SAMARA UNIVERSITY

A. A. Gorshkalev D. A. Uglanov, A. A. Shimanov

Investigation of gas flow in a heated channel

Methodical instructions for laboratory work

САМАРА 2017

CONTENTS

1. DESCRIPTION OF LABORATORY INSTALLATION

Purpose of the work: using numerical simulation in Fluent program to estimate the effect of friction and heating on the subsonic air flow moving in a cylindrical pipe of unchanged cross-section, determine the velocity diagrams in the cross section of the gas flow, the change in static pressure and other parameters of the subsonic gas flow along the channel axis. Compare the data with the experimental data.

The working site of the installation for this laboratory work (Fig. 1) is a vertical cylindrical tube with an internal diameter d = 9 mm and a length l = 930 mm, connected to the common vacuum system pipeline through a flow valve.

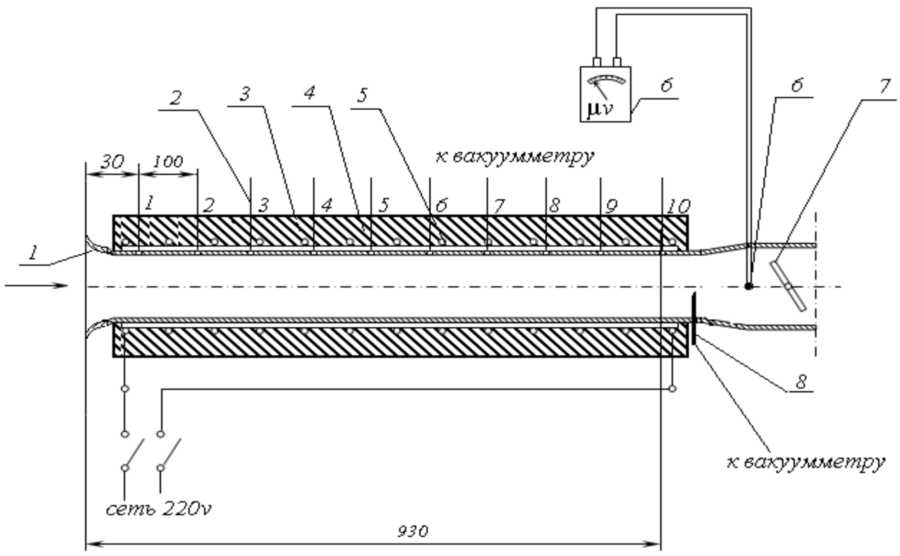


Figure 1 - Scheme of the working area of the installation

Atmospheric air from the room through the inlet pipe 1 is drawn through the pipe through a vacuum pump. The change in pressure drop in the pipe (change of air flow regimes) is effected by the flow valve in the pipeline or throttle 7 in the pipe (Figure 1). The air is heated by means of an electric spiral 5 wound around the pipe and fixed with a casing 3 with asbestos thermal insulation 4. Thus, based on what was said above, it can be assumed in the calculation that the working medium in the channel is air. The pressure and temperature at the inlet are equal to the corresponding parameters of the environment. The outlet pressure is equal to the backpressure measured during the experiment.

2. CONSTRUCTION OF THE CALCULATION MODEL

2.1 Construction of the calculation model

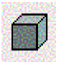
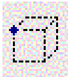
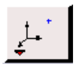
As is known from previous studies, the calculation model is built in the Gambit program. At the first stage of constructing the model, the contour points of the channel will be constructed. Then, on their basis, the boundaries of the computational domain will be obtained with the help of arcs of a circle, segments and splines, which will form the basis for creating surfaces of a two-dimensional computational model.

Table 1 – Coordinates of the profile of the investigated channel

x, мм	y, мм
921	0
921	-9
0	-9
-9	-20,5
-9	11,5
-25	-20,5
-25	11,5
0	0
Coordinate X of the center of the rounding of the upper edge, mm	2
Coordinate Y of the center of the rounding of the upper edge, mm	15
Radius of the input circle r_1, mm	16
Coordinate X of the center of the rounding of the lower edge, mm	2
Coordinate Y of the center of the rounding of the lower edge, mm	-24
Radius of the output circle r_2, mm	4,5

2.2 Construction of base points

Using the following commands, the menu for building points by coordinates is called:

Geometry  → Vertex  → Create vertex 

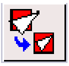
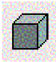
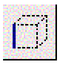
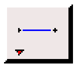
In the menu that appears, in the Global field (in the global coordinate system), one should alternately enter coordinates of all points in accordance with table 1. For example, the coordinates of the first point (921; 0) are entered, and then they are confirmed by pressing the Apply button. Similarly, you should enter all the points from the channel profile table. In the event of an error, the action can be canceled with the help of the button  ("cancel"). To see all the constructed points, you can use the button  ("enter into the screen"). The result of the image is shown in Fig. 2.

Figure 2 – Constructed baseline profile points

2.3 Construction of the channel profile contour

The channel profile is formed by two arcs of circles (the leading edge) and segments. It is most convenient to begin construction from the segments. To do this, call the appropriate menu with the following commands:

Geometry  → Edge  → Create edge 

To build a channel outline, you must select the corresponding points in the order by using the mouse and pressing the Shift key. To accept the action, click the Apply button. The result of the action is shown in Figure 3.

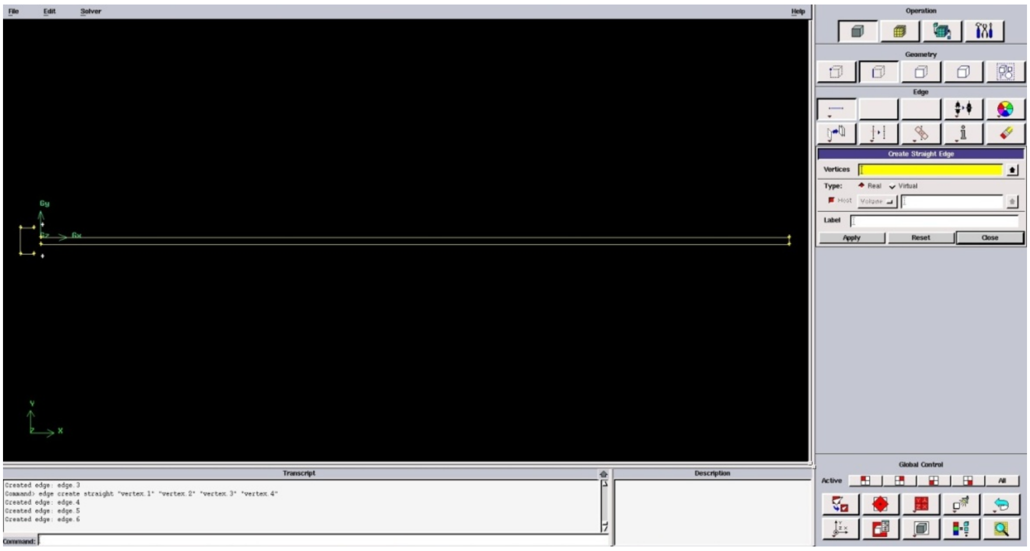
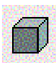
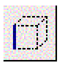


Figure 3 - Line construction menu and channel build result

The next stage is the construction of the arcs that form the edges of the channel entrance opening. To do this, call the menu for building arcs (Figure 4):

In this menu, you need to do the following:

Geometry  → Edge  → Create arc 

– in the Method column, you need to select the construction method (using the centre and two points);

– in the Center field select the center of the circle of the input edge;

– in the End-points field select adjacent points of the input edge;

– leave the Arc field unchanged;

– Start the command with the Apply button

The result of the command is shown in Figure 5.

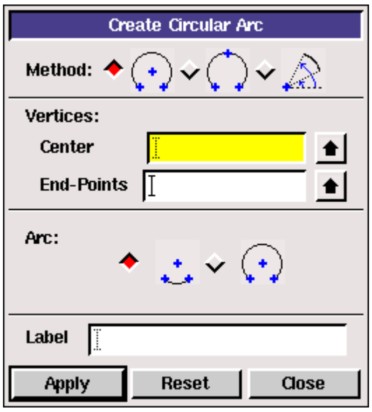


Figure 4 – Arc Build Menu

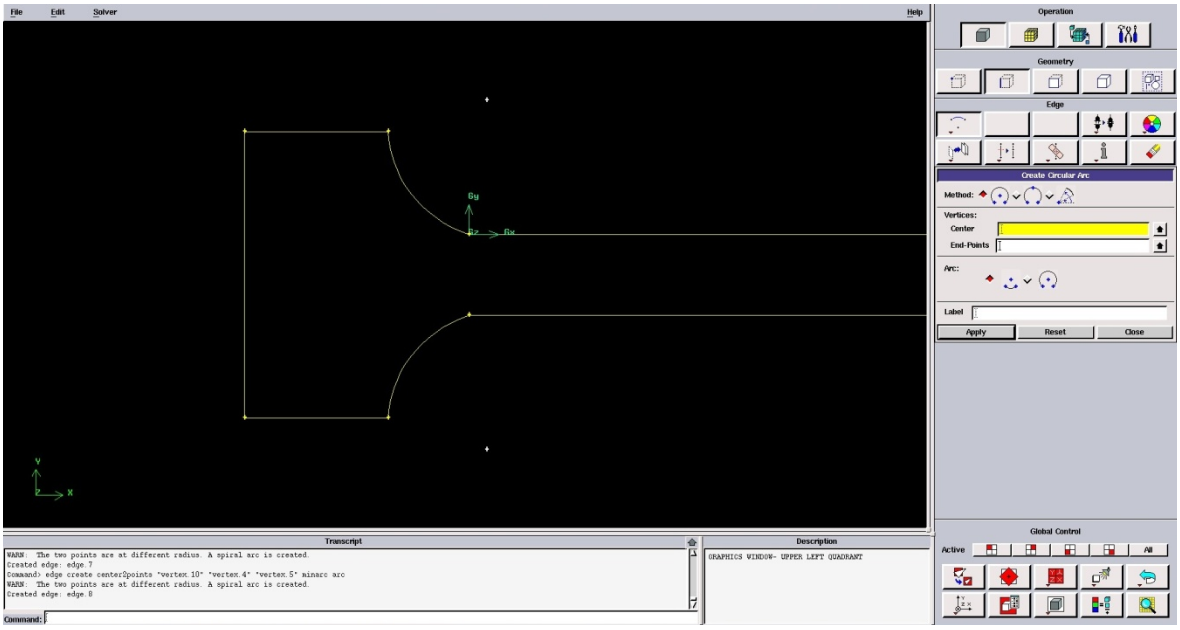
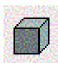
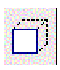
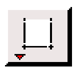


Figure 5 – Result of constructing arcs

2.4 Surface construction

The basis for constructing a finite-element grid of a two-dimensional model is a surface. It is built using the surface build menu (Figure 6):

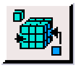
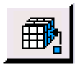
Geometry  → Face  → From Wireframe 

In the appeared menu it is necessary to put the cursor in the Edges window and use the mouse to select the lines forming the closed contour of the calculation area. To build the surface, you should click the Apply button. As a result, the lines of the contour can change color.

Figure 6 – Menu for plotting surfaces using lines

2.5. Specifying boundary surfaces

To exit to the boundary conditions setting menu (Figure 7), press the following buttons in the main menu:

Zones  → Specify Boundary Types 

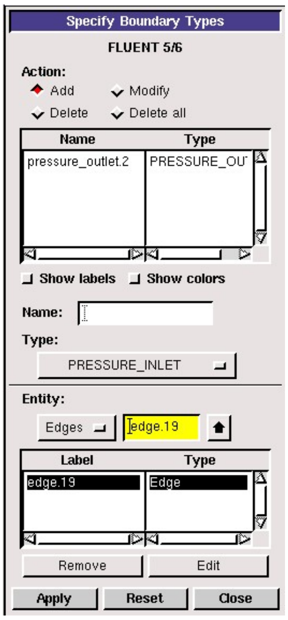


Figure 7 – Boundary conditions menu

In this menu, it is necessary to set the boundary condition pressure inlet and the boundary condition pressure outlet. The result of the construction of the boundary conditions is shown in Figure 8. On the rest of the model bounding surfaces, which were not marked as boundary surfaces, by default the wall boundary condition will be established(Wall).

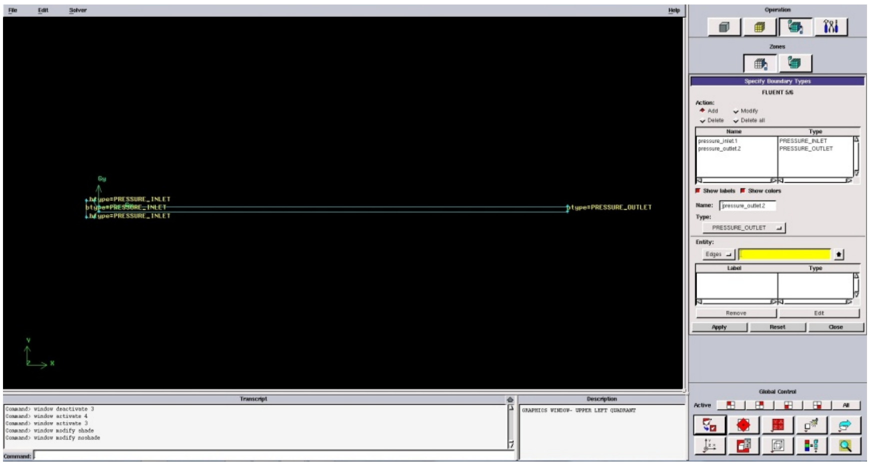
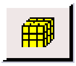
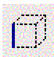


Figure 8 – The result of setting the boundary conditions

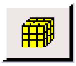
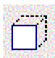
2.6 Construction of finite element mesh

For the correct specification of the periodical boundary condition, it is necessary that the boundary data have a bound finite-element grid. You can establish the connection with the help of the following commands:

Mesh  → Edge  → Link edge 

In the dialog box that appears, use the mouse to select a pair of periodic elements, for example side lines of the input area, and confirm the execution of the command with the Apply button.

To solve this problem with sufficient accuracy it is possible to use unstructured decomposition of the investigated channel. The surface is split using the following command:

Mesh  → Face  → Mesh face 

As a result, the menu shown in figure 9 appears.

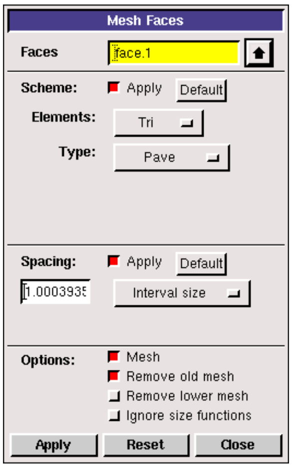


Figure 9 – The grid building menu

It is required to perform the following manipulations:

– In the Face field, select the surfaces that will be broken.

– In the Elements field, the end element type is Quad (quadrilateral), Tri (triangular), or Quad / Tri (mixed).

– In the Type field, select the Pave circuit, which is used to perform the debugging.

– The Spacing field (Figure 9) introduces the required size of the final element (for example, 1mm).

– For the construction of a finite-element mesh with the selected parameters, click the Apply button.

The result of the command is shown in Figure 10.

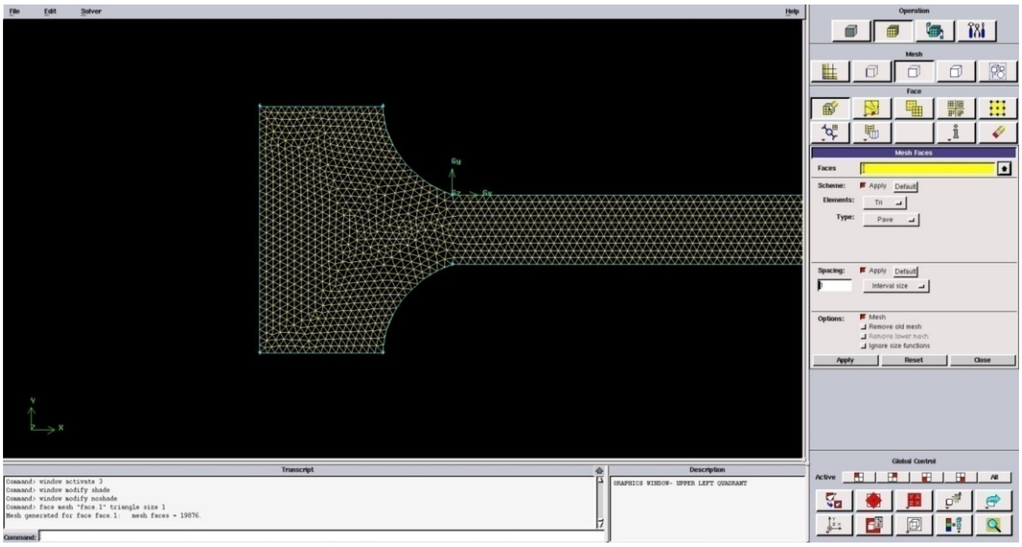
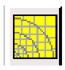


Figure 10 – The result of constructing a triangular unstructured finite element mesh

To obtain more accurate calculations, it is proposed to thicken the finite element mesh only in places with large gradients of parameters. This action is performed using the Size Function menu (Figure 11), which is called by the command:

Tools  → Size function 

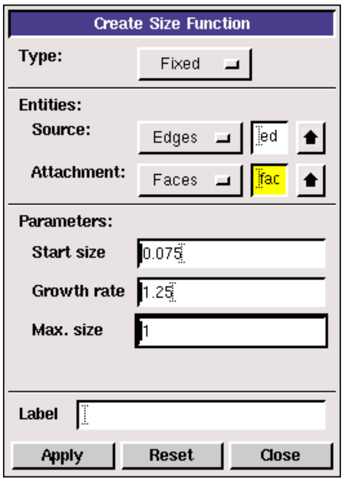


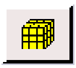
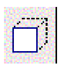
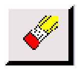
Figure 11 – Size Function Menu

In the appeared window in the Source field it is necessary to describe that the source of condensation is the Edges line. Then use the mouse pointer to select the lines next to which the thickening will take place. In the case under consideration, these are the walls of the channel.

In the Attachment field, select the element to which the size function will be applied. In the case under consideration, this is the surface of the computational model. Therefore, in the list select Face and use the mouse pointer to mark the surface of the calculation area.

In the Start size field enter the size of the final element near the surface of the thickening, for example, 0.075 mm. In the Growth rate field, the number of cells of the i + 1-th row is more than the cells of the i-th row (for example, 1.25). In the Max.size field, the maximum size of the finite element cells is entered, after which the thickening stops (for example, 1 mm). To apply the entered settings, click the Apply button.

You can see the action of the Size Function command by re-constructing the grid on the surface of the model as it was done before. Before this, you must delete the previously built grid using the command:

Mesh  → Face  → Delete face mesh 

Use the mouse to select the surface from which you want to remove the grid. The 'Remove unused lower mesh' button should be left clicked. This will remove the partitioning into the final elements of the faces (the relationship of the partition will not be violated). To confirm the deletion command, click the Apply button. The result of dividing the computational domain by a triangular unstructured grid in view of the Size Function is shown in Figure 12.

