt{\frac{2}{k+1}} T_0 R \cdot \frac{k-1}{k+1}, \quad (22) \]

where:

- sound critical velocity (constant along with the trickle)
- gas constant

Then the velocity components can be expressed through the potential according to expression (23):

\[ \begin{align*}
V_r &= V_{max} \cdot \sin \left( \varphi \frac{k-1}{k+1} \right) \\
V_n &= V_{max} \cdot \cos \left( \varphi \frac{k-1}{k+1} \right) \cdot \frac{k-1}{k+1},
\end{align*} \quad (23) \]

the projection \( V_r \) is unchanged along, and the flow is potential, from the flow features we determine the equation components gradients (23), we obtain the expressions for the system (24):

\[ \begin{align*}
\frac{\partial V}{\partial r} &= \frac{\partial V_r}{\partial r} = 0 \\
\frac{\partial V_n}{\partial r} &= V_n = a.
\end{align*} \quad (24) \]

Using solution (20) at each point in space, it is possible to determine the tangents to the first and second families characteristics and calculate the kinematic parameters according to (22-24). The state parameters are calculated based on the gas-dynamic functions (7) and the state equation 4) [9]. The grid selected for the study and the results obtained are shown in figure 4 – dotted lines represent the velocity vectors in the calculated grid nodes, and thin lines represent the two families characteristics.

Next, it remains to find the catch coordinates on the new calculation layer and repeat the calculation until the specified velocity is reached. New nodes are obtained at the tow different families characteristics curves intersection emanating from neighbouring nodes [10].

A graphic representation is shown in figure 5.

![Figure 4](image-url)

**Figure 4.** The kinematic parameters’ calculation on the design points first layer after the vacuum fans.
Figure 5. New nodes kinematic parameters and coordinates calculation in the diffuser.

Figure 5 also shows that the grid nodes in the through calculation are condensed closer to the diffuser symmetry axis. It is also necessary to take into account the axisymmetric flow features [5]. Then we get a new picture, shown in figure 6.

Figure 6. New nodes kinematic parameters and coordinates calculation in the diffuser, taking into account the axial symmetry features.

For further calculations, it is necessary to get rid of the grid thickening. To do this, we applied linear interpolation [7] when finding the gas new coordinates and physical parameters at new points. Thus, it is possible to study the gas flow in the diffuser in the two regions AB*C and B*CD, which are illustrated in figure 7.
In a supersonic flow, the solution converges automatically [11]; it is enough to ensure the convergence along the grid. The results are shown in table 2.

As can be seen from the calculation results presented in the table, the supersonic flow is accelerated. When the speed $M = 3$ is reached, the calculation stops according to the specified conditions. To continue the calculation, it is necessary to take into account the supersonic flow additional features, take into account the boundary condition on the CE line in figure 7.

**Table 2.** The velocity exact value determination when turning the supersonic flow in the Laval nozzle diffuser.

| $V_y$, m/s | $V_x$, m/s | $V$, m/s | $M$, 1 |
|------------|------------|----------|--------|
| -17.67480175 | 886.0375899 | 886.21 | 2.769 |
| -18.36481592 | 887.7636681 | 887.95 | 2.775 |
| -20.46184693 | 890.3192517 | 890.55 | 2.783 |
| -20.73623482 | 892.0559064 | 892.30 | 2.788 |
| -22.32218809 | 894.6219611 | 894.90 | 2.797 |
| -23.67059136 | 897.1930597 | 897.51 | 2.805 |
| -26.11792160 | 900.5920583 | 900.97 | 2.816 |
| -27.03520707 | 903.1738302 | 903.58 | 2.824 |
| -29.00069784 | 906.5826842 | 907.05 | 2.835 |
| -30.74571006 | 909.9965382 | 910.52 | 2.845 |
| -32.29669598 | 913.4157827 | 913.99 | 2.856 |
| -33.67447931 | 916.8406564 | 917.46 | 2.867 |
| -35.93901220 | 921.0899536 | 921.79 | 2.881 |
| -38.00108970 | 925.3440935 | 926.12 | 2.894 |
| -39.88134881 | 929.6036374 | 930.46 | 2.908 |
| -41.59640938 | 933.8689982 | 934.79 | 2.921 |
| -44.09115943 | 938.9552525 | 939.99 | 2.937 |
| -46.40216489 | 944.0464596 | 945.19 | 2.954 |
| -48.54493948 | 949.1432856 | 950.38 | 2.970 |

**Figure 7.** Design zones in the diffuser.
5. Conclusion

To carry out a Laval nozzle geometry end-to-end calculation, shown schematically in figures 1 and 7, and when finding the gas kinematic parameters, you can use various techniques in different areas. In this article, we considered an ideal incompressible inviscid gas plane flow.

