Министерство образования и науки Российской Федерации
федеральное государственное автономное
образовательное учреждение высшего образования
«Самарский национальный исследовательский университет
имени академика С.П. Королёва»
(Самарский университет)
Институт двигателей и энергетических установок
Кафедра теплотехники и тепловых двигателей

*А.А. ГОРШКАЛЕВ, Д. А. УГЛАНОВ, А.А. ШИМАНОВ*

# *Numerical modeling of thermal processes in heat exchangers*

**Методические указания к лабораторной работе**

Самара 2017

# Contents

# INTRODUCTION

Heat exchangers are devices designed to exchange heat between the heating and heating media. The latter is commonly called heat carriers.

The need to transfer heat from one coolant to another arises in many branches of technology: energy, chemical, metallurgical, oil, food and other industries.

In the boiler unit, the heat released during fuel combustion is transferred to water and steam, i.e. the boiler unit is a set of heat exchangers. In the nuclear power plant, the heat released by the nuclear reactor is perceived as the primary coolant, which itself becomes radioactive. The engine uses a secondary heat carrier, which receives heat from the primary heat exchanger. The process of regeneration in the gas turbine plant is carried out by transferring heat in the heat exchanger from the exhausted combustion products to the compressed air.

The wide distribution of heat exchangers causes a variety of their design.

Thermal processes occurring in heat exchangers can be very diverse: heating, cooling, evaporation, boiling, condensation, melting, solidification and more complex processes which are a combination of these. In the process of heat exchange, several heat carriers can participate: heat from one of them can be transferred to several and from several to one.

## Chapterr 1. PREPARATION OF THE CALCULATION MODEL

### 1.1 Shell modelling using ANSYS Workbench

The calculation is based on the software shell ANSYS Workbench 14.5.

First you need to run ANSYS Workbench: Start → All Programs →ANSYS 14.5→Workbench 14.5. The working window of the program opens (Figure 2.1).

As a project, you must select Fluid Flow (Fluent) (Figure 1.1).

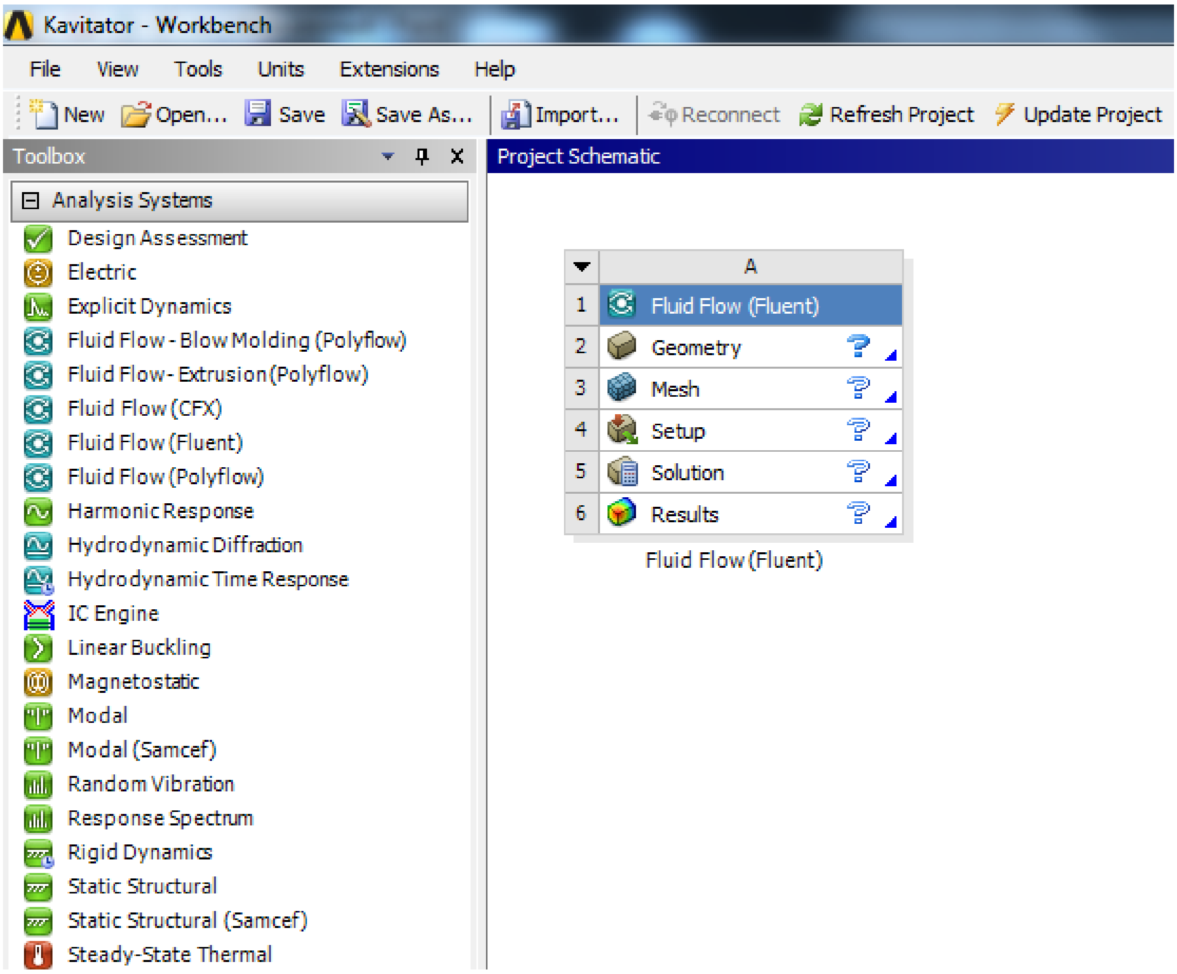


Figure 1.1 – Appearance of the Workbench work window and calculation charts for Fluid Flow (Fluent)

To import a geometric model, go to the Geometry module.