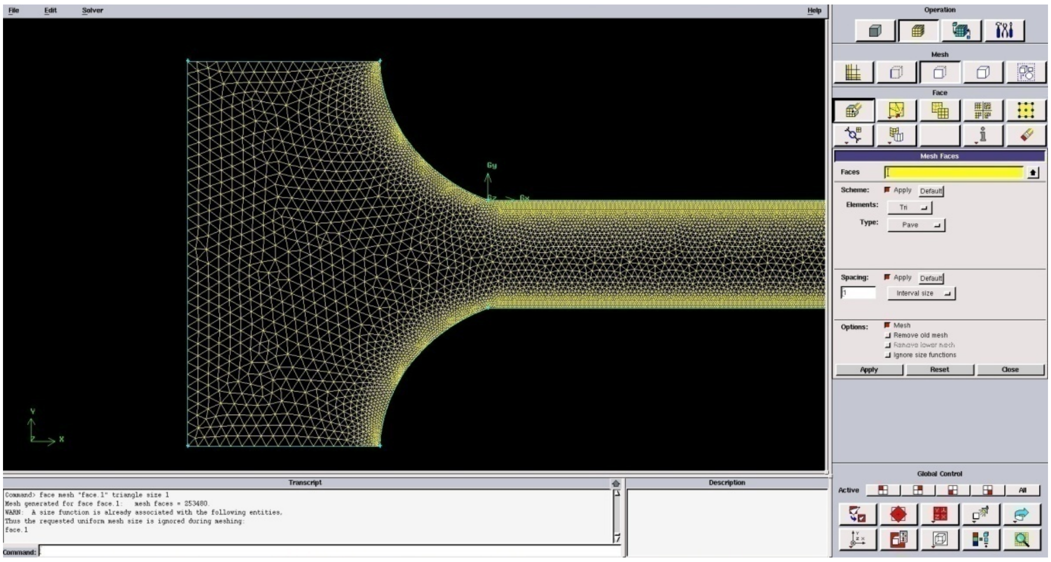


Figure 12 – The result of constructing a triangular unstructured finite element mesh with thickening near the walls

2.7 Transfer of the constructed settlement model to Fluent

To export the created model, select the following items in the upper menu:

BM: File → Export → Mesh.

In the appeared window, you need to enter the name of the exchange file and using the Browse button you can select the place where the file should be saved. Since the computational model is two-dimensional, it is necessary to click the Export 2D (X-Y) Mesh button and then press the Accept button.

2.8 Closing Gambit

This action is performed by the command:

BM: File → Exit.

3. CALCULATION AND ANALYSIS OF RECEIVED DATA

The Fluent program runs and all other actions are performed directly in it:

3.1 Reading a calculation model created in Gambit

To execute this command, you need to select in the main menu:

MM: File → Read → Mesh.

In the appeared standard window of the Windows Explorer, you need to find the place where the exchange file was saved, select it and confirm with OK.

3.2 Checking the finite element mesh for errors

It is implemented with the help of the command:

MM: Mesh → Check.

If an error is found, a corresponding message will be issued. In this case, you need to return to the Gambit program, find the error and fix it.

3.3 Scaling of finite element mesh

Since the channel model was implemented in the Gambit module, and the dimensions of the computational models in the Fluent program must be set in meters, it is necessary to reduce the constructed grid by 1000 times. To do this, the program calls a special command from the main menu (Figure 13):

MM: Mesh → Scale.

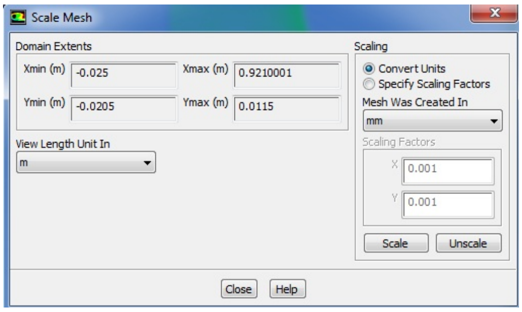


Figure 13 – Scale Mesh Men

In the Scale Mesh menu in the Mesh was created in field, you must select millimeter mm (or another unit of measurement in which the calculation model was created), and then click the Scale button. Scaling will be done. After the operation is completed, the menu must be closed using the Close button.

3.4 Viewing a finite element grid

To view the finite element mesh of the model under study, it is possible with the help of the command:

MM: Display → Mesh.

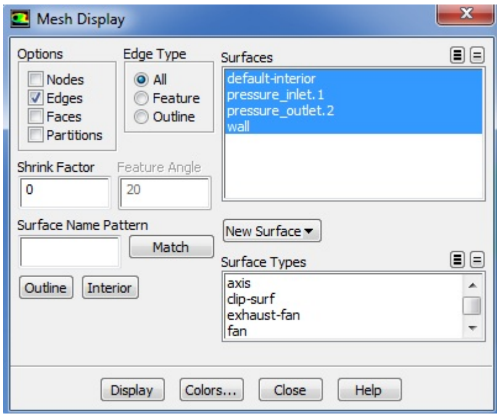


Figure 14 - Mesh Display Menu

In the Surfaces window (Figure 14), you need to select all the boundary surfaces that the user wants to view. After performing these manipulations, you need to press the Display button. As a result, the calculated grid appears in the graphics window (Figure 15).

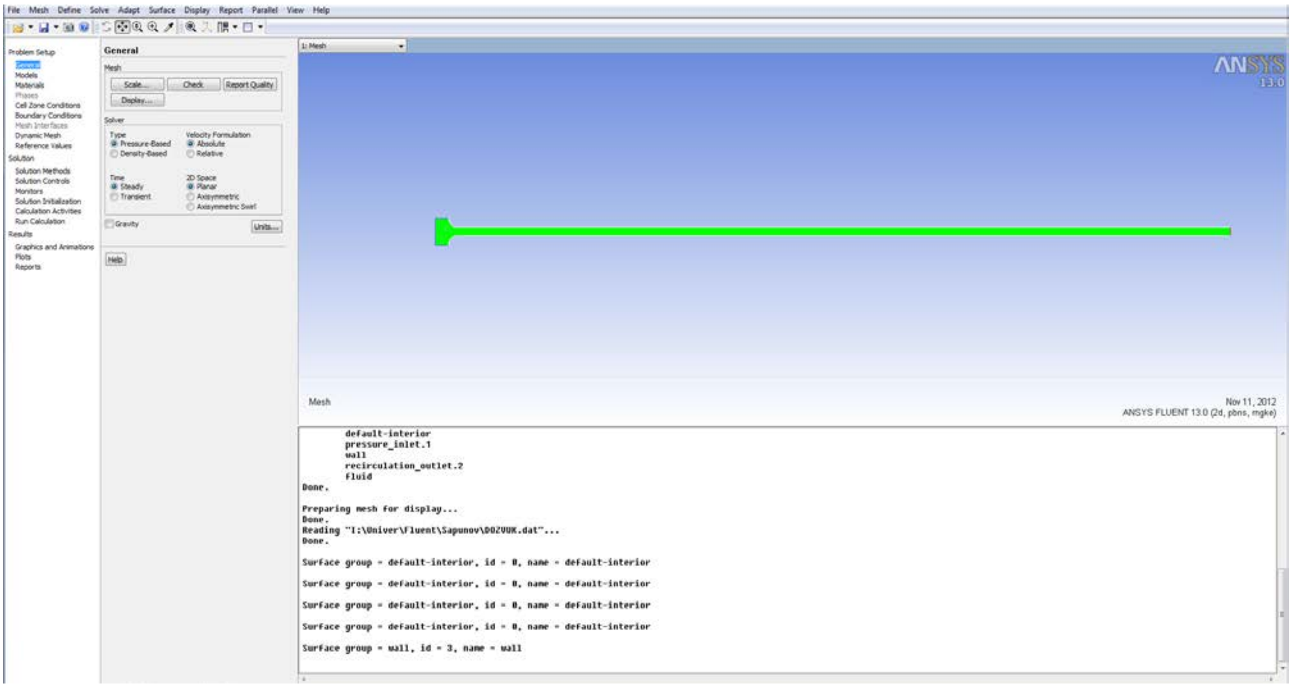


Figure 15 - Result of displaying the grid

3.5 Setting the Solver Options

When solving this problem, it is necessary to choose a solver with the help of which a solution will be made. This choice is made with the help of the Solver command:

MM: Define → General.

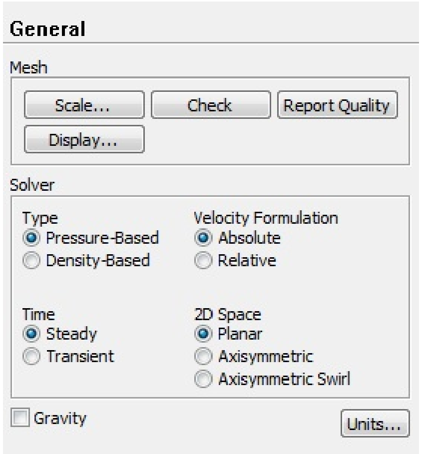


Figure 16 - The Solver Menu

And in this menu (Figure 16) you need to set the following settings:

In the Solver field, you should choose the algorithm for solving the problem under consideration. It is advisable to select Pressure Based;

In the 2D Space field, Planar is selected - flat;

in the Time field it is established that the solution of this task will be Steady or Transient (in this task - Steady).Рисунок 16 - Меню Solver

3.6 Accounting for the calculation of the energy equation

When solving this gas dynamics problem it is necessary to take into account the change in the flow temperature and heat exchange. Therefore, you should connect the energy equation to the solution using the command:

MM: Define → Models→ Energy.

In the window of this program, check the Energy Equation box and click OK.

3.7 Determination of the model of turbulence

To specify the model of turbulence, select the command:

MM: Define → Models→ Viscous.

To solve the problem in the list of turbulence models, it is necessary to select the turbulence model k-ε (k-epsilon). In the menu that appears, the RNG model is marked and a tick in the Viscous Heating column is used to account for the heat released from the viscous friction of the gas layers.

3.8 Setting the properties of the working body

This operation is performed in the Material menu (Figure 17):

ГМ: Define → Materials.

The working body in this task is air, which in the default Fluent program settings. When solving problems of gas flow in canals with heating, one should take into account the compressibility of the body. Therefore, it is necessary to determine the dependence of the gas density on the flow parameters. To do this, use the equation of state of an ideal gas (Mendeleev-Klaiperon) and therefore
In the Materials menu, in the Density list, select Ideal-gas.

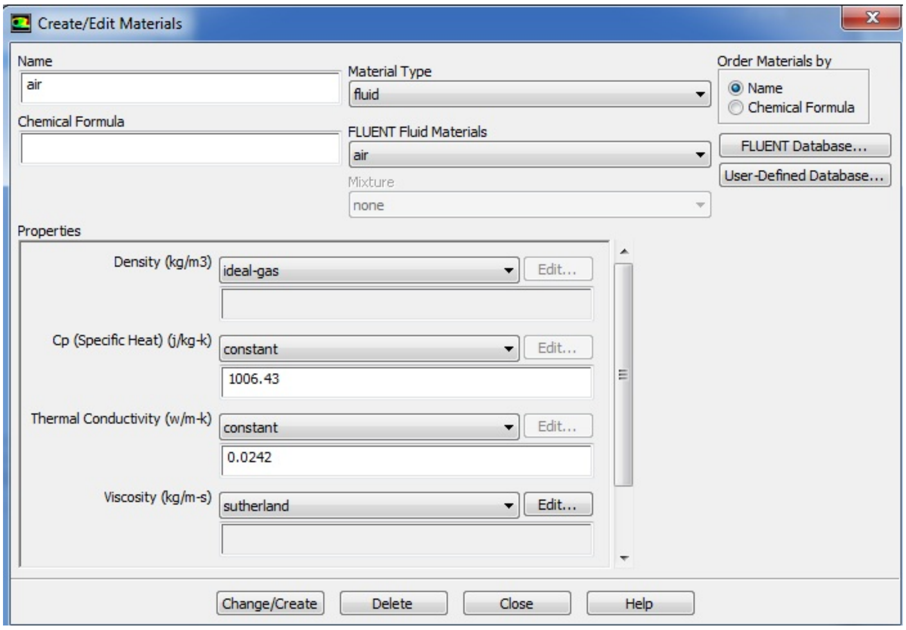


Figure 17 - The Materials menu

The dependence of viscosity on temperature is usually given in the form of the Sutherland equation:

$$\mu=\mu_0\left(\frac{T}{273}\right)^{\frac{3}{2}}\frac{273+C}{T+C}.$$

To describe this in the program on the Materials menu in the Viscosity list, select Sutherland. All changes to the properties of the working body are saved by pressing the Change / Create button. After the operation is completed, the menu should be closed with the help of the Close button.

3.9 Setting the reference pressure

As a "reference pressure", 0 should be taken to ensure that the calculation results and the initial data are specified in absolute values. You can change the value of "reference pressure" in the menu that appears as a result of the command:

MM: Define → Operating Condition.

To simplify the processing of results in the problem being solved, it is advisable to take "reference pressure" equal to zero, and enter its value in the field Operating pressure.

3.10 Setting Boundary Conditions

The Boundary Condition command calls the menu for setting boundary conditions (Figure 18):

MM: Define → Boundary Condition.

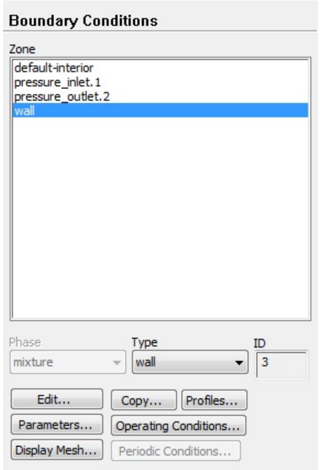


Figure 18 - Boundary Condition menu

Then, in the Zone window, select the desired boundary condition, be sure that in the Type window, the type of the boundary condition is specified correctly, and click then Set. As noted above, the following conditions will be met in the problem under consideration:

– at the output boundary, a static pressure equal to the back pressure, which is recorded during the experiment, is set;

– at the input boundary, the total pressure is set equal to atmospheric pressure;

– the heat exchange condition is set on the channel walls.

The input boundary condition "pressure-inlet" is set in the following sequence (Figure 19):

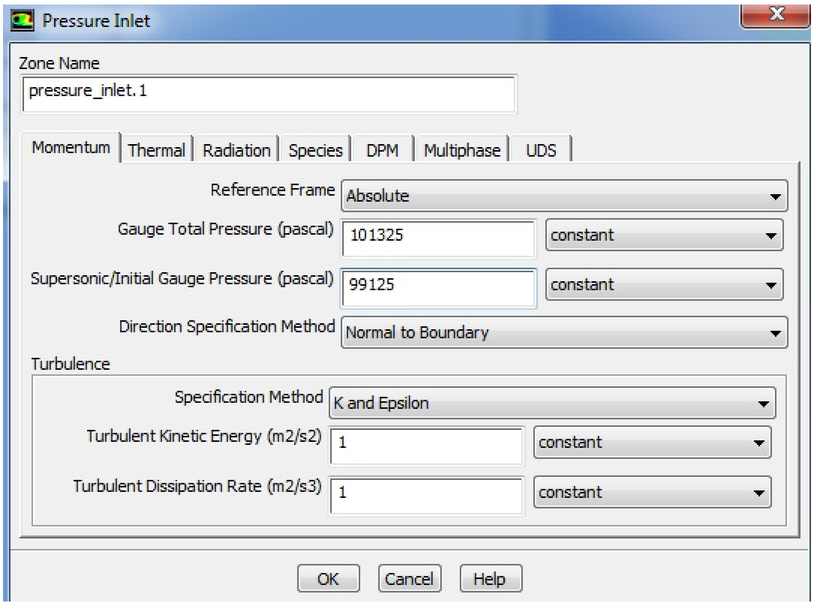


Figure 19 - The menu Pressure inlet

In the field Gauge Total Pressure is entered the value of the total pressure at the entrance to the calculation area. In the problem under consideration, it is equal to atmospheric.

The Supersonic Gauge Pressure field specifies the static flow pressure for this case. The flow velocity at the inlet to the channel is zero, and therefore in this field the pressure is equal to atmospheric pressure.

In the Direction Specification Method field, the direction of the velocity vector at the input boundary is determined, which in this case is normal to the input boundary. Therefore, in the Direction Specification Method field, select Normal to Boundary (perpendicular to the border).

To set the full flow temperature at the input, click on the Thermal tab at the top of the menu, and enter the temperature value in the available Total Temperature field. For the example under consideration, the temperature is equal to the atmospheric measured during the experiment.

Similar settings are made to set the boundary condition on the output border in the Boundary Condition menu, where you need to select the name of this border and press the Set button. This will cause the Pressure Outlet menu (Figure 20) to appear, in which settings similar to the Pressure-inlet must be made.

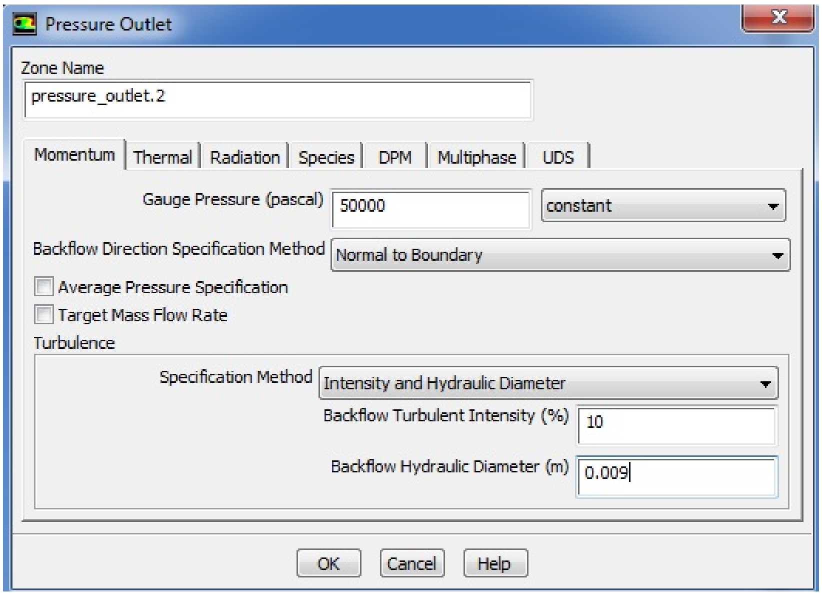


Figure 20 – The Pressure Outlet Menu

In addition to the boundary conditions listed above, the boundary condition of the wall (wall) is present in the problem being solved, which is heated from the outside of the channel. Therefore, in the boundary condition Wall (Figure 21), the value of the heat flux is set. Heat Flux=17321w/m^2.

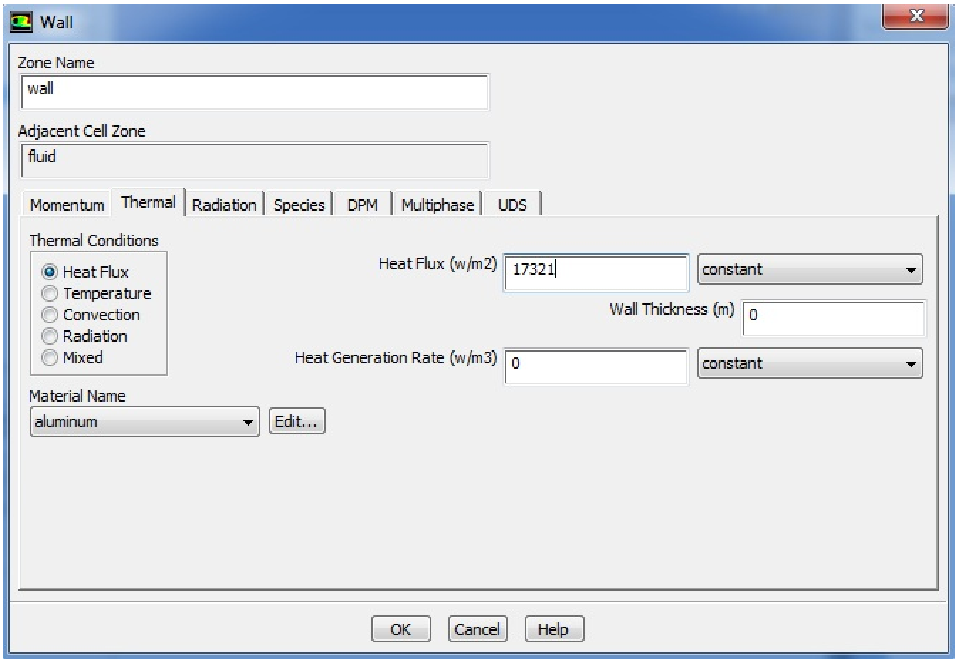


Figure 21 - Wall Menu

3.11 Setting account parameters

The menu for setting account parameters is invoked with the command:

MM: Solve → Methods.