In the confuser in two-dimensional space, the equation for the potential gas flow was solved the Dirichlet problem - the velocity field was determined. A similar problem can also be solved for a gases’ mixture taking into account their physicochemical transformations (see section 4.1).

In the nozzle critical section area, a numerical solution instead, it is possible to accurately determine the velocity values by solving the problem in the Prandtl-Mayer flow rarefaction waves fans. As the initial conditions, the critical section dimensions obtained in the confuser and the gas state parameters (the deceleration parameters are unchanged) are used (see section 4.2).

In the diffuser, the velocities exact values at the given grid nodes can be used as initial conditions, and the further calculation was carried out by the characteristics’ method, which allows not only determining the kinematic parameters but also calculating the computational grid new nodes position. The axisymmetric flow features are also taken into account (see section 4.3).

The obtained numerical solution converges both in the velocity components steady-state values and in the calculated grid, and the resulting velocity parameters can be seen in the current work (tables 1 and 2). The nozzle geometry is determined by the extreme design nodes position.

The inverse problem has not been completely solved; to take into account the supersonic flow peculiarities at Mach numbers greater than 3, more effects should be taken into account and additional boundary conditions must be set, therefore, the authors leave the diffuser further calculation using the characteristics' method up to hypersonic speeds (Mach numbers greater than 7) for future research.

The plans for future research are to apply the characteristics’ method for a gases' mixture and to carry out an end-to-end calculation when calculating the confuser, taking into account the physical and chemical transformations, continuing the work [8].

This calculation disadvantage is a boundary layer absence (as a viscous effects result) along the nozzle walls. But if in a subsonic flow the viscosity can be taken into account in the same way as physicochemical transformations (combustion products), then phenomena called separated flows will appear in the diffuser, which calculates by the selected methods impossible. The viscosity effect on the shape and the nozzle dimensions in comparison with the numerical solution obtained in this work can be taken into account experimentally or using the finite element method.

References

[1] Krasnov N F, Koshevoy V N and Kalugin V T 1988 The Separated Flows Aerodynamics (Moscow, Russia: Higher school)

[2] Krivtsov A V 2013 The Premixed Components in ANSYS Fluent Homogeneous Combustion Processes Study (Samara, Russia: SSAU)

[3] Pirumov U and Suvorova V 1986 Numerical solution of an inverse problem of nozzle theory for a two-phase gas-particle mixture Fluid Dynamics 21 595-603

[4] Landau L D and Lifshits E M 1986 A Compressible Gas one-Dimensional Motion (§ 97 Gas Outflow Through the Nozzle) (Theoretical Physics Vol. 6 Hydrodynamics) (Moscow, Russia: Science. Ch. ed. phys-mat. lit.)

[5] Shifrin E G 2001 Potential and Vortex Transonic Ideal Gas Flows (Moscow, Russia: Fizmatlit)

[6] Shifrin E 2015 Potential 2D flows of real gases. Invariant equations in natural coordinates Hodograph transformation International Journal of Applied Engineering Research 10 34190-3

[7] Samarsky A A 1971 Introduction to the Difference Schemes Theory (Moscow, Russia: Science)

[8] Shifrin E, Chikitkin A, Petrov M and Dushkov R 2018 Design of Laval nozzle with account for
real gas *Aerospace* 5(3) 96-102

[9] Krasnov N F 1976 *Aerodynamics* 1 (Moscow, Russia: Higher school)
[10] Kochin N E, Kibel I A and Rose N V 1963 *Theoretical Hydromechanics* 2 (Moscow, Russia: Fizmatlit)
[11] Kuzmin V I and Gadzaov A F 2012 *Methods for constructing models based on empirical data* (Moscow, Russia: MIREA)