The program prompts you to select a unit of measure. You must select millimeters (Figure 1.2).

After that, the geometrical model of the flow part of the heat generator is loaded (File → Import External Geometry File). The Import icon appears in the tree. Click on it and select Generate. After that, the model should be visualized in the Graphics window (Figure 1.3).

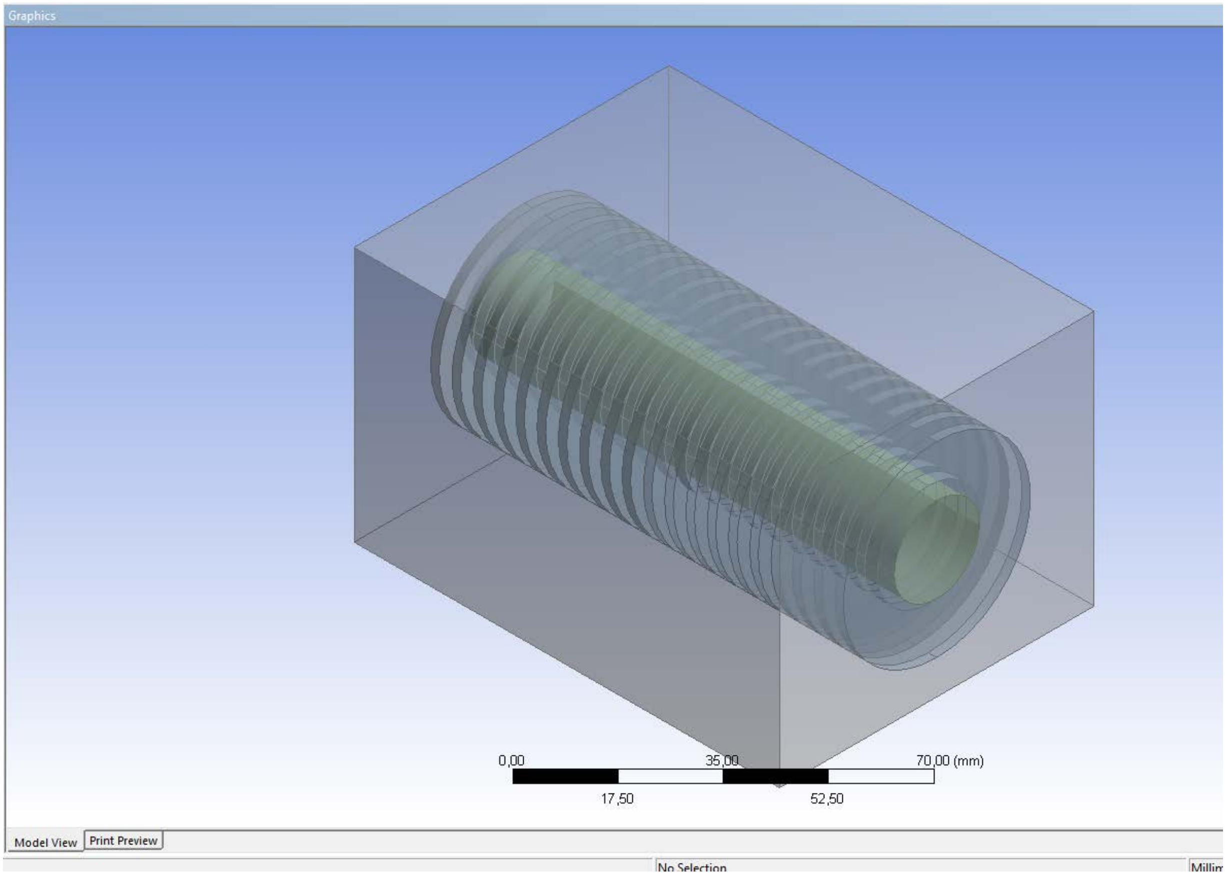


Figure 1.3 – Appearance of the heat exchanger section

The design model is a section of the heat exchanger. The heat exchanger consists of tubes with fins, inside of which water flows. Outside, the finned surface of the pipes is blown with hot air, the heat of which is transferred through the steel pipe to cold water. The calculation model must be divided into 3 main parts: gas path, tube and water. To do this, open the Parts section in the build tree, select the body and click the right button in the resulting window to select Rename (Figure 1.4).