The Solution Methods menu appears on the screen (Figure 22).

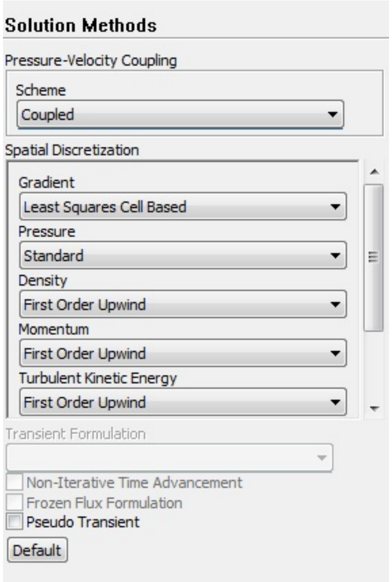


Figure 22 – Menu Solution Methods

In this menu, you need to make the following settings:

In the Spatial Discretization zone, the first order of sampling should be set at the beginning of the calculation, and then it should be increased.

In the Pressure-Velocity Coupling zone, it is advisable to select the Coupled algorithm to solve the problem of the current in the channel under investigation, and to improve stability and convergence, reduce the Courant **number to 50**.

3.12 Setting Initial Values for the Parameters

The menu for setting the native conditions (Figure 4.23) is called by the command:

MM: Solve → Initialization.

In the Compute From field of this menu, you must select the input border. As a result, the recommended values of the initial parameters will be calculated from the input boundary conditions. To accept them, click Initialize.

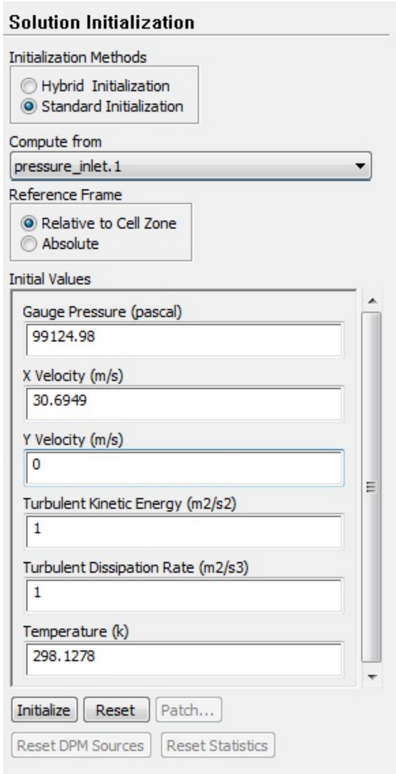


Figure 23 – Initial settings menu

3.13 Configuring the display of the solution process

To monitor the residuals in the calculation process, and also to set the stopping criterion for the solution, you must call up the Residual Monitors menu (Figure 24) using the command:

MM: Solve → Monitors→ Residual.

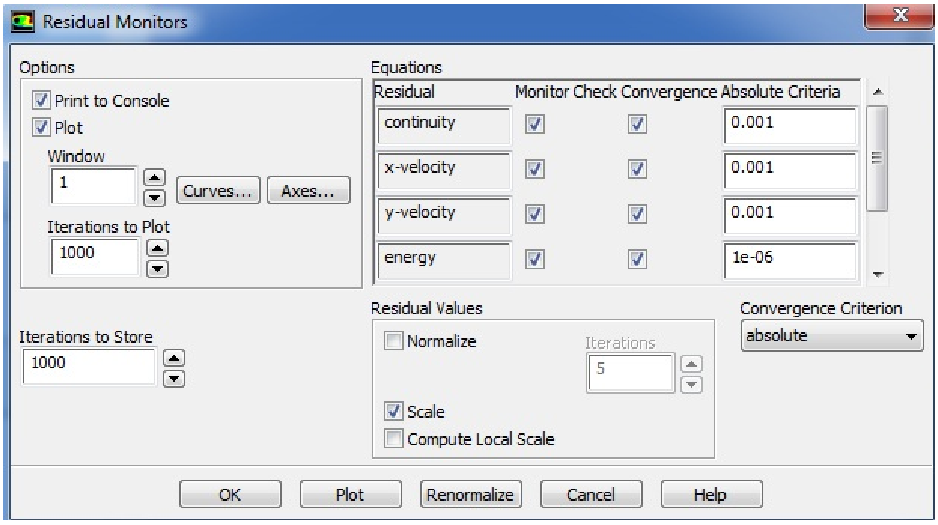


Figure 24 – Residual Monitors menu

In the Option field, check the boxes next to Plot and Print, which will allow you to see the discrepancies on the screen. In the Convergence Criterion field, in order to obtain a fairly accurate solution of each equation, it is necessary to set the value of the residuals 10^{-3} (except 10^{-6} for the energy equation).

To visualize the change in the cost difference between the input and output boundaries in the solution process, you need to call the command:

MM: Solve → Monitors→ Surface→ Create

The menu shown in Figure 25 appears.

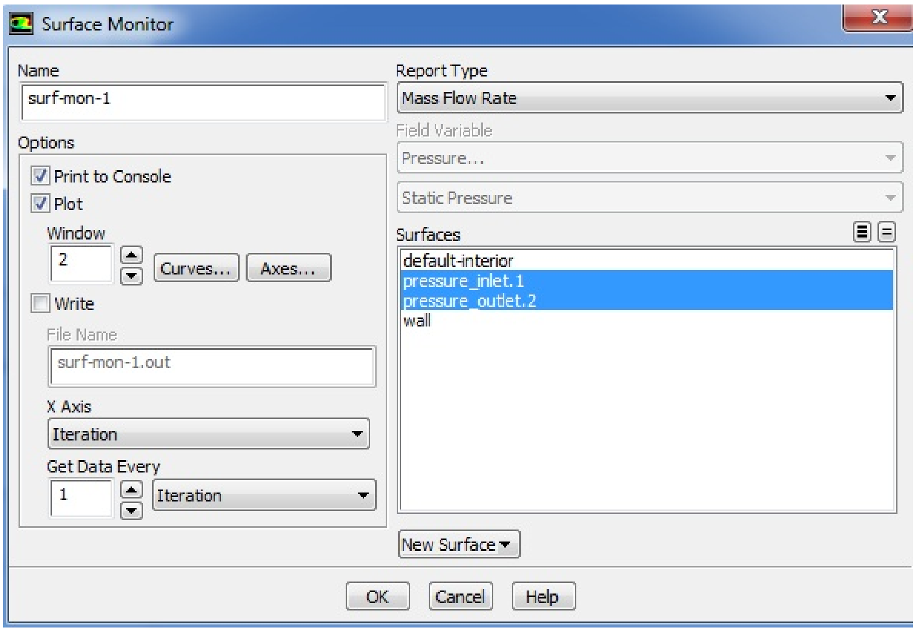


Figure 25 - Define Surface Monitor Menu

In this menu, the Mass Flow Rate item is selected in the Report Type field, and the input and output limits are selected in the Surfaces list.

3.14 Saving calculation model

It is carried out by means of a command call:

MM: File → Write → Case.

Then, in the Explorer window that appears, select the area where the model and its name will be saved.

3.15 Running the solution

It is carried out by selecting the command:

MM: Solve → Run Calculation.

In the Number of Iteration field, enter the number of iterations (for example, 1000), and then the Calculate button is pressed. After the solution is started, the dependence of the residuals on the iteration number for each equation appears in the graphics window (Figure 26), as well as a window showing the change in the difference in the flow rates (Figure 27). As soon as the residual values for each of the equations have reached the specified values, immediately the Fluent program remains the solution and will display message «solution is converged» 1 in the message window.

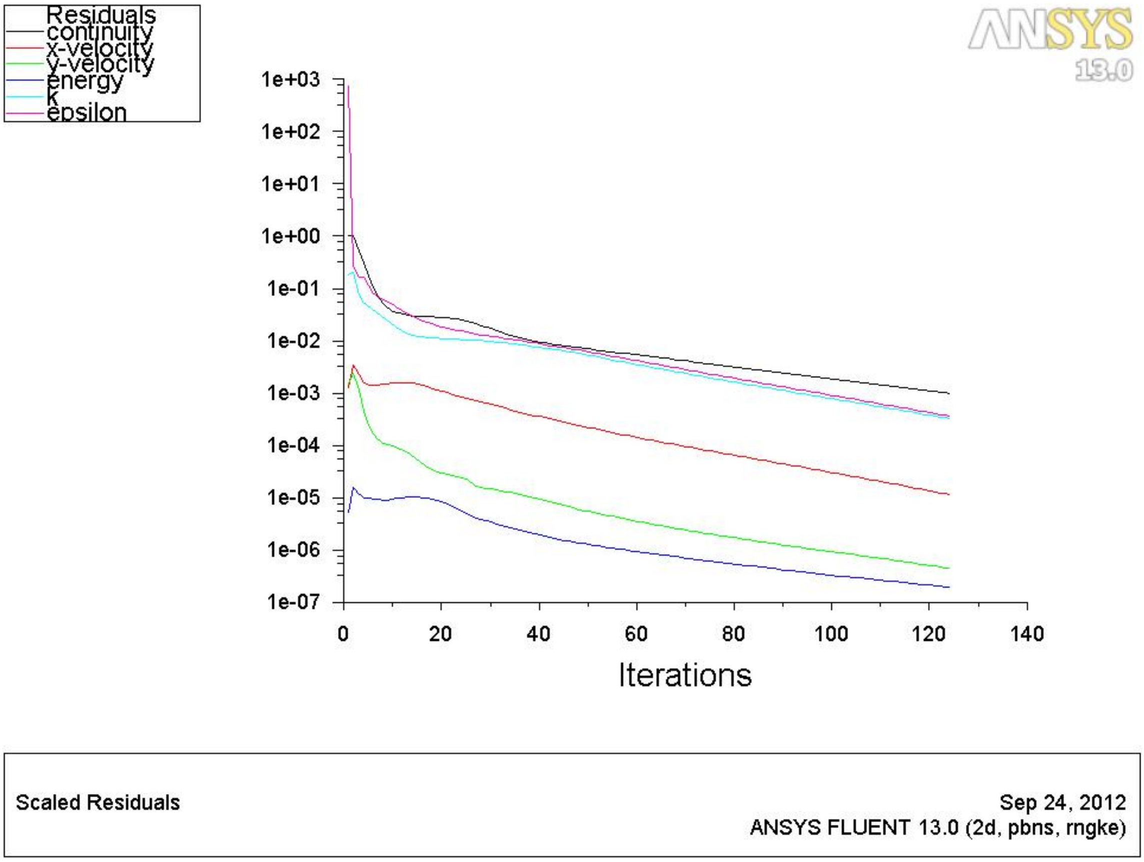


Figure 26 – Variation of discrepancies by iteration

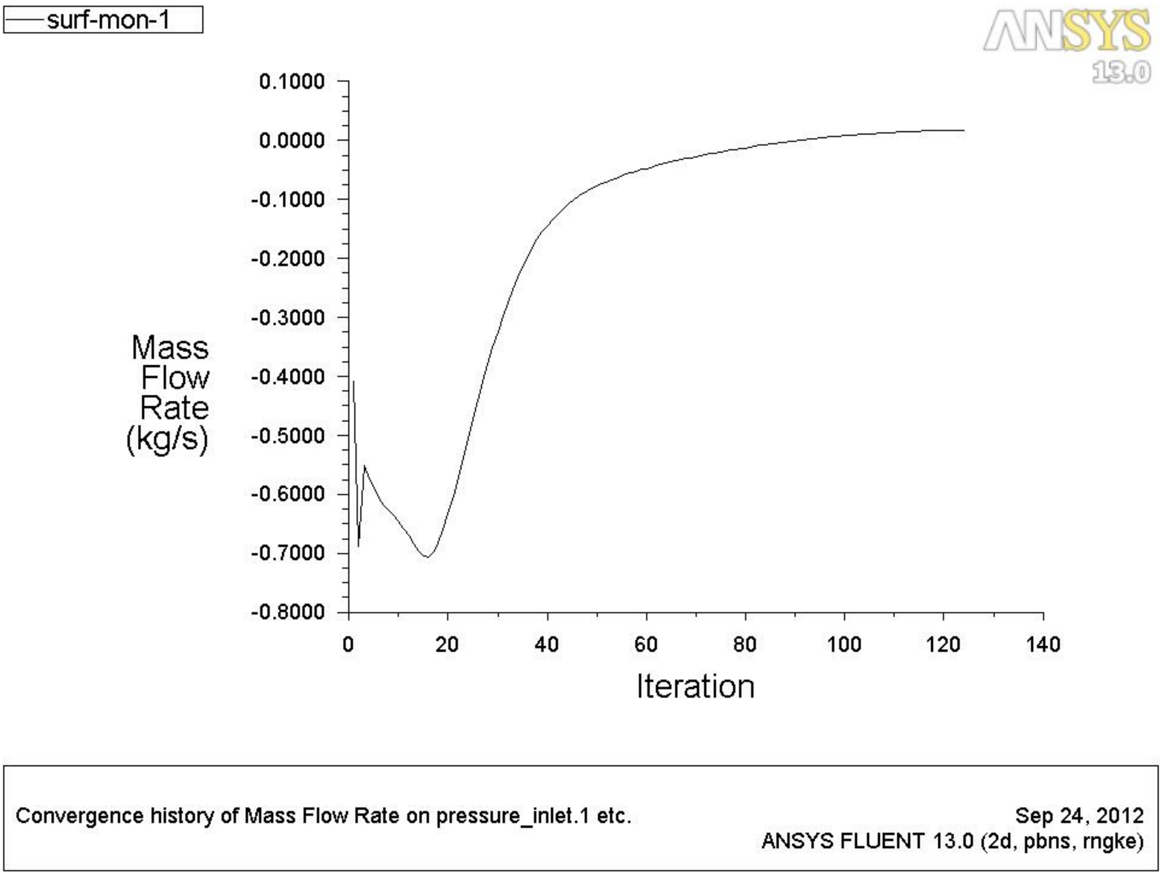


Figure 27 - The change in the difference in costs between the input and the output by iteration

3.16 Increase in the order of sampling

The implementation of this procedure will allow obtaining more accurate calculation results. To do this, you need to go back to the Solution Methods menu (Figure 28):

MM: Solve $\rightarrow$ Methods.

After that, in the Spatial Discretization field, in front of all solved equations, you should set the Second Order Upwing. Then the calculation starts again. The calculation resumes from the place from which it was stopped, but with the new settings of the controller, using the achieved results as a first approximation.

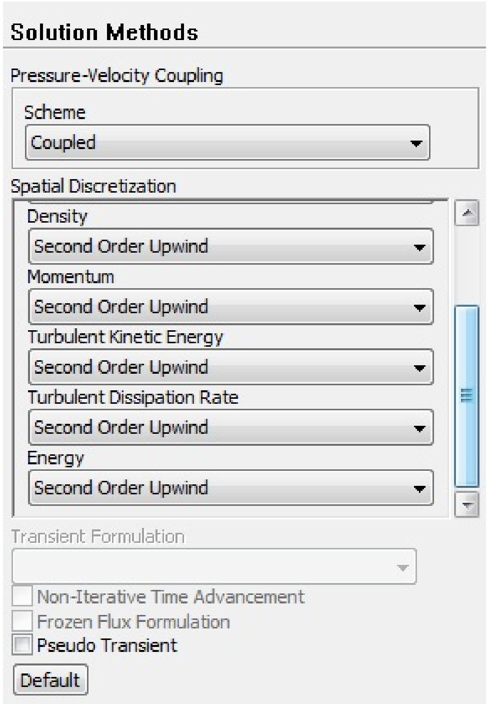


Figure 28 - The Menu of the Solution Methods

3.17 Visualization of the distribution fields of parameters in the calculation area

To do this, run the command:

MM: Display → Contours.

As a result, the Contours menu appears (Figure 29). In the Filled window of this menu, you must set the check box to watch the parameter distribution fields, and then select the parameter in the Contours of field whose modification you want to display. Thus, by performing the necessary settings in this menu, you should obtain the fields of distributions of the total and static pressures, temperatures, flow velocity in the channel (Figure 30 ... 32).

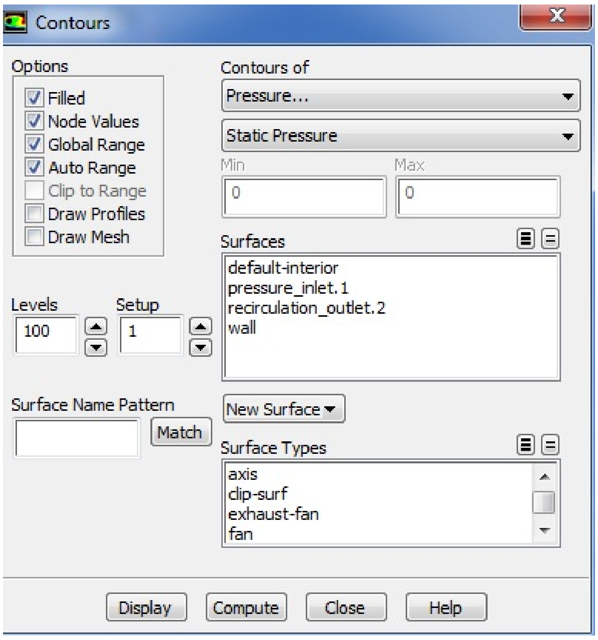


Figure 29 – Menu Contours

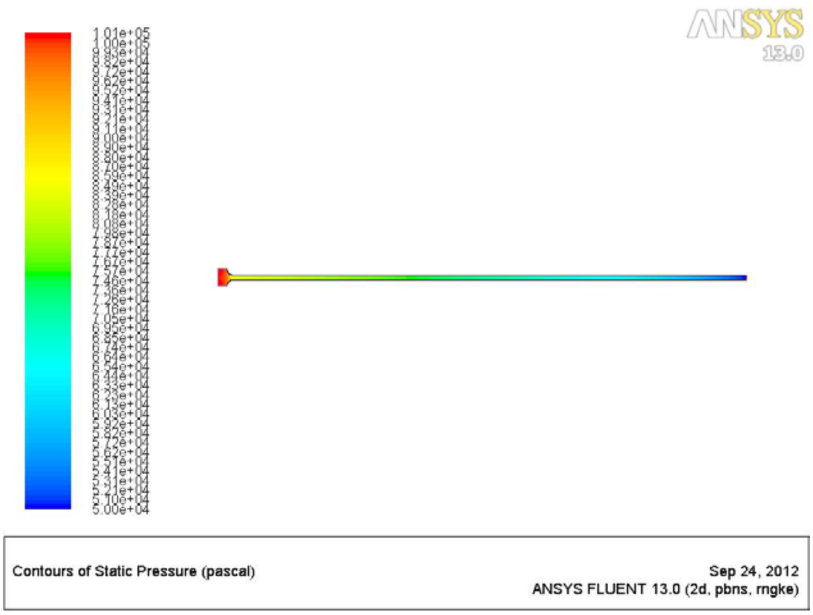


Figure 30 – Static pressure field in the channel

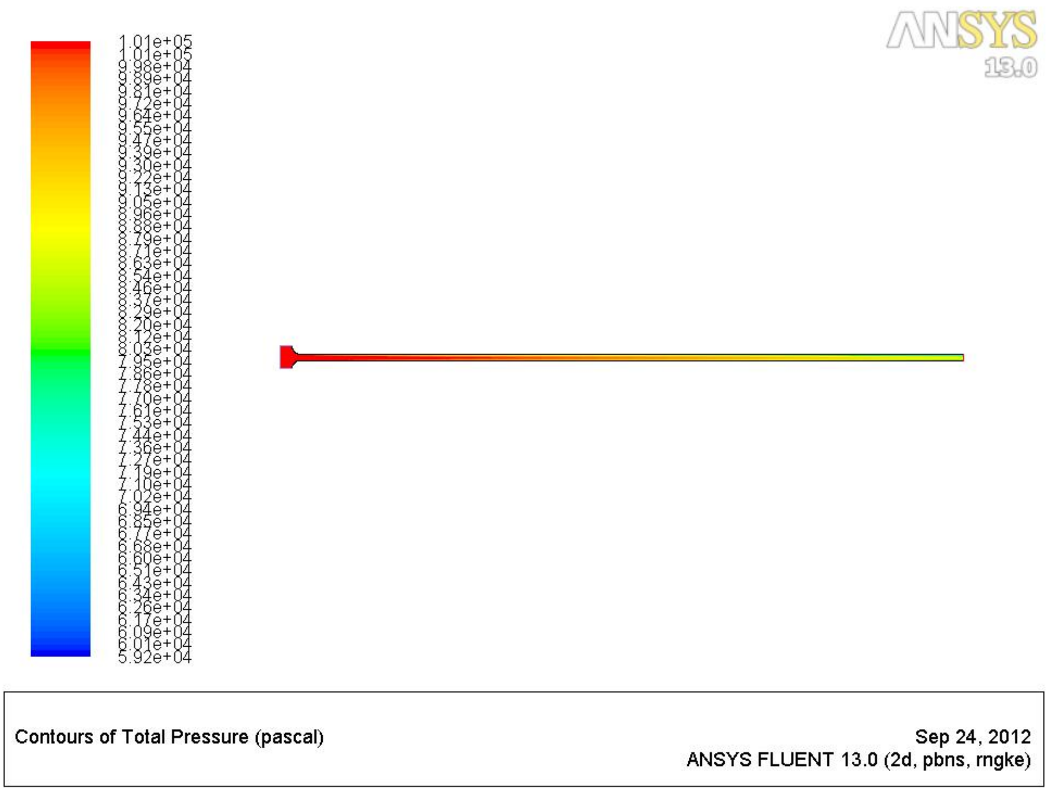


Figure 31 – Full pressure field in the channel

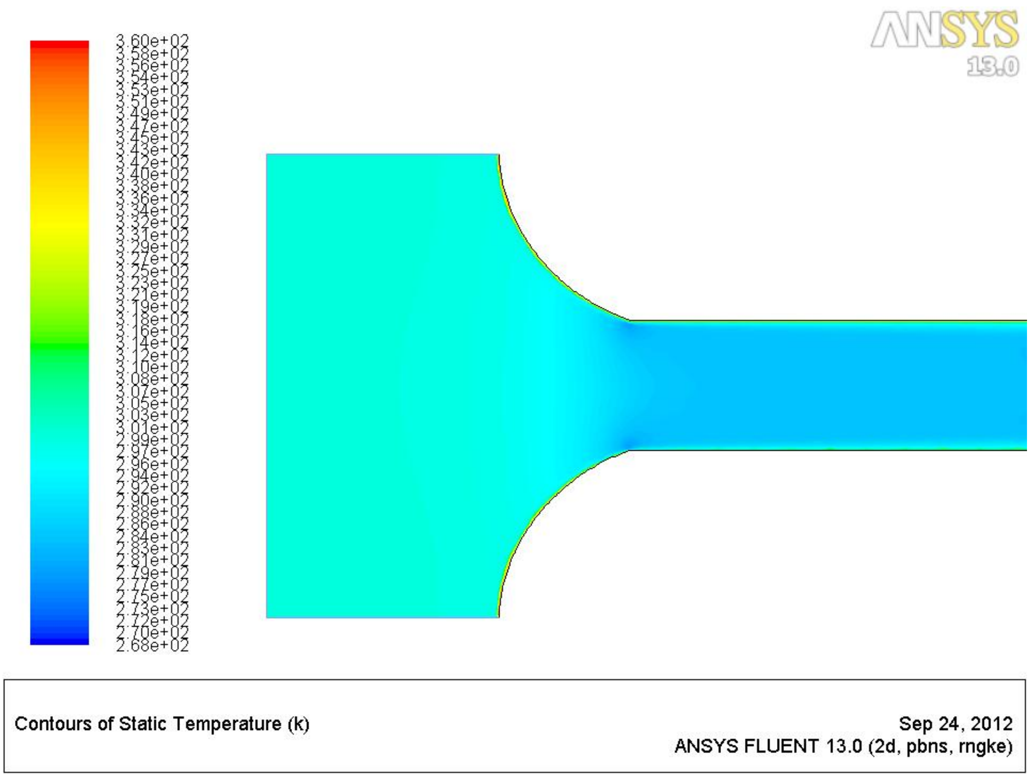


Figure 32 – Static temperature field in the channel

3.18 Visualization of velocity vectors

To display the velocity vectors in the calculation area, use the Vectors menu (Figure 33), which is started with the command:

MM: Display → Vectors.

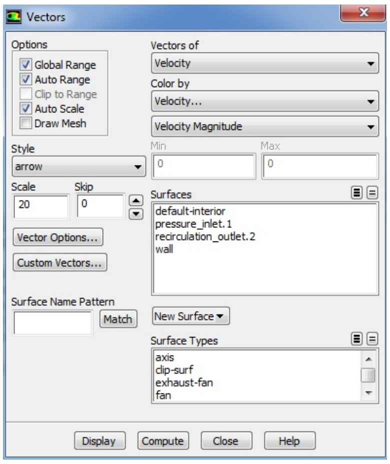


Figure 33 – Vectors menu

The size of the vectors can be changed in the Scale field, and the number of vectors can be reduced by setting a non-zero number in the Skip field. This number determines how many vectors will not be displayed. As an illustration, the distribution of vectors at the entrance edge of the channel is shown in Fig. Setting a checkmark in the Mesh field opens access to the grid display menu. This makes it possible to simultaneously display the grid and the distribution fields of the parameters (Figure 35).

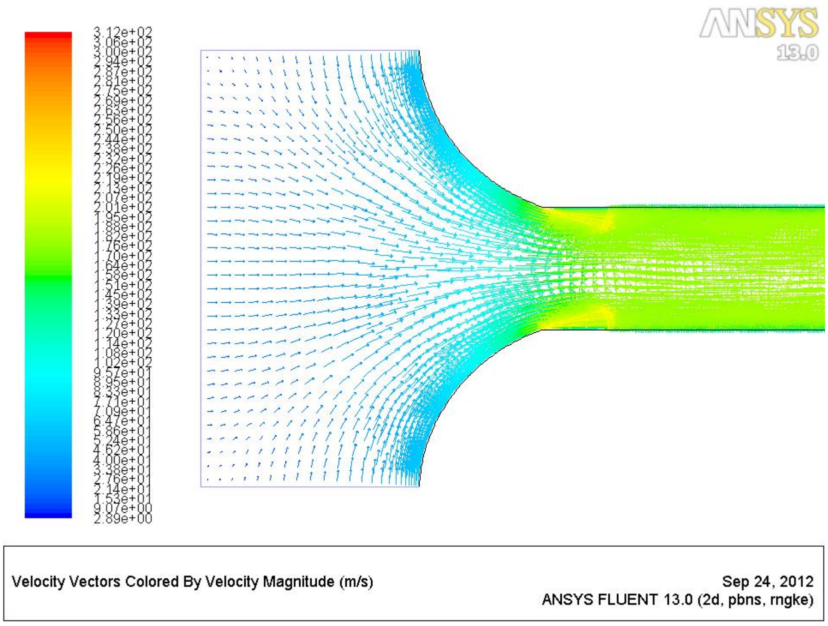


Figure 34 – The distribution of vectors near the channel's leading edge

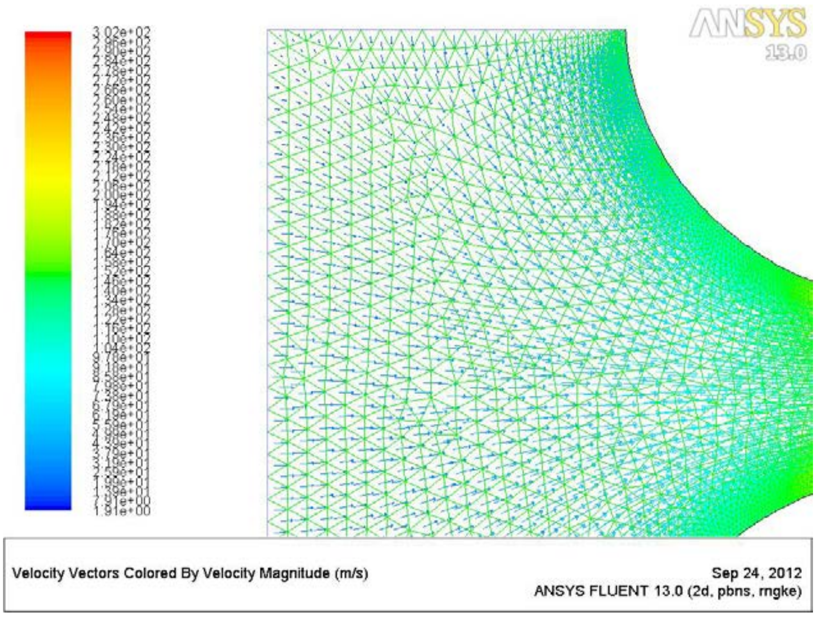


Figure 35 – Simultaneous representation of vectors and grid

3.19 Creating graphs for changing parameters

Then it is necessary to plot the distribution of any calculated parameter along an arbitrary surface or line.

In this work, it is necessary to plot the velocity, total and static pressures, total and static temperatures, gas density, Mach number along the channel axis Y = 0.

To begin with, build the necessary surfaces with the command:

MM: Surface → Iso Surface

In the Surface of Constant field, select the grid (Mesh) →Y-Coordinate. In the Iso-Values field, a coordinate is plotted along the Y axis through which the plane must pass (in our case, Y = -0.0045m). Under the New Surface Name, the name of the created plane is written, for example Y = -4.5mm. To create a plane, click on the Create button. Then the build menu opens, which is called with the command:

MM: Plot → XY-plot.

In the XY-plot menu, tick the Position on Y Axis check box. In order to construct a change in the total pressure at the exit boundary, it is necessary to select the total pressure (Pressure → Total Pressure) as the parameter on the y-axis, and the coordinate x (Mesh → Y-Coordinate) as the parameter along the x-axis. In the Surfaces field, you specify the surface or line on which the distribution is constructed (in the example under consideration, Y = -4.5mm). To create a graph of the settings made, you need to press the Plot button. The changes in other parameters are constructed analogously. If you tick the checkbox in the Write to file box, the resulting graph can be written to a text file that can be used in the future either in the Fluent program or in other programs, for example MS Excel.

Graphs of temperature, pressure and velocity distribution are shown in Figures 36 – 40.

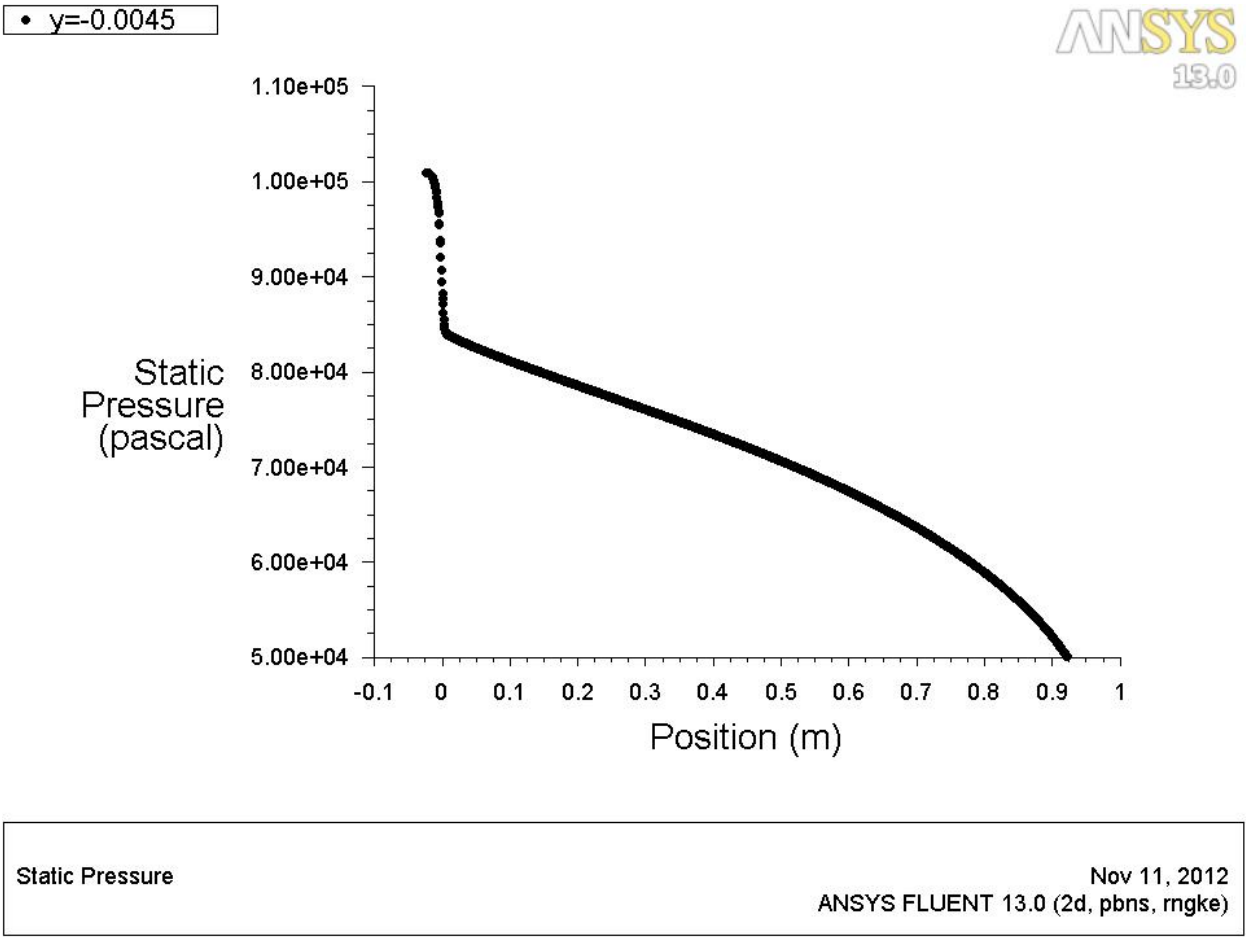


Figure 36 – Graph of the distribution of static pressure along the length of the channel

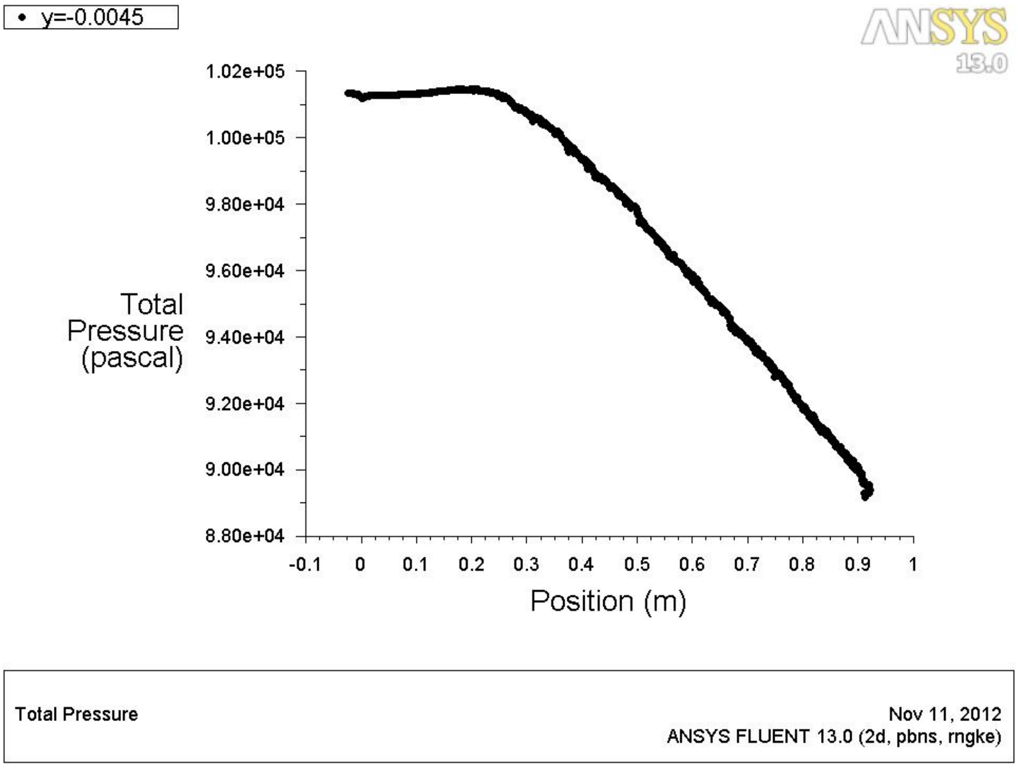


Figure 37 – Graph of the distribution of total pressure along the length of the channel

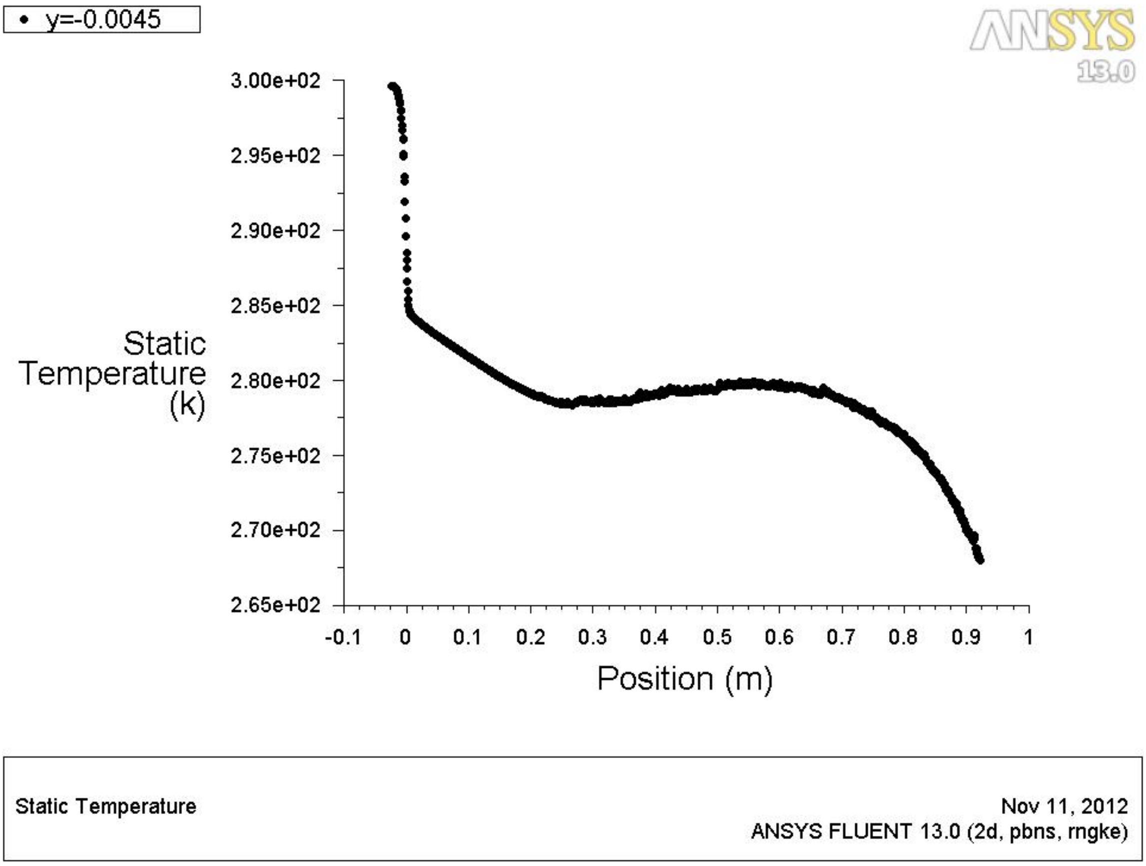


Figure 38 – Graph of the distribution of the static temperature along the length of the channel