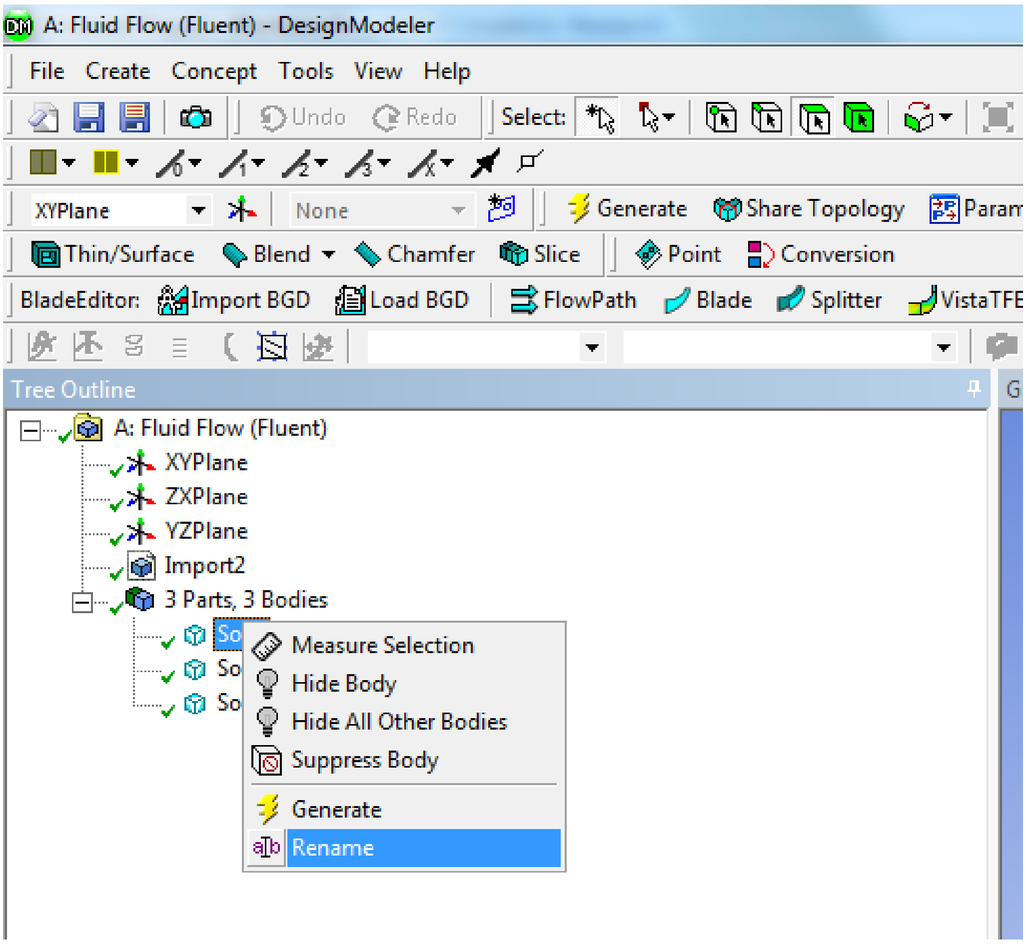


Figure 1.4 – Renaming the main parts of the model

After importing the original model by default, all 3 main parts are solid bodies. For gas (gas) and water (water) tracts, it is necessary to change the body type from solid to liquid (Figure 1.5). In order for the Details of Body menu to appear, you must select the body in the Parts tab as in the previous step.

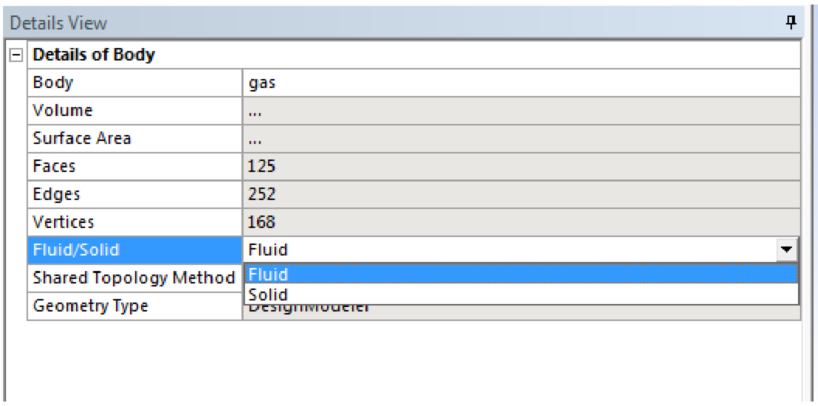


Figure 1.5 – Changing the body type

After that, you can close the Design Modeler screen.

## 1.2 Finite Element Mesh Construction

The workpiece mesh is built in the Meshing module. The heat exchanger model will be loaded automatically (Figure 1.6).

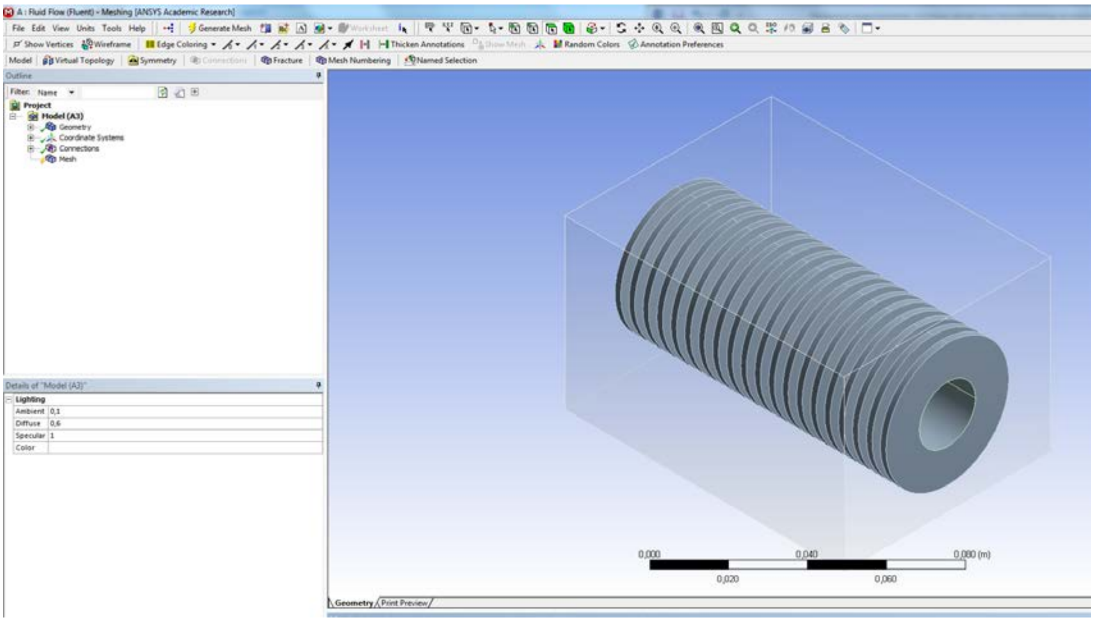

Figure 1.6 – The working window of the Meshing module and the appearance of the loaded geometric model of the heat exchanger section.