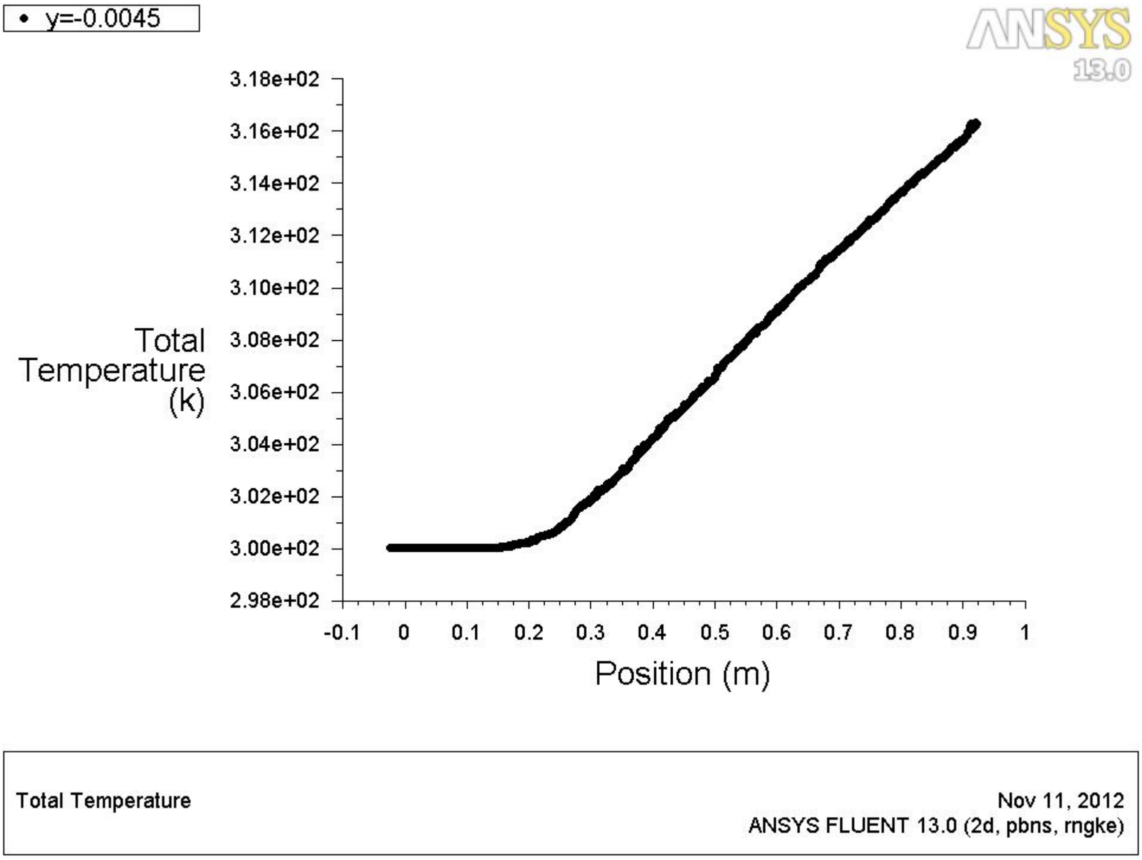


Figure 39 – Graph of the distribution of the total temperature along the length of the channel

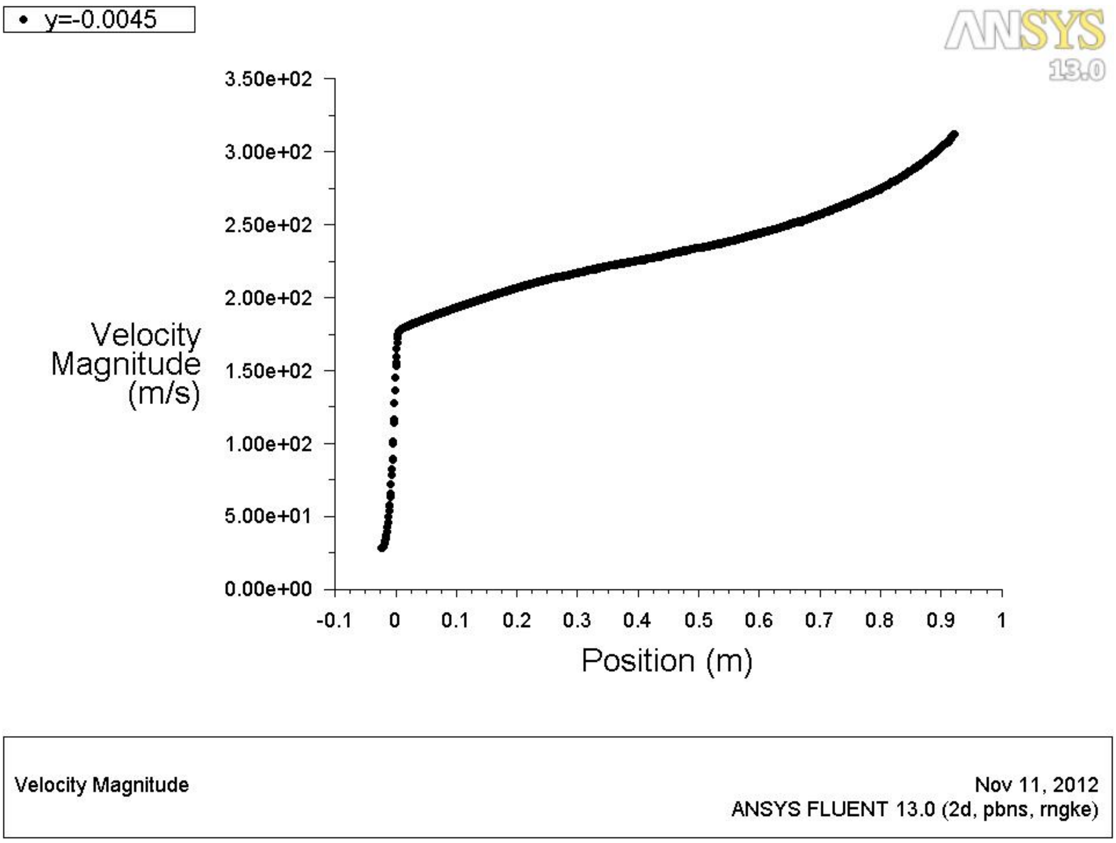


Figure 40 – Schedule of velocity distribution along the length of the channel

4. ADDITIONAL TASK №1

With the help of numerical modeling in the Fluent program, it is necessary to additionally estimate the influence of friction and heating on the subsonic air flow moving in a cylindrical pipe of unchanged cross-section, determine the velocity diagrams in the cross section of the gas flow, the change in static pressure and other parameters of the subsonic gas flow along the channel axis.

Law of treatment of impacts

The parameters of the gas flow vary only when the environment is exposed to it. In this case, the following environmental impacts on gas (gas-dynamic system) are possible:

1. Geometric (ds) - change in the size of the channel cross-section along the flow (expansion or narrowing of the channel);
2. Expendable (dG) - change of mass flow rate of gas in the channel by supply or withdrawal of additional mass through the side surface;
3. Mechanical action (dl_tech) - the exchange of mechanical energy in the form of technical work between the flow of gas and the environment (the operation of a turbine or compressor);
4. Thermal action (dq_n) - supply or removal of heat;
5. Effects of friction (dl_tr) - taking into account the influence of the really existing viscous friction forces within the ideal gas model.

Depending on the sign of each exposure, this influence in the opposite way affects the parameters of the subsonic and supersonic flows. The influence of this or that effect on the parameters of a subsonic or supersonic flow can be determined from the equation of the law of inversion of influences. This equation is obtained from a joint solution of a system of four equations-continuity, energy in thermal form, energy in mechanical form (Bernoulli's equations), and states recorded in a differential form:

$$\frac{d\rho}{\rho} + \frac{dw}{w} + \frac{ds}{s} = 0; \tag{1}$$

$$dq_{н} - dl_{тех} = c_p dT + wdw; \tag{2}$$

$$-\frac{dp}{\rho} = wdw + dl_{тех} + dl_{тр}; \tag{3}$$

$$\frac{dp}{\rho} = RdT + RT\frac{d\rho}{\rho}; \quad (4)$$

where w, p, T и ρ – velocity, pressure, temperature and gas density, R is the specific gas constant, s is the cross-sectional area of the channel, $l_{тр}$ is the hydraulic loss (work of frictional forces associated with the viscosity of the gas), c_p is the specific heat at constant pressure.

Substituting expressions 1-3 into equations 4 and carrying out the transformations, we obtain the equation of the inversion law:

$$(M^2 - 1)\frac{dw}{w} = \frac{ds}{s} - \frac{dG}{G} - \frac{k-1}{a^2}dq - \frac{1}{a^2}dl_{тех} - \frac{k}{a^2}dl_{тр} \quad (5)$$

In this equation: M is the Mach number, a is the speed of sound, k is the exponent adiabats.

From equation 5 follows:

- any physical impact of the same sign in the opposite way affects subsonic and supersonic gas flows;

- transition through the speed of sound with the help of one-sided action is not possible. This phenomenon is called the current crisis;

- transition through the speed of sound is possible only if in the critical section the sign of influence is changed to the opposite one.

In order to realize the thermal effect on the flow, it is necessary to set the temperature in the Thermal tab in the Thermal tab at the setting of the boundary conditions in clause 3.10 in the boundary condition of the wall (Figure 4.41). Next, you need to make the settings in accordance with paragraphs 3.11-3.16 and start the calculation. Analogous calculations must be made at a wall temperature of 400, 600 and 900 K. Upon completion of the calculation, it is necessary to visualize the distribution fields of static pressure, total pressure and temperature (Figure 42-52).

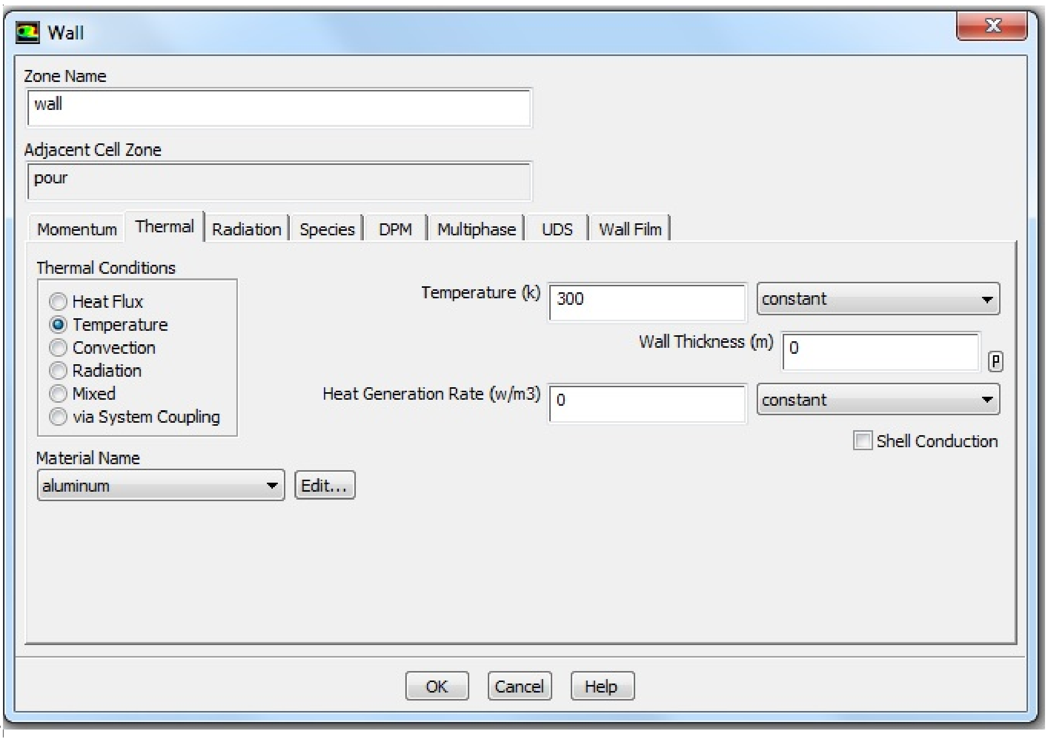


Figure 41 – Wall Menu

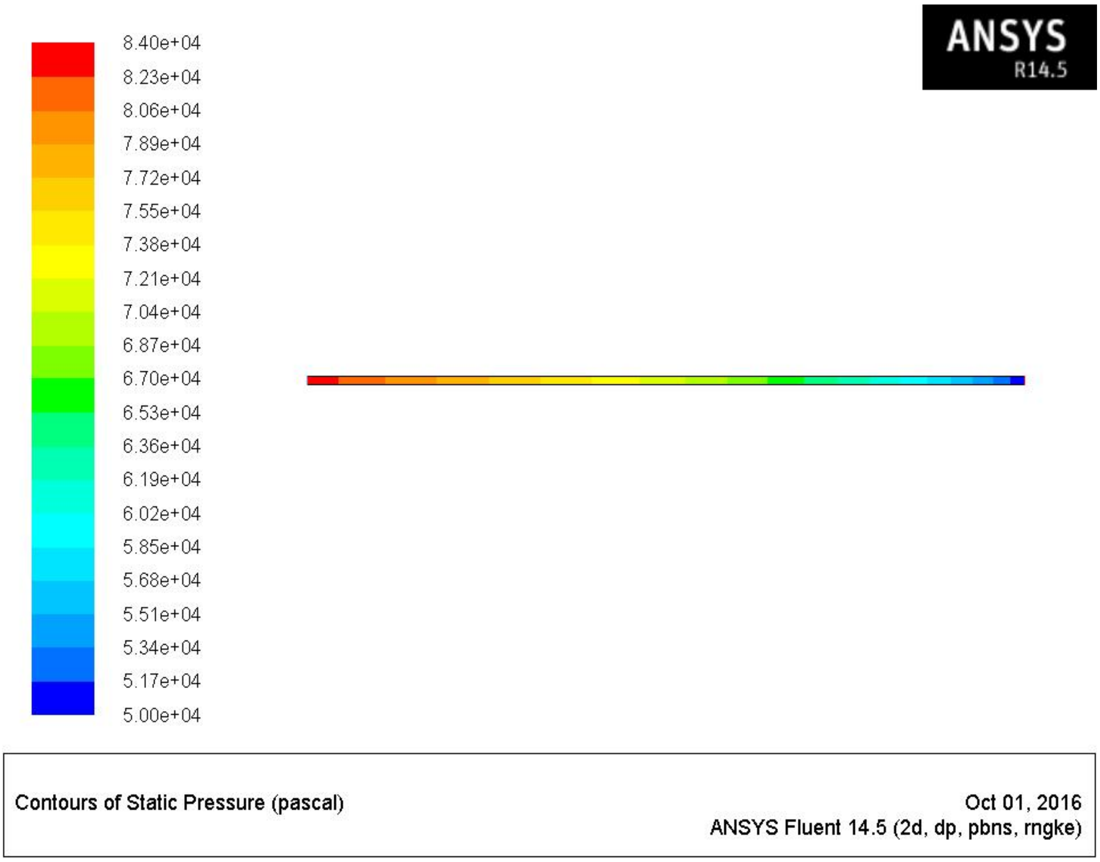


Figure 42 – Fields of distribution of static pressure in the channel at temperature 300 K walls

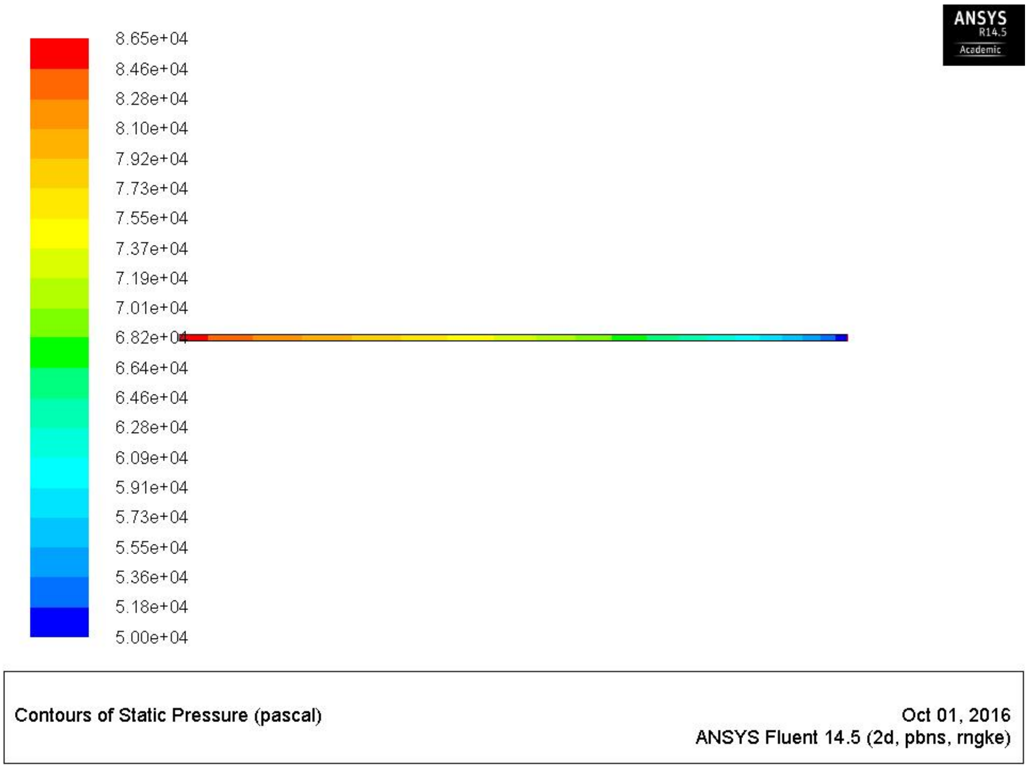


Figure 43 – Fields of distribution of static pressure in the channel at temperature walls 400 K

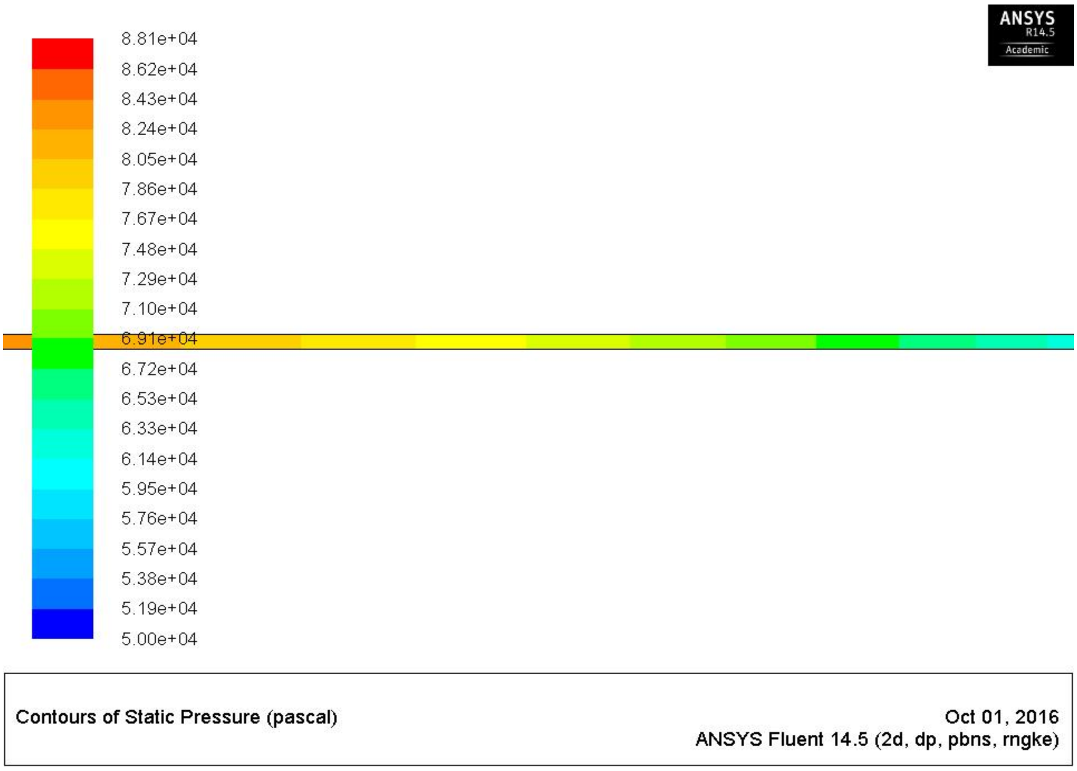


Figure 44 – Fields of distribution of static pressure in the channel at temperature walls 600 K

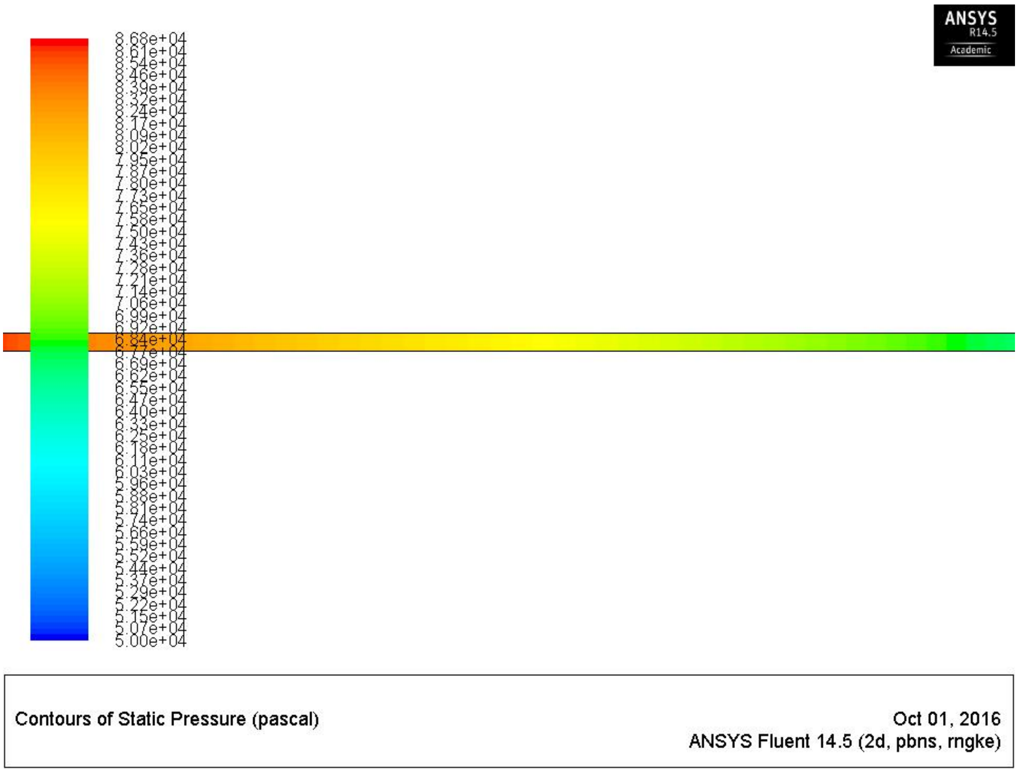


Figure 45 – Fields of distribution of static pressure in the channel at temperature walls 900 K

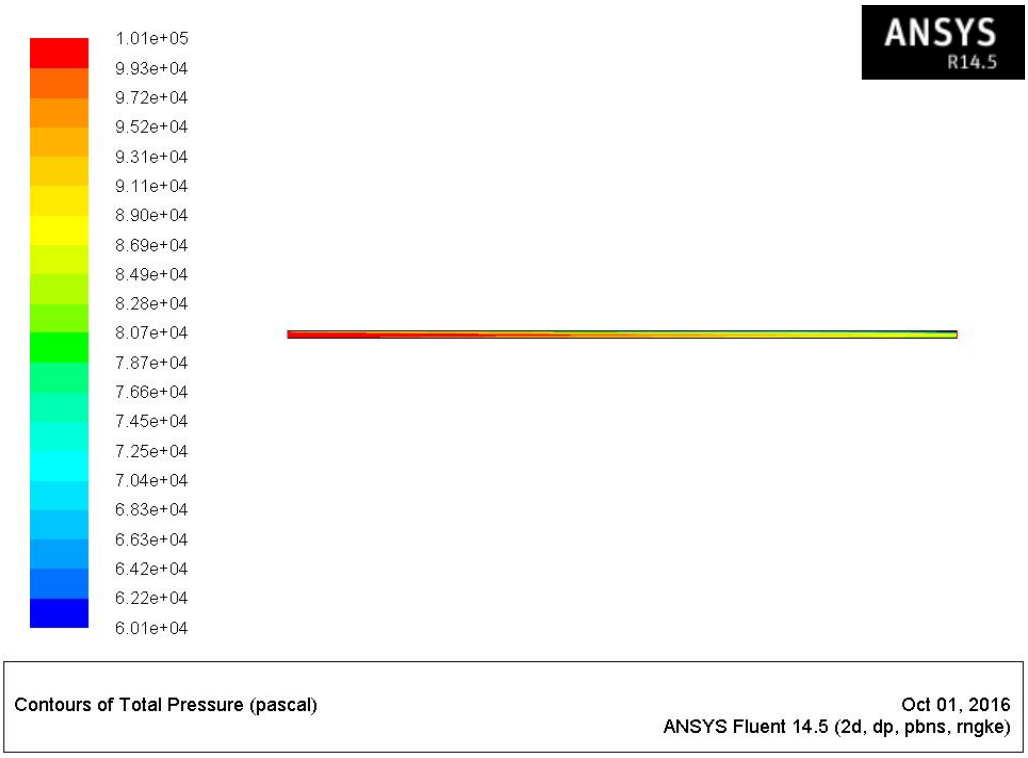


Figure 46 – Fields of distribution of the total pressure in the channel at a wall temperature of 300 K

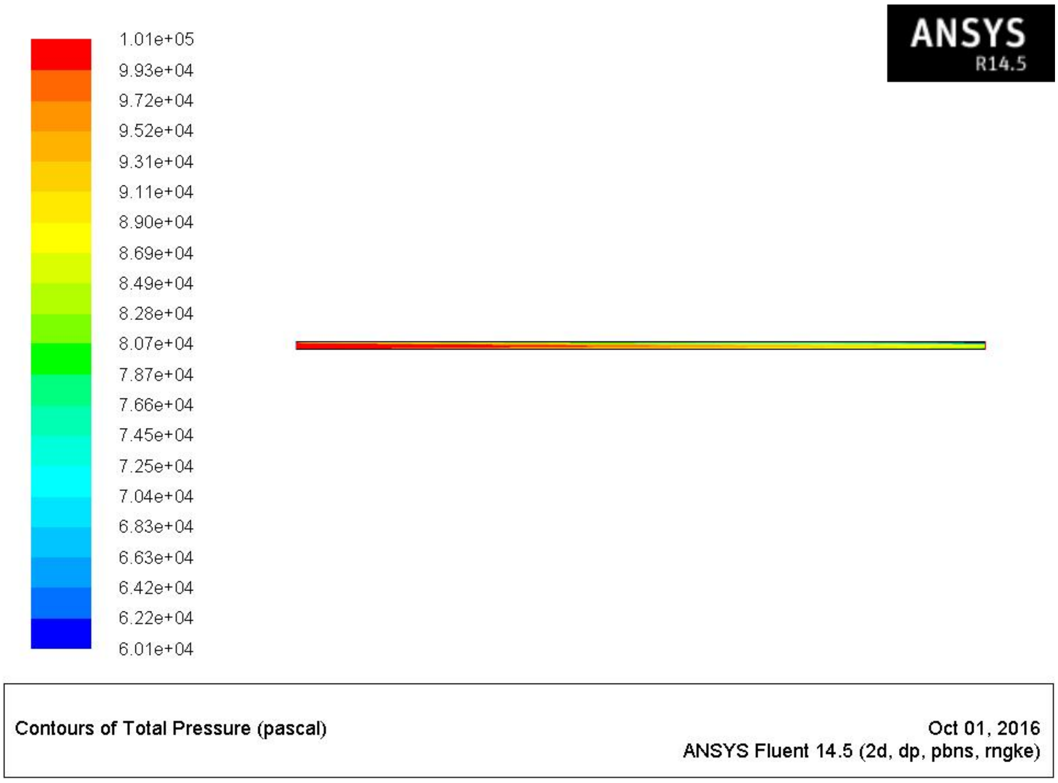


Рисунок 47 – Поля распределения полного давления в канале при температуре стенки 400 К

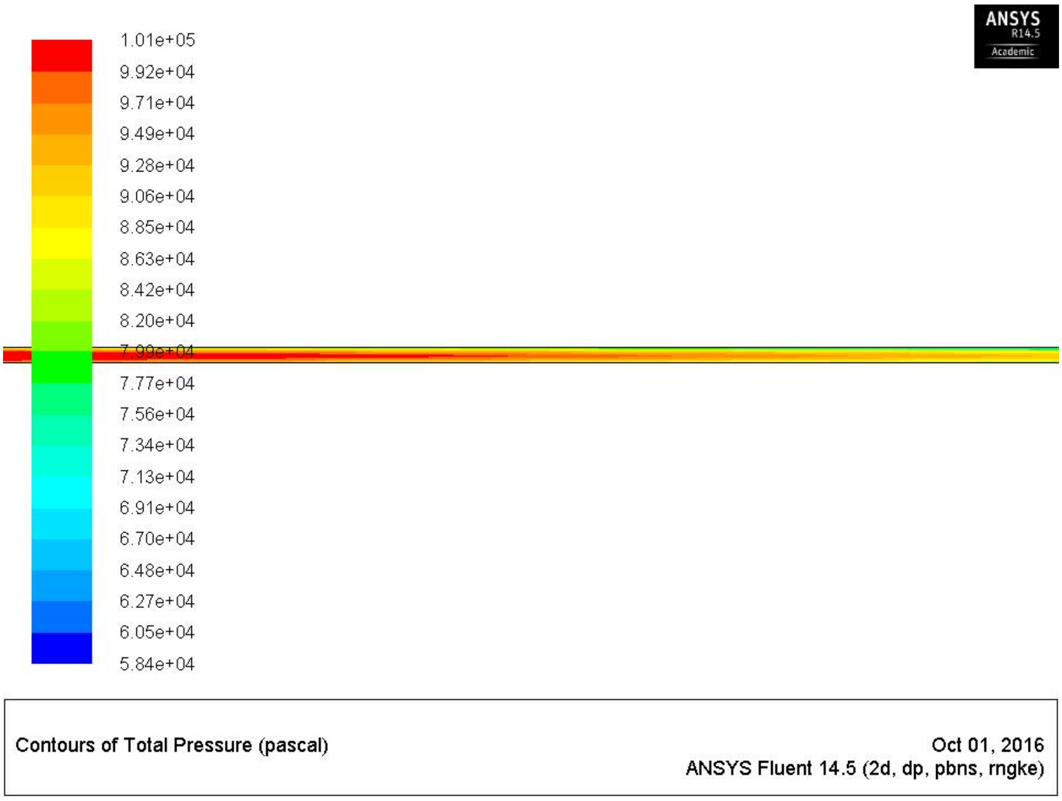


Figure 47 – Fields of distribution of the total pressure in the channel at a wall temperature of 400 K

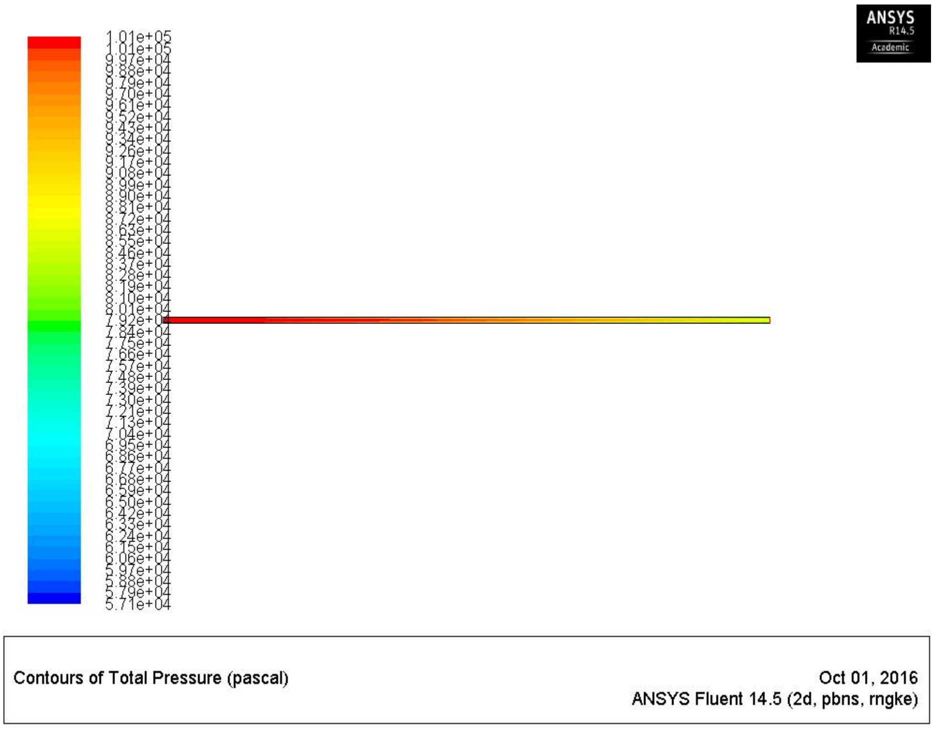


Figure 49 – Fields of distribution of the total pressure in the channel at a wall temperature of 900 K

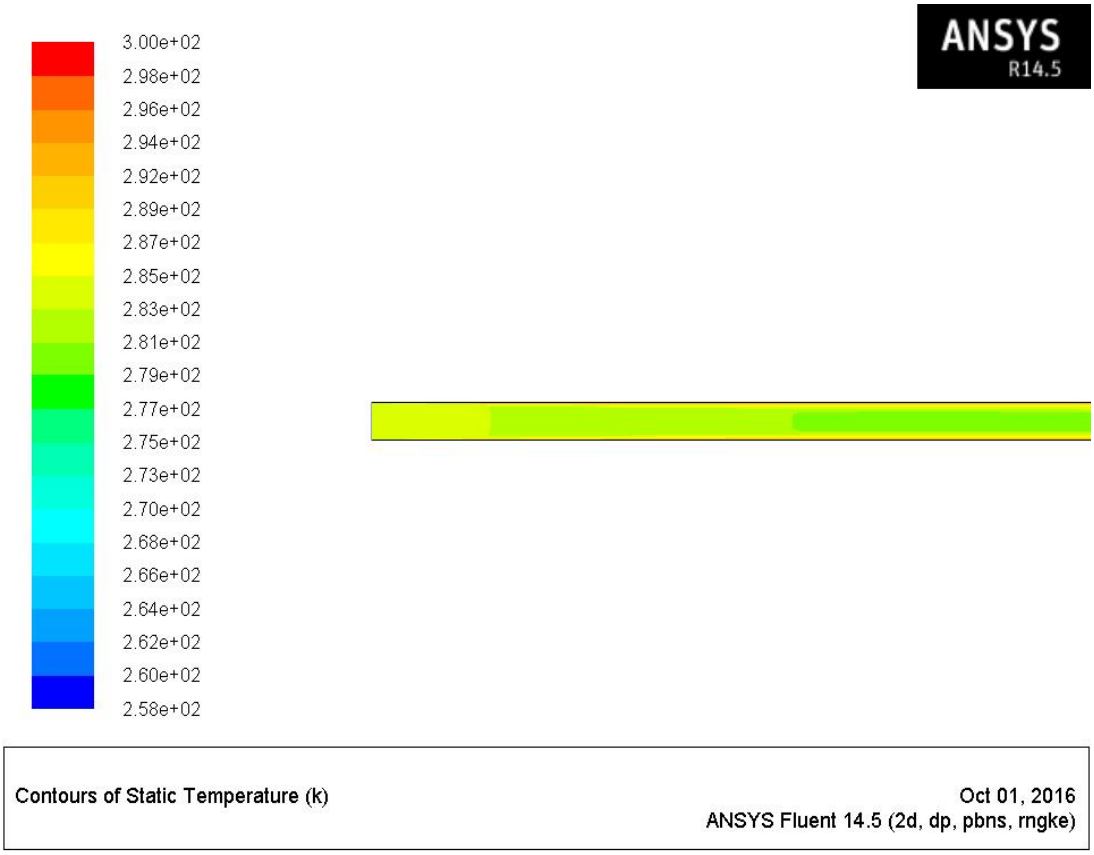


Figure 50 – Temperature distribution fields in the channel at a wall temperature of 300 K