For a correct calculation, the grid at the finned surface, the pipe itself and the water path must be made smaller. To do this, in the build tree, select the Mesh tab and press the right button. In the window that appears, select Insert → Sizing (Figure 1.7).

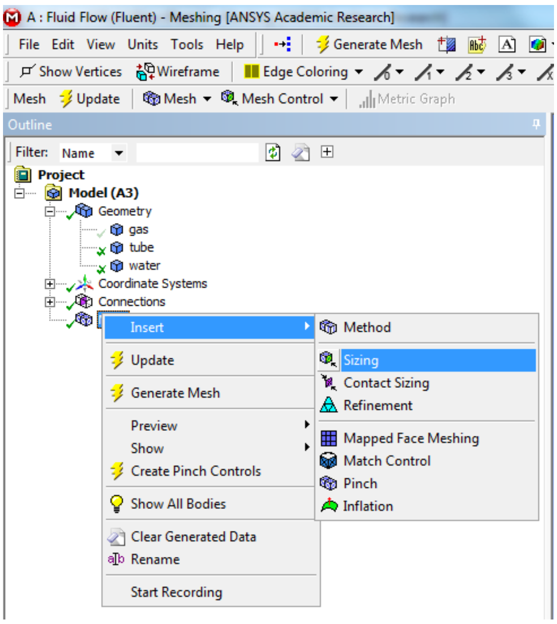


Figure 1.7 – Sizing grid size adjustment menu

The size of the element (Element Size) on the surface of the edges that come into contact with the gas and the body of the pipe itself is 0.5 mm. The size of the element for the water cavity – 1 mm.

After that, right click on Mesh and select Update. The program will begin the process of dividing the volumes by a finite element mesh. After its termination the model will look as follows (Figure 1.8):

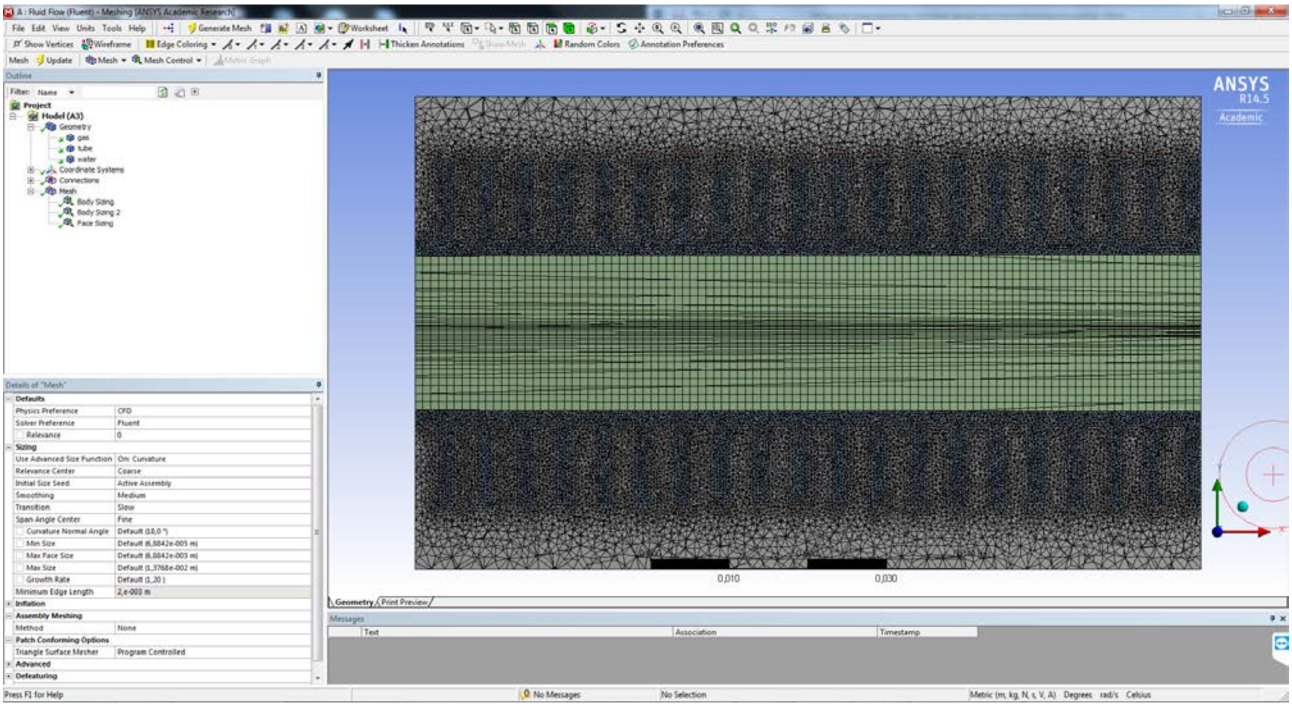


Figure 1.8 – Appearance of the mesh

Now we need to designate the surfaces for the boundary conditions for the entry and exit of water and gas. Also, to further adjust the process of transferring heat from gas to water through the pipe, it is necessary to separately identify the contact surfaces of the gas and finned tube, the inner surface of the pipe and water (4 total surfaces). To do this, select the desired surface, press the PCM and select the Named Selection item. A dialog box will open (Figure 1.9). The direction of movement of the water is controlled by the X axis, the direction of the gas flow with the Y axis. The result of the selection of the surface selection is shown in Figure 1.10. All surfaces are indicated in the tree of construction.