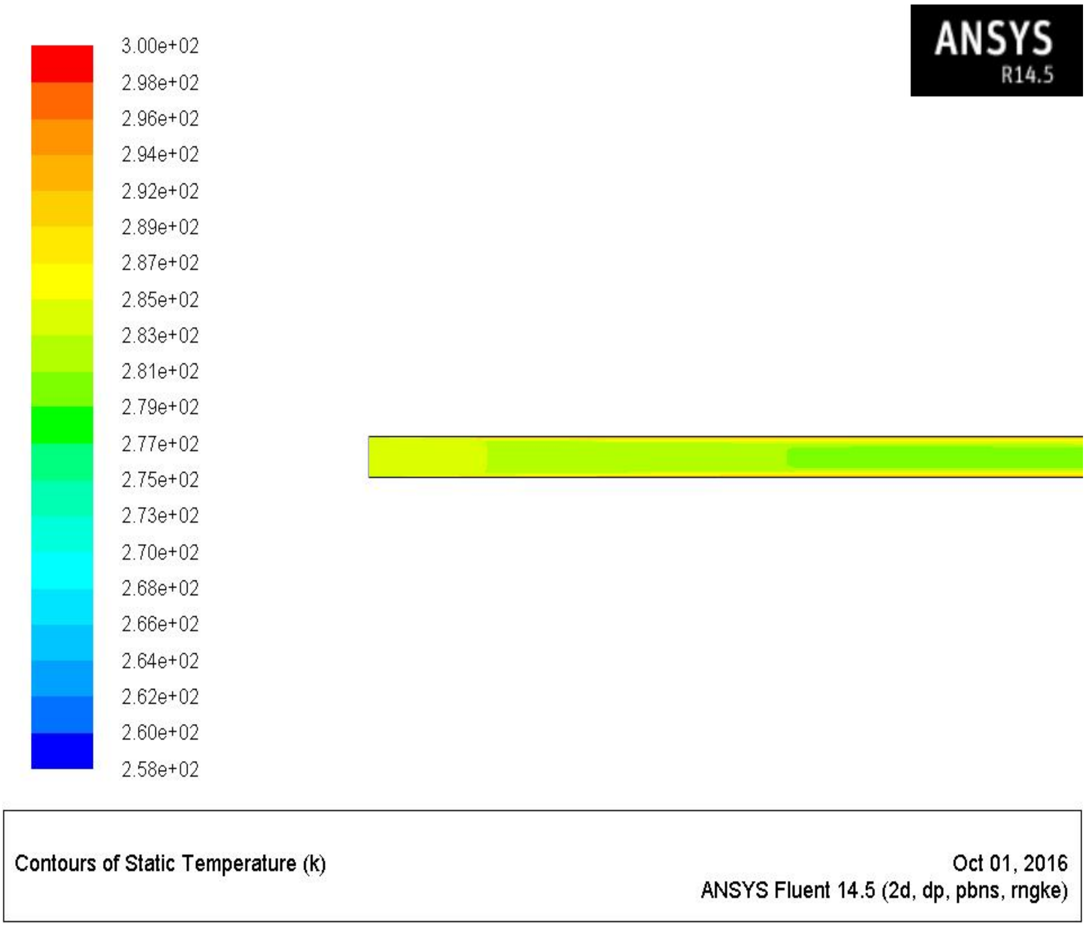


Figure 51 – Fields of temperature distribution in the channel at a wall temperature of 400 K

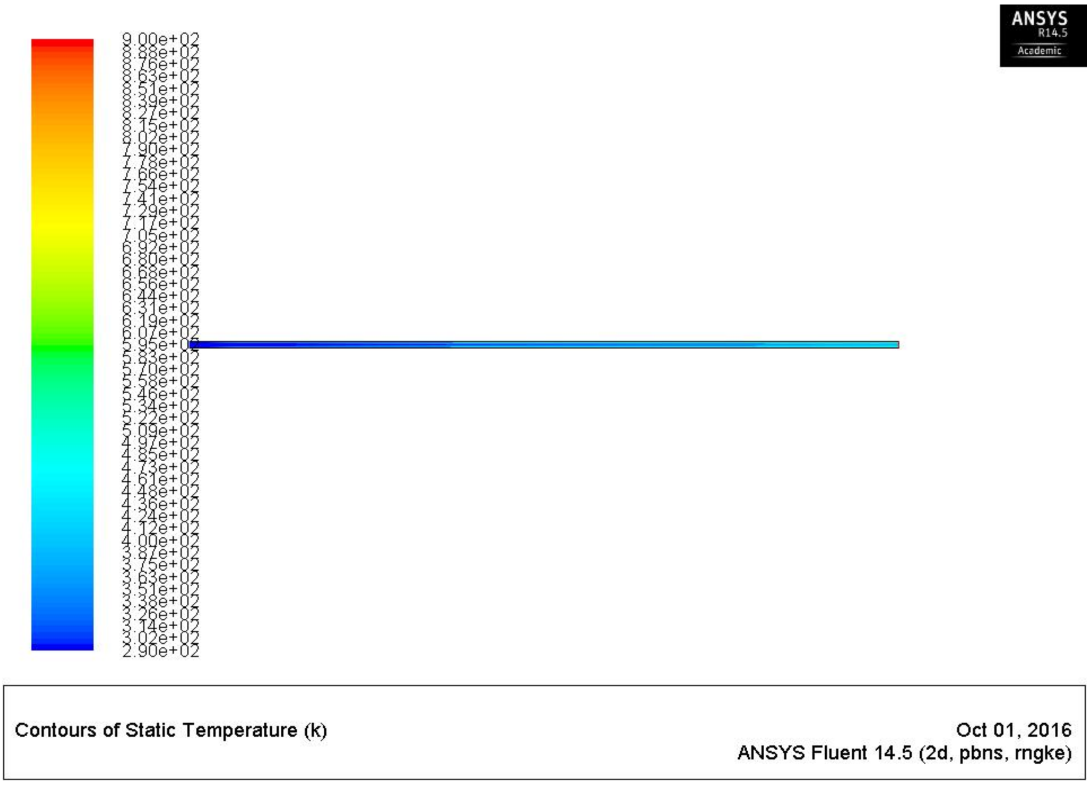


Figure 52 – Fields of temperature distribution in the channel at a wall temperature of 900 K

Also, similar to paragraph 3.19, it is necessary to plot the changes in the parameters of static pressure, total pressure, static temperature, total temperature and velocity along the channel length (Figure 43-57).

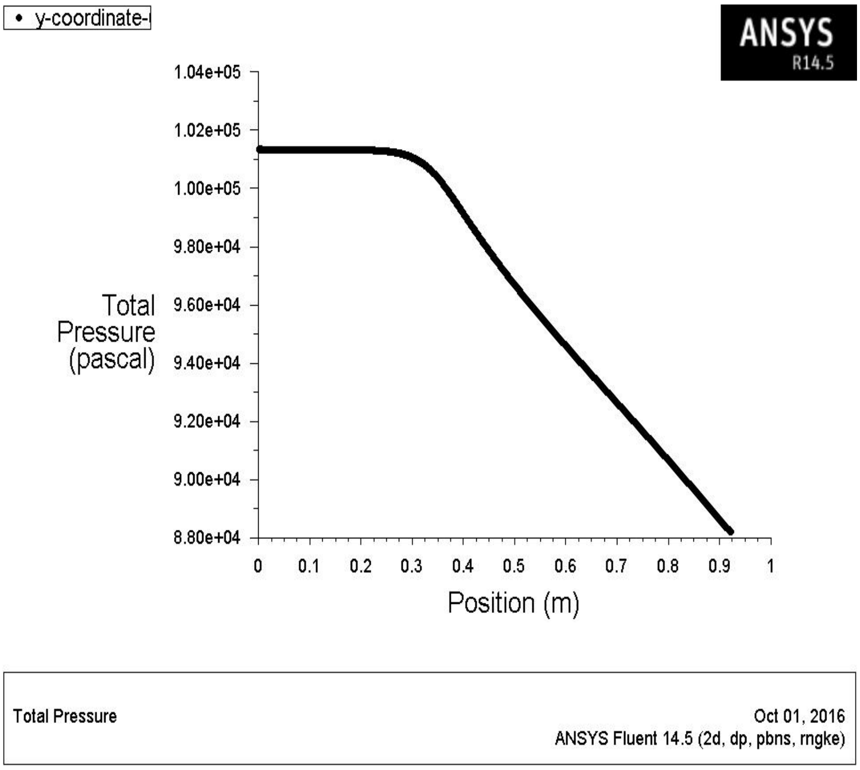


Figure 43 – Graph of the distribution of the total pressure along the channel length at a wall temperature of 300 K

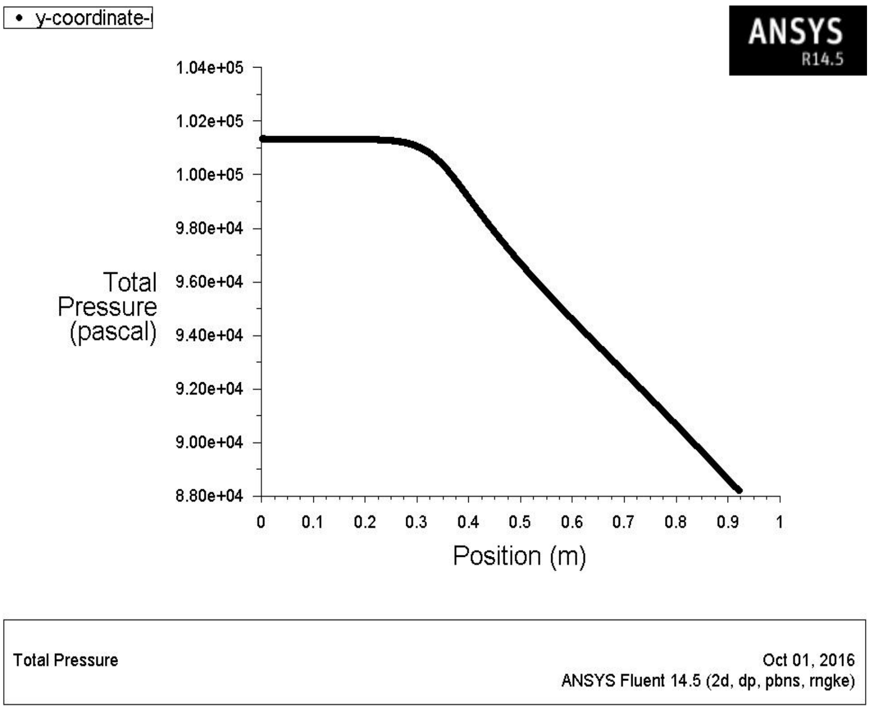


Figure 44 – Graph of the distribution of total pressure along the length of the channel at a wall temperature of 400 K

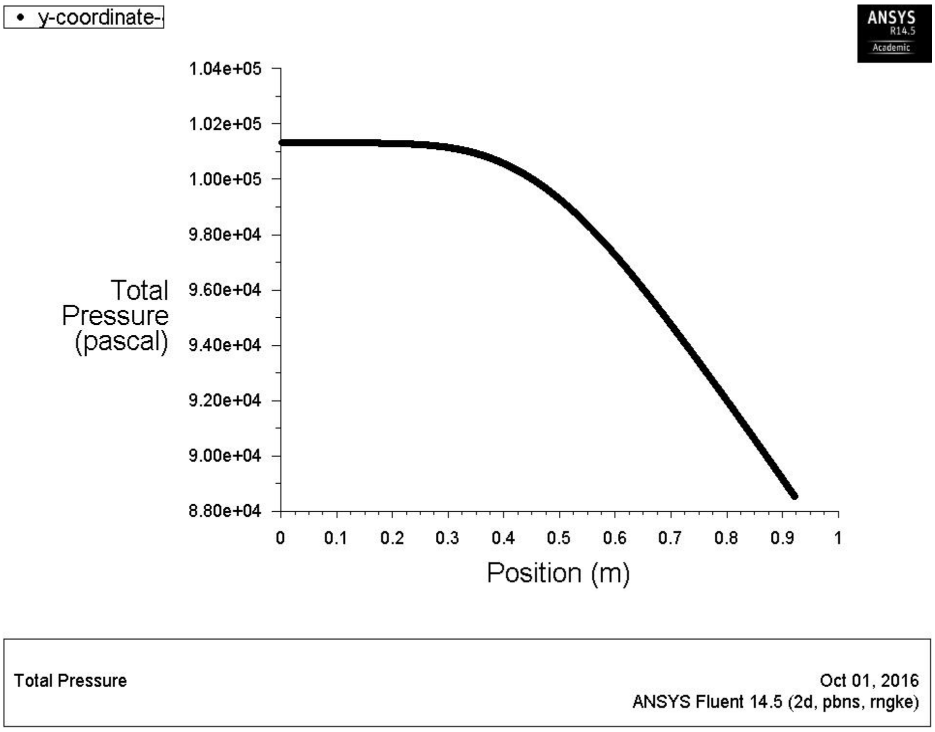


Figure 45 – Graph of the distribution of the total pressure along the length of the channel at a wall temperature of 900 K

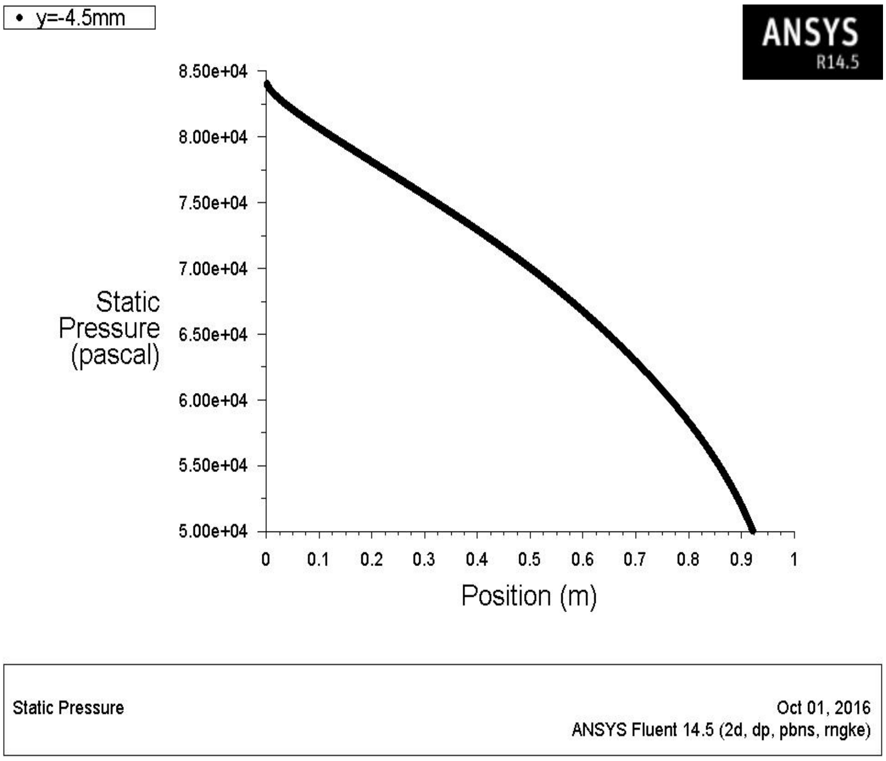


Figure 46 – Graph of the distribution of static pressure along the length of the channel at a wall temperature of 300 K

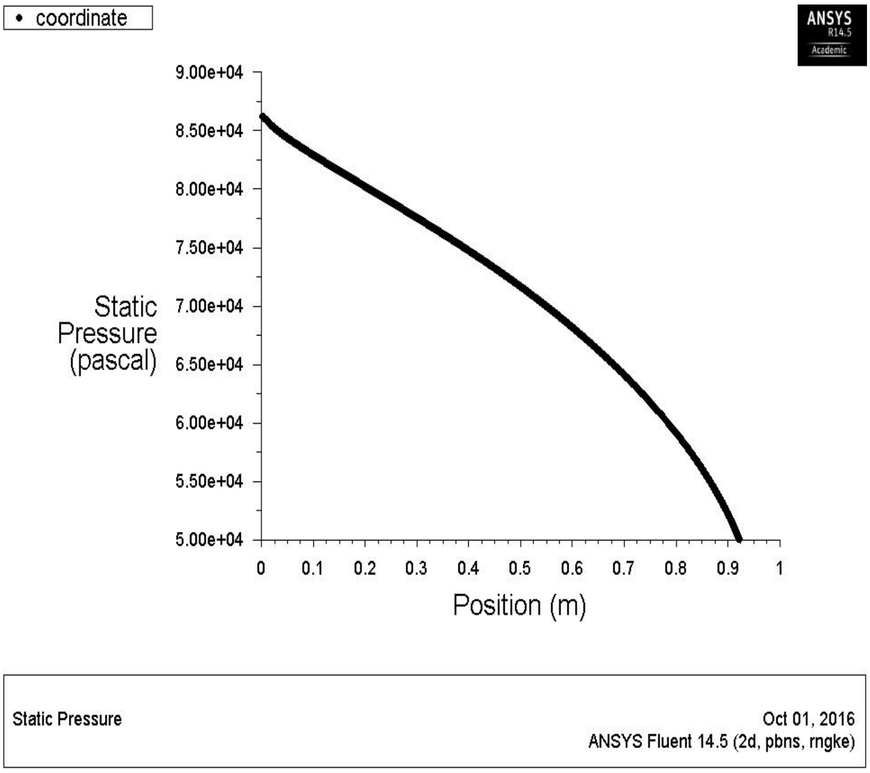


Figure 47 – Graph of the distribution of static pressure along the length of the channel at a wall temperature of 400 K

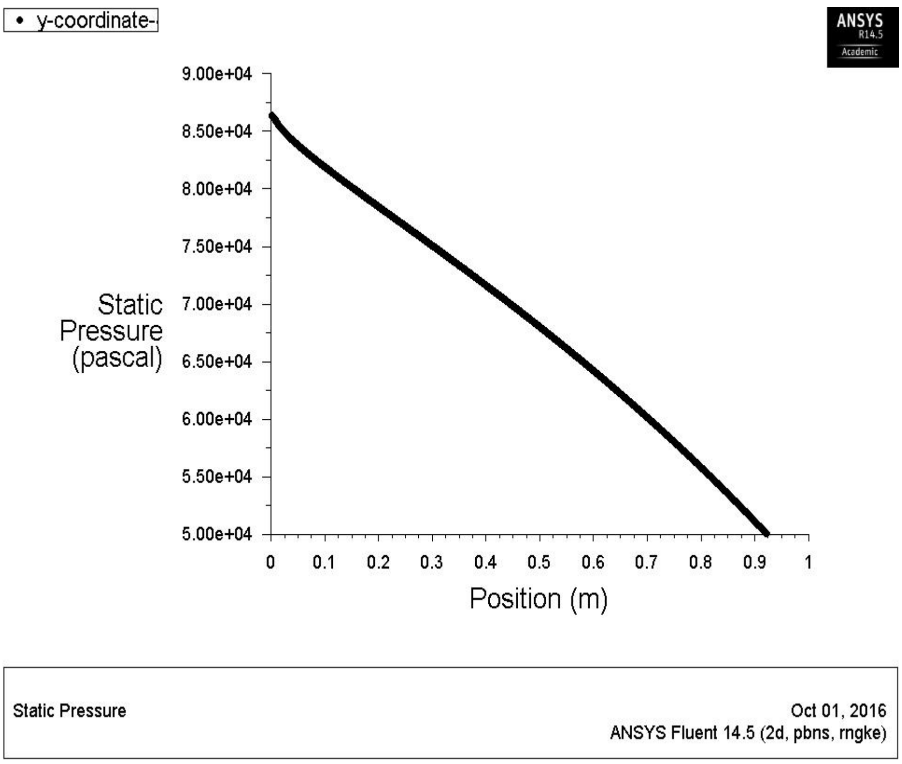


Figure 48 – Graph of the distribution of static pressure along the length of the channel at a wall temperature of 900 K

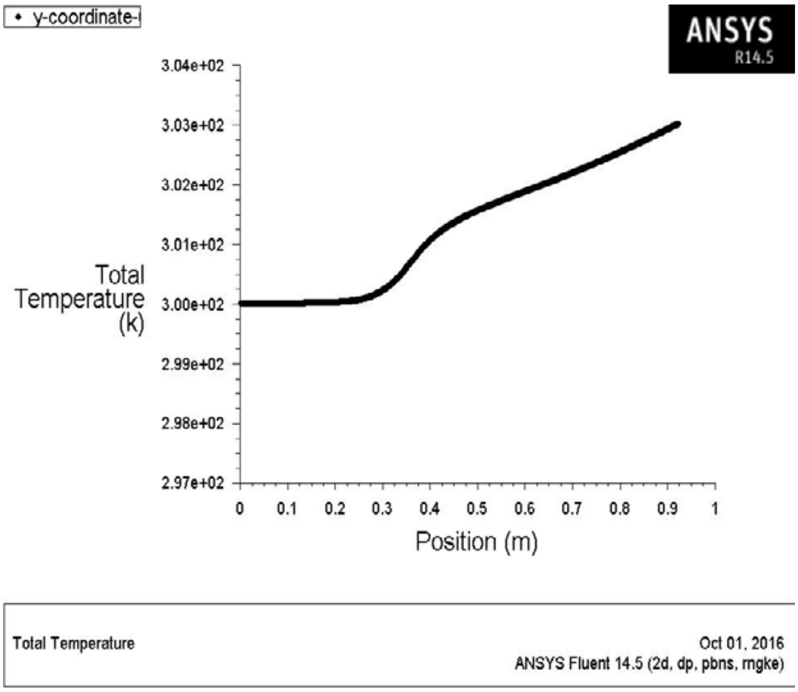


Figure 49 – Graph of the distribution of the total temperature along the channel length at a wall temperature of 300 K