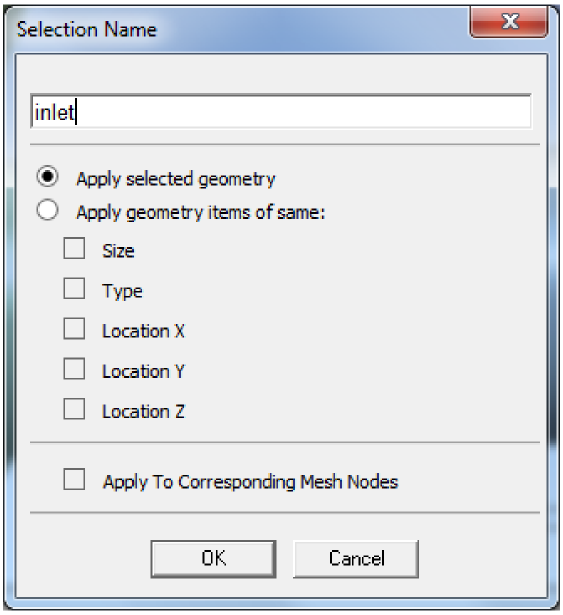


Figure 1.9 – Appearance of the surface name selection window

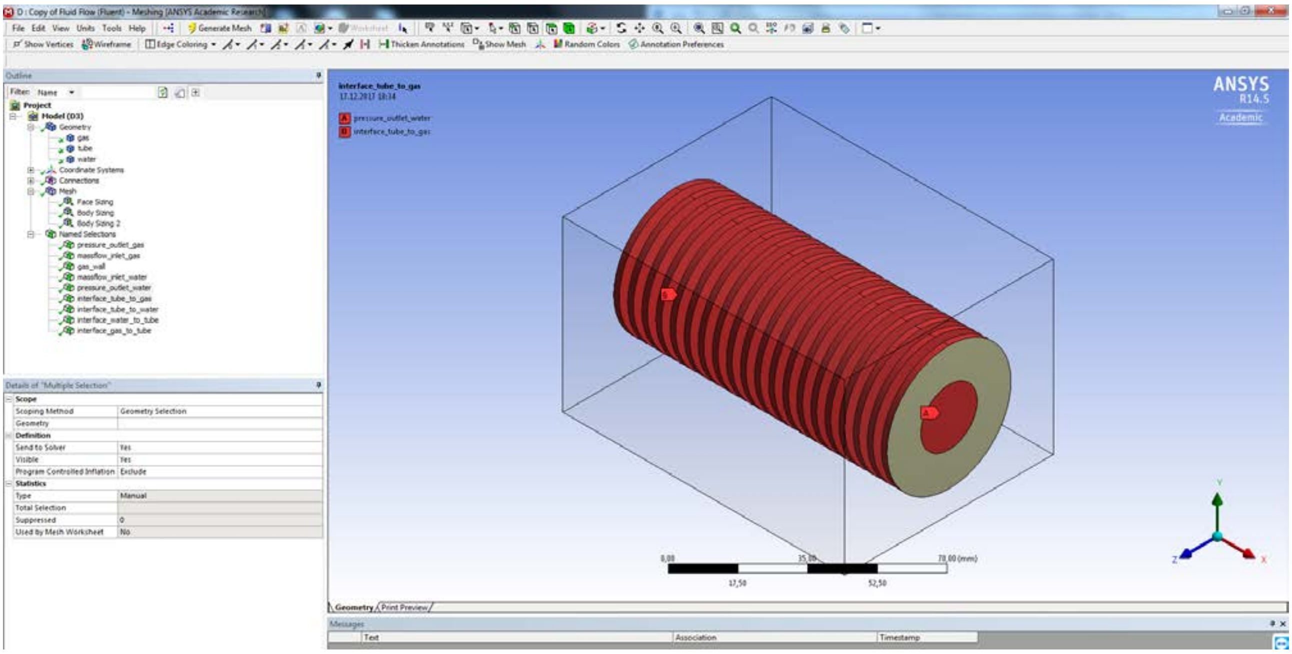


Figure 1.10 – Appearance of the Surface Name Selection Window

After that, the Meshing module can be closed.

# Chapter 2. MODELLING HEAT EXCHANGERS

## 2.1 Running the ANSYS Fluent program and its features

Further actions with the calculation model, which are performed in the ANSYS Fluent program, are the setting of boundary conditions, adjustment of the solver parameters, execution of calculations and processing of the results.

To start the program, you need to enter the module Setup in the ANSYS Workbench. The dialog window for running ANSYS Fluent will open (Figure 2.1).

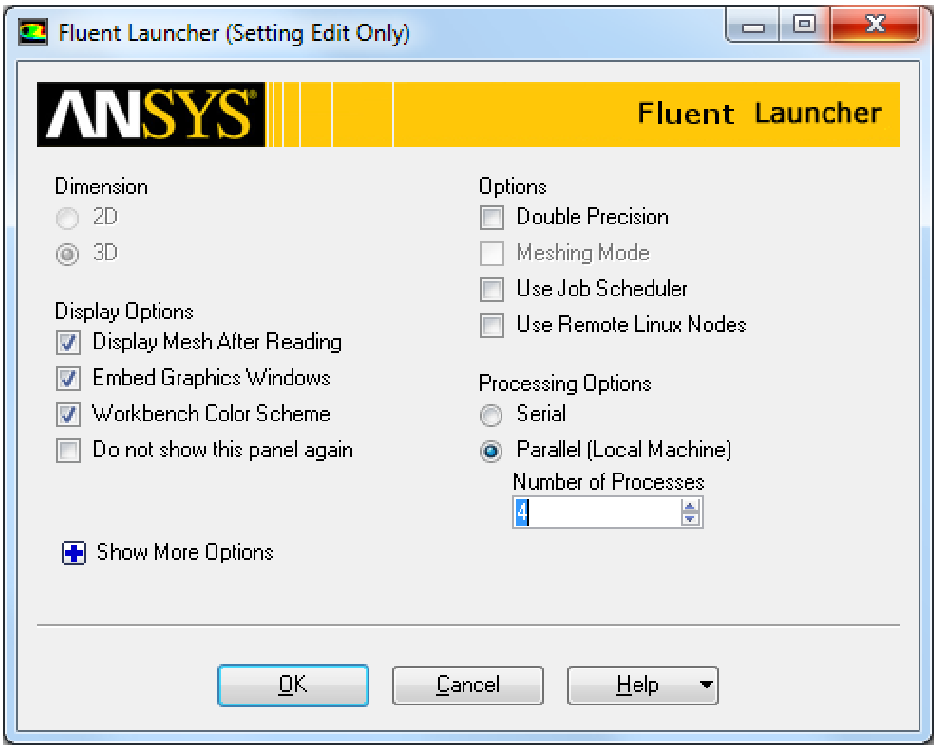


Figure 2.1 – The menu for starting the program ANSYS Fluent (Fluent Launcher)

In the case under consideration, the problem is three-dimensional. After selecting, press the OK button. As a result, the ANSYS Fluent working window appears on the computer screen (Figure 2.2).

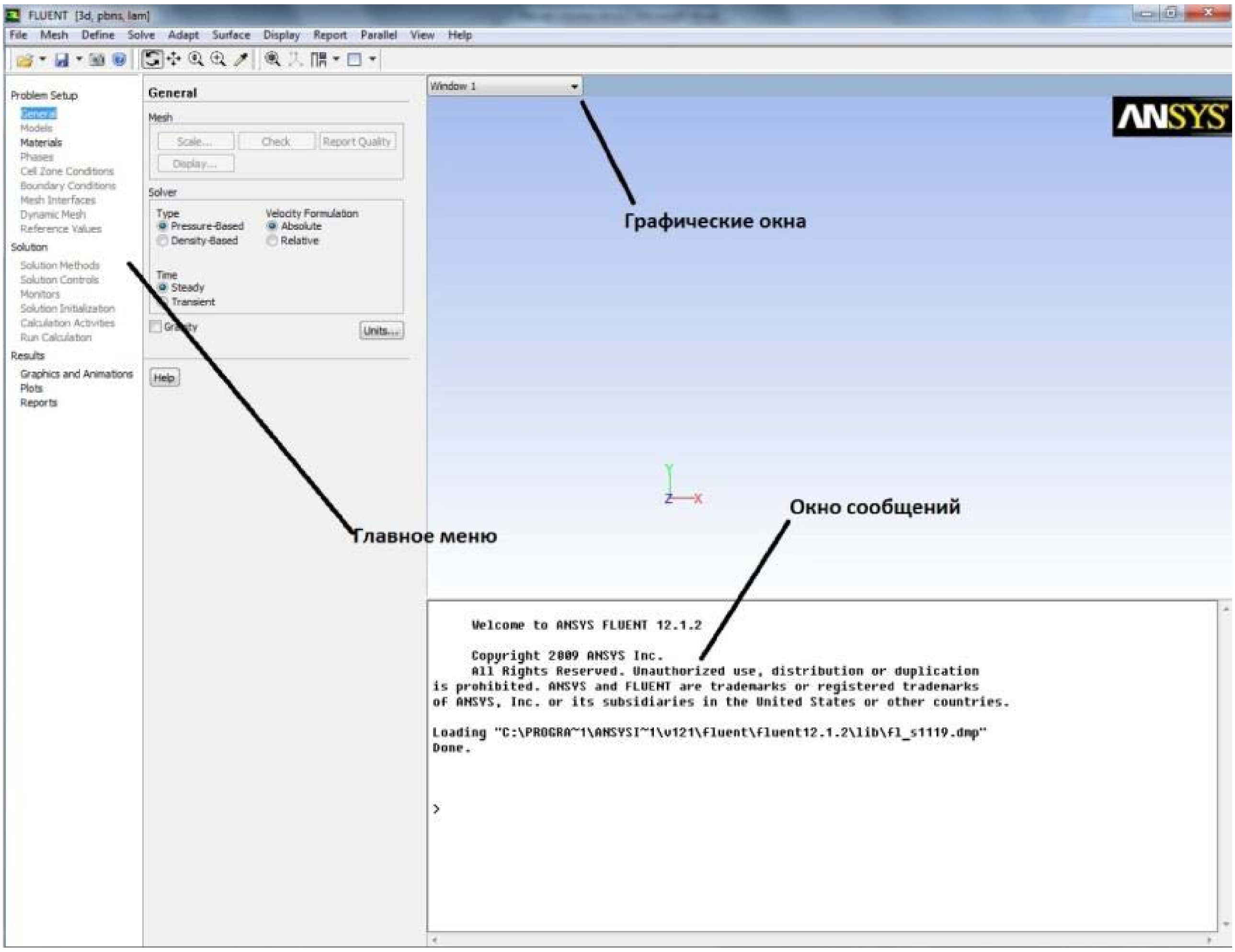


Figure 2.2 – ANSYS Fluent working window

The program window is quite simple and consists of three main parts:

- the main menu, through which access to all commands and menus of the program;

- the message window where the command line is located and the results of the commands execution are displayed;

- Graphical windows in which the results of calculations and constructions are displayed.