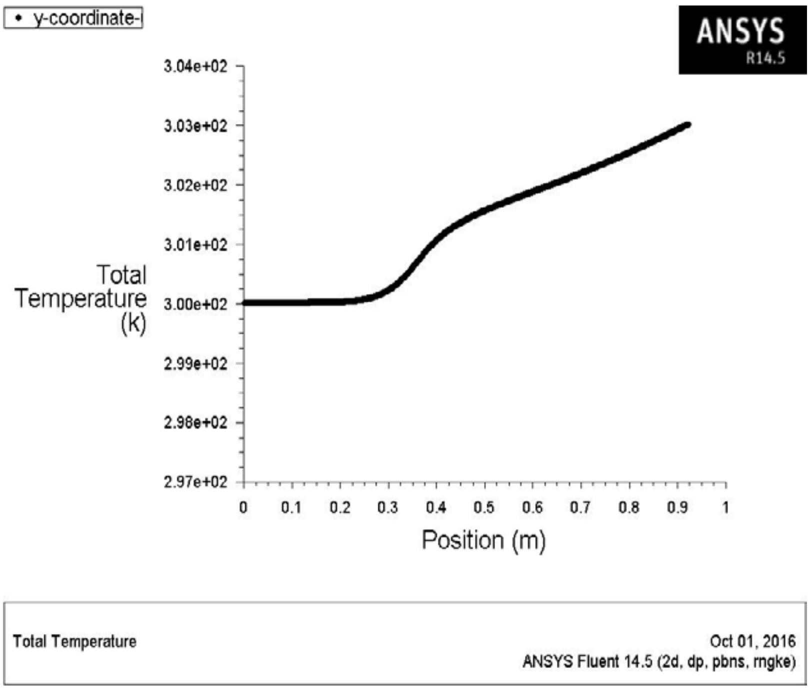


Figure 50 – Graph of the distribution of the total temperature along the channel length at a wall temperature of 400 K

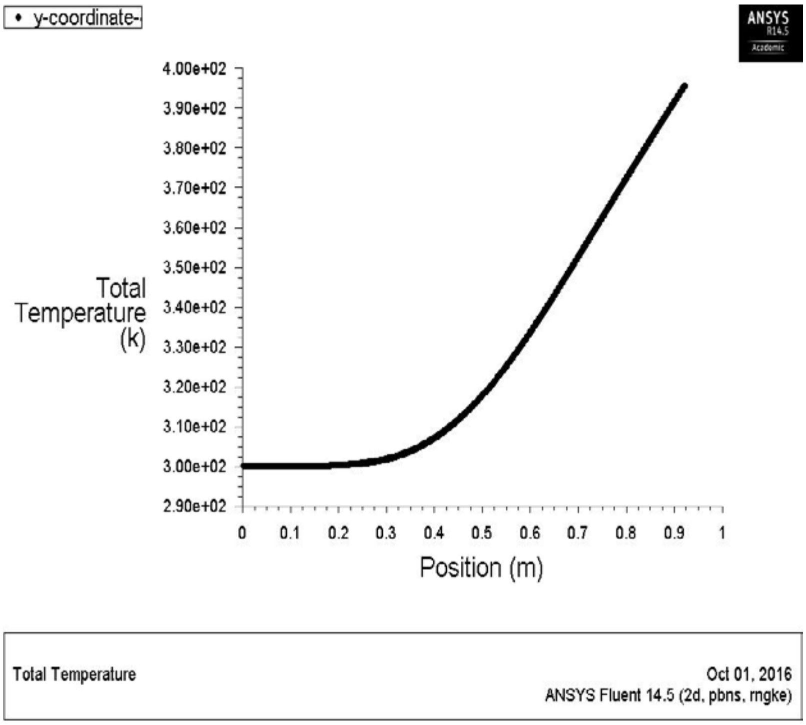


Figure 51 – Graph of the distribution of the total temperature along the channel length at a wall temperature of 900 K

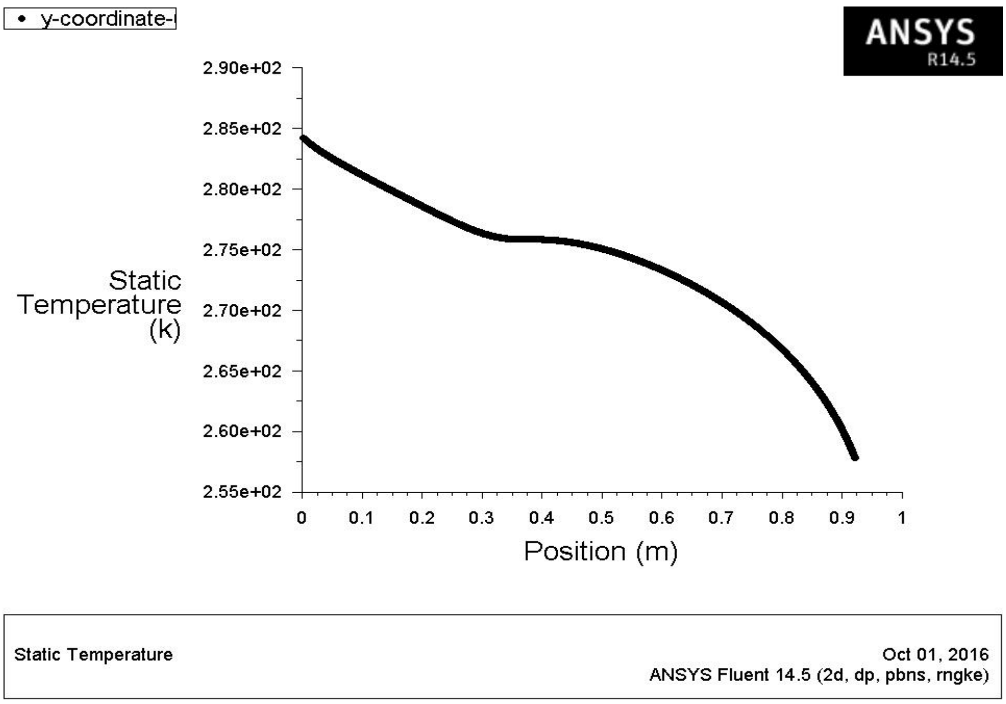


Figure 52 – Graph of the distribution of the static temperature along the channel length at a wall temperature of 300 K

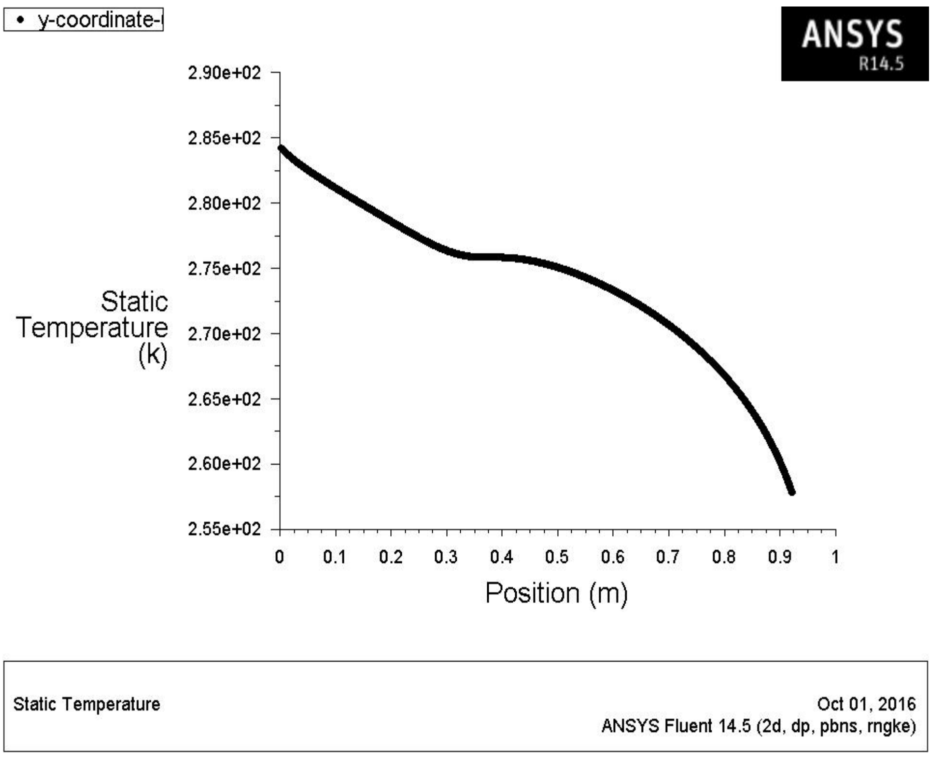


Figure 53 – Graph of the distribution of the static temperature along the channel length at a wall temperature of 400 K

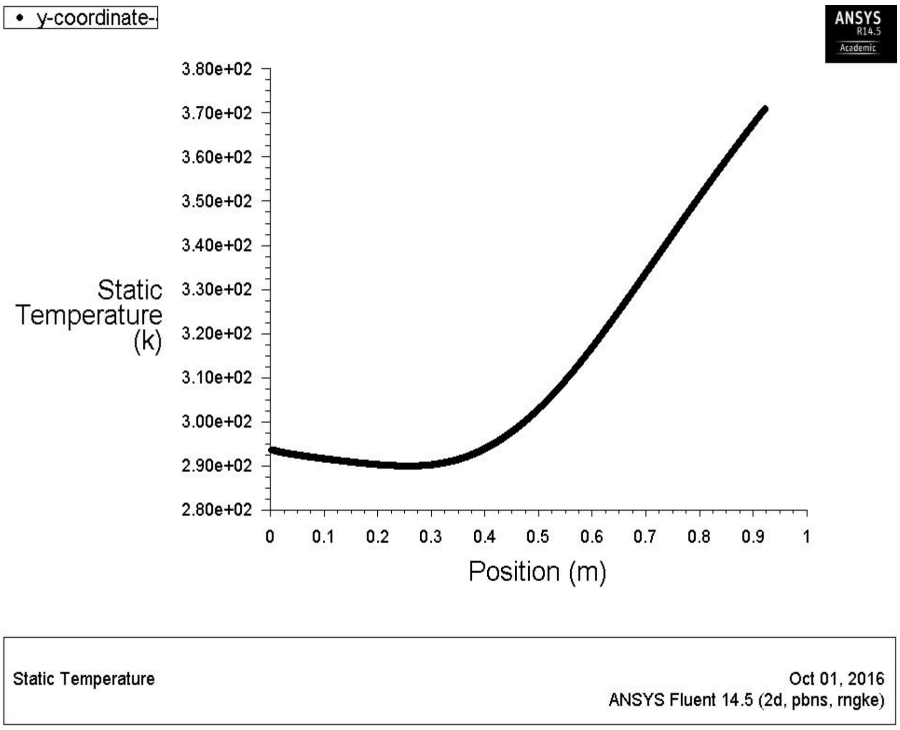


Figure 54 – Graph of the distribution of the static temperature along the channel length at a wall temperature of 900 K

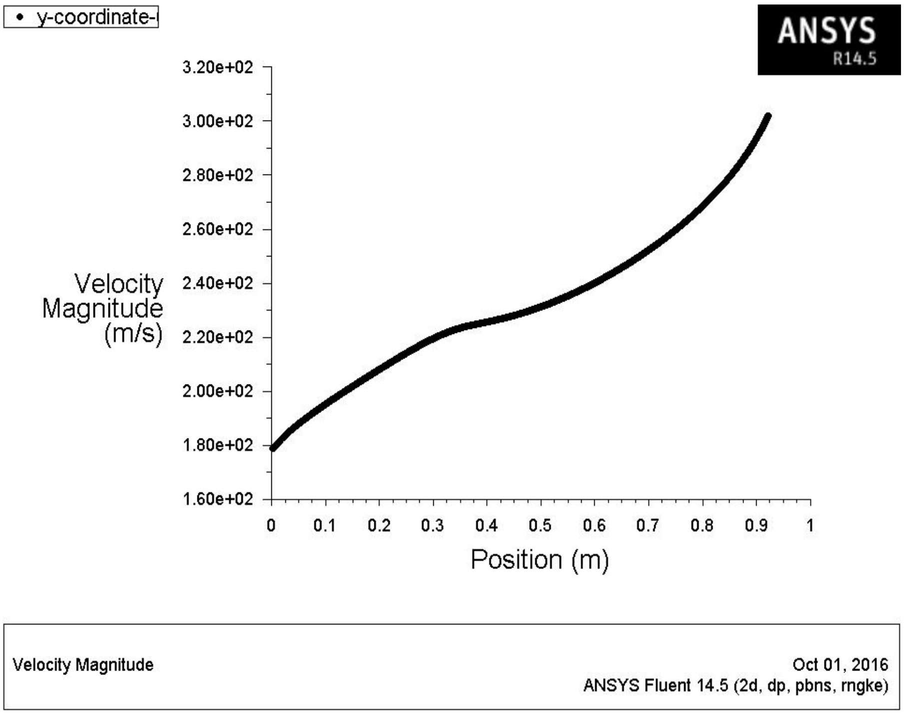


Figure 55 – Graph of the distribution of the flow velocity along the channel length at a wall temperature of 300 K

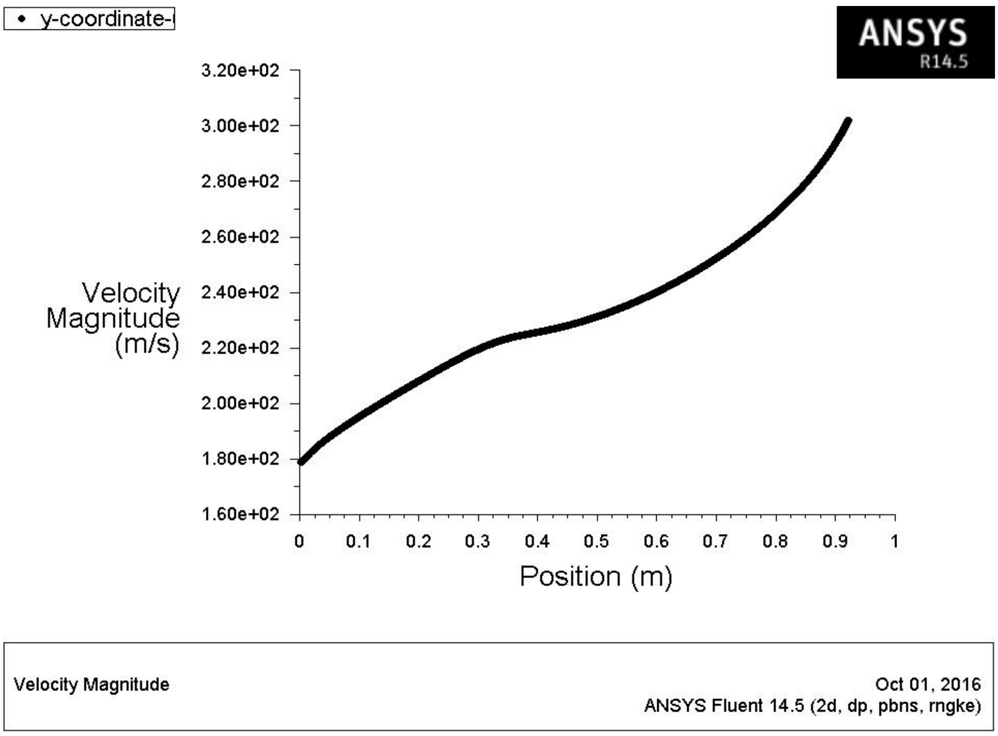


Figure 56 – Graph of the distribution of the flow velocity along the channel length at a wall temperature of 400 K

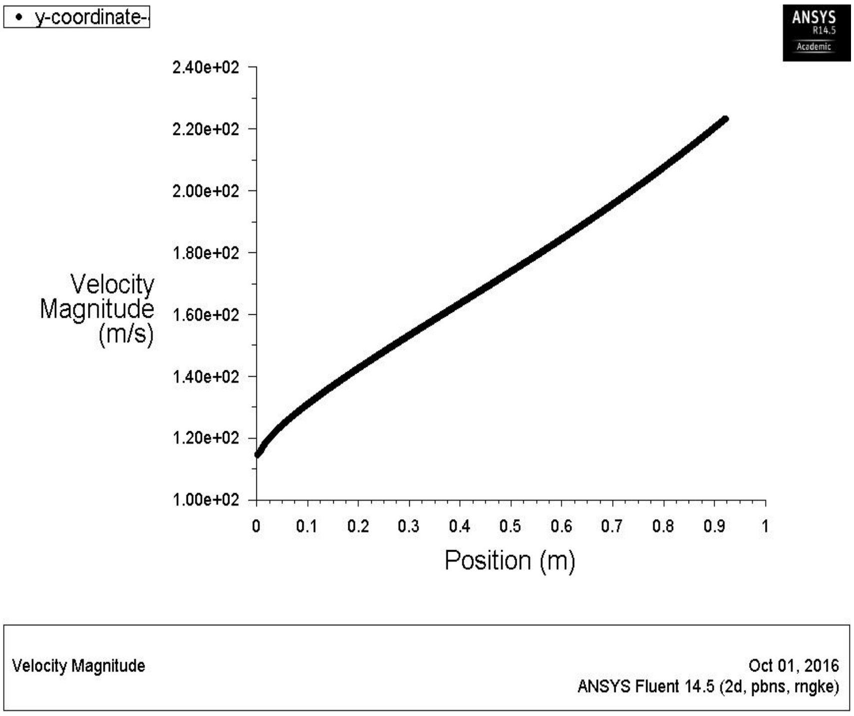


Figure 57 – Graph of the distribution of the flow velocity along the channel length at a wall temperature of 900 K

On the basis of the data obtained, it is necessary to plot the Mach number, the flow velocity and the total pressure from the wall temperature.

5. ADDITIONAL TASK №2

1. Calculate the process of flow of the working fluid in the channel without heating its wall. To do this, in the boundary condition of the wall (wall), the value of the heat flux is set to zero (Heat Flux = 0 w / m2). Then repeat all the steps from Step 18 to Step 27

2. Change the output value of pressure p0 in accordance with the experiments. If experiments were not carried out, then establish the following pressures at the exit section of the channel: p0 = 85000Pa, p0 = 45000Pa. Repeat for each version of the pressure output all steps from Step 16 to Step 27.

3. Change the working medium that is current in the channel and analyze the calculation results (Step 16 - Step 27).

4. Change the output pressure value p1 * and repeat all the steps from Step 16 to Step 27.

5. Compare the results obtained during the processing of the training experiment with the results of numerical simulation.

6. Analyze the data and draw conclusions.

LIST OF REFERENCES

1. Батурин О.В., Матвеев В.Н., Расчетное определение характеристик элементарных лопаточных венцов турбины, – Самара: Самар. гос. аэрокосм. ун-т, 2007. - 118с.
2. Определение размеров капель распыленного жидкого топлива лазерно-оптическим методом малоуглового дифракционного рассеяния света. Метод. указания / Самар. гос. аэрокосм. ун-т.; Сост. В.А.Курочкин, А.С.Наталевич, А.М.Цыганов, А.А. Диденко, Самара 2003 г. 25 с.