Downloading a Grid File

Since the ANSYS Fluent is started from the Workbench software environment, the attached grid will be loaded automatically.

Checking the finite element mesh for errors

Checking the calculated grid for errors is performed using the command:

GM: Mesh → Check

After its launch, the program will begin to check the finite element mesh (Figure 2.3), and full information about the finite element mesh will appear in the message window. If an error is found, a corresponding message will be issued. In this case, you need to return to the Meshing program, find the error and fix it.

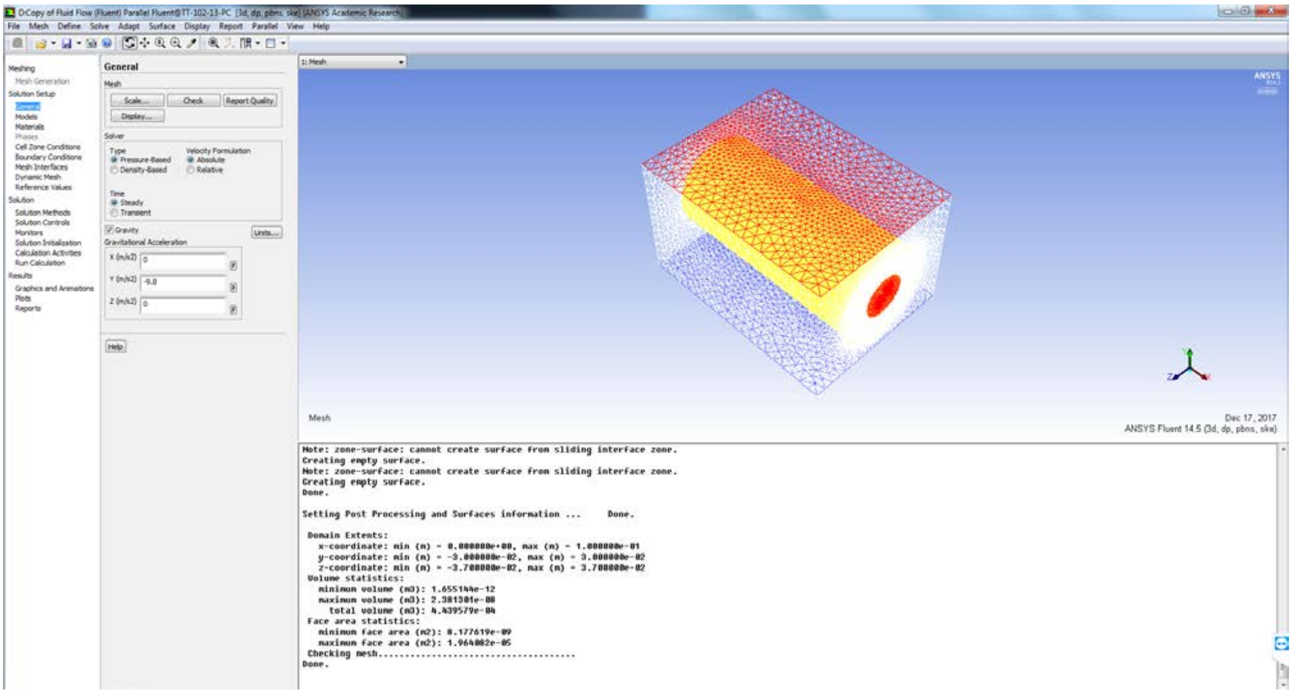

Figure 2.3 – Mesh Display

Scaling of finite element mesh

Geometric dimensions in ANSYS Fluent must be specified in meters, so in case of creating a model in millimeters, it is necessary to scale it. In our case, this step is not needed, because when you move a geometric model in ANSYS Design Modeler, the scaling is done automatically.

View Finite Element Grid

You can view the finite element mesh of the loaded model using the command:

MM: Display → Mesh

In the appeared Mesh Display menu (Figure 2.14) in the Surfaces window it is required to select all surfaces forming the grid model. It should be noted that the names in the list coincide with the names of the surfaces specified in Meshing. To view the selected grid elements, click the Display button.

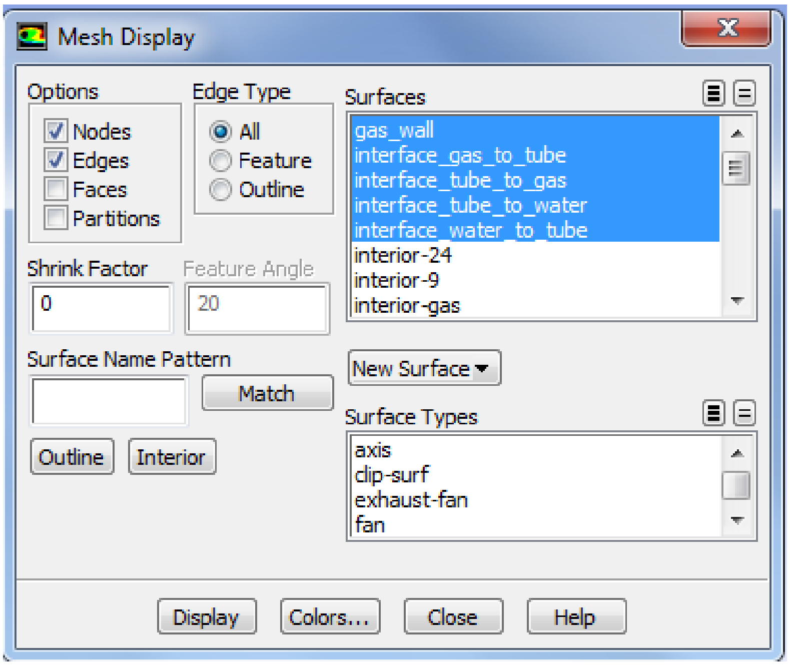


Figure 2.4 – Mesh Display

As a result of the command, a graphic window will appear in which the calculated grid or selected model elements will be displayed.

If you deselect the default in the Surfaces window, you can see only the outline of the model in the window, without a calculated grid.

## 2.2 Setting up Calculation Parameters

Setting the Solver Options

As the first action in describing the computational model, it is necessary to choose a solver that will be used to solve the problem, as well as to determine the stationarity or nonstationary of the problem. This selection is made in the Solver menu:

MM: Define

In the Solver menu (Figure 2.5), you need to pay attention to the following points. The ANSYS Fluent program allows the use of two decision algorithms: Pressure Based or Density Based. The first of these was originally developed for low-speed flows, but was subsequently modified and extended to other currents. The setup algorithm was designed to calculate high-speed trans- and supersonic flows. To solve the problem under consideration it is expedient to choose Pressure Based.

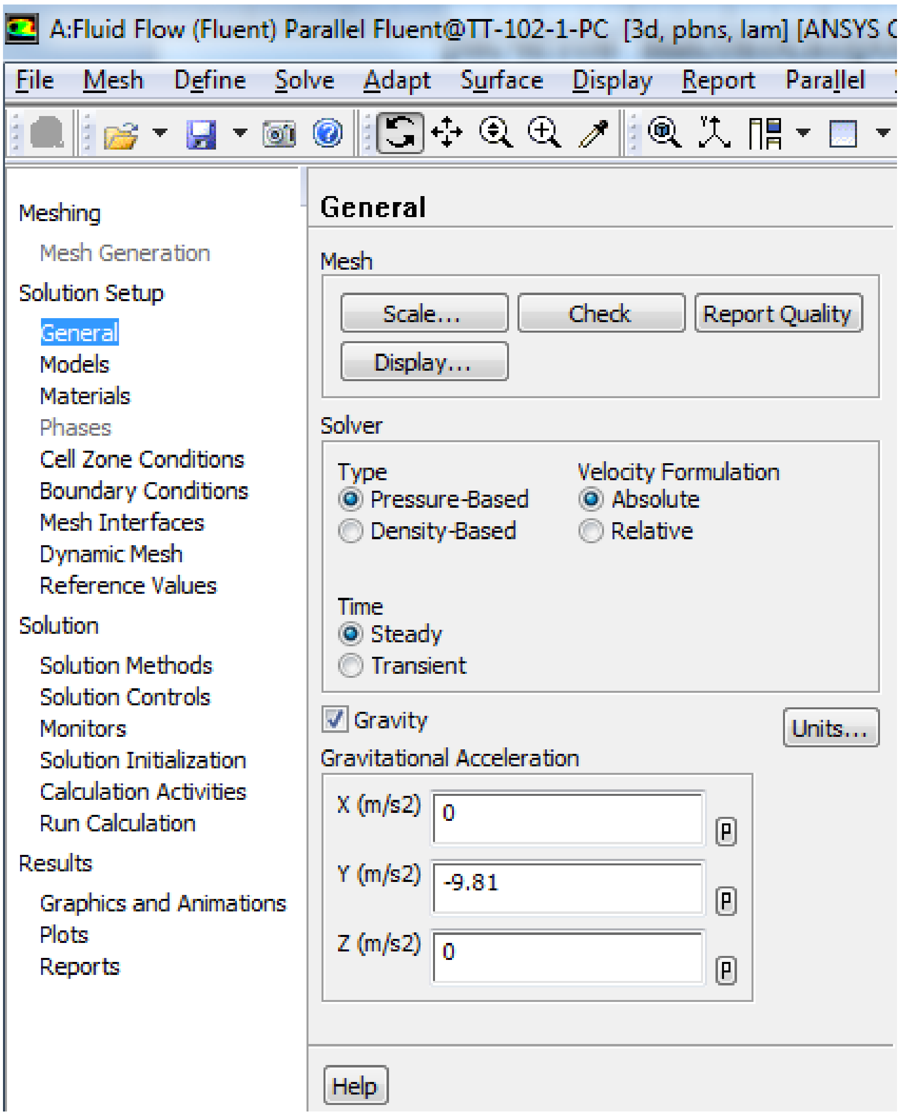


Figure 2.5 – The solver setting menu (Solver)

In the Time field, it is selected whether the solution is stationary Steady or non-stationary Transient. That is, whether the flow parameters will depend on the time or not. The problem under consideration is three-dimensional stationary. In addition, in this problem, gravity must be taken into account. To do this, select the Gravity item and specify the value of the acceleration of free fall equal to -9.81 m / s2.

Choice of the model of turbulence

The flow of a liquid is characterized by the presence of turbulence - a disordered motion of the vortex masses. In this case, transverse components are superimposed on the main velocity direction, causing strong mixing of the liquid / gas. When studying the flow in the heat generator channel, it is advisable to use the k-epsilon turbulence model, since it allows obtaining solutions with acceptable accuracy, and for this model it converges well (Figure 2.6).

MM: Define → Models → Viscous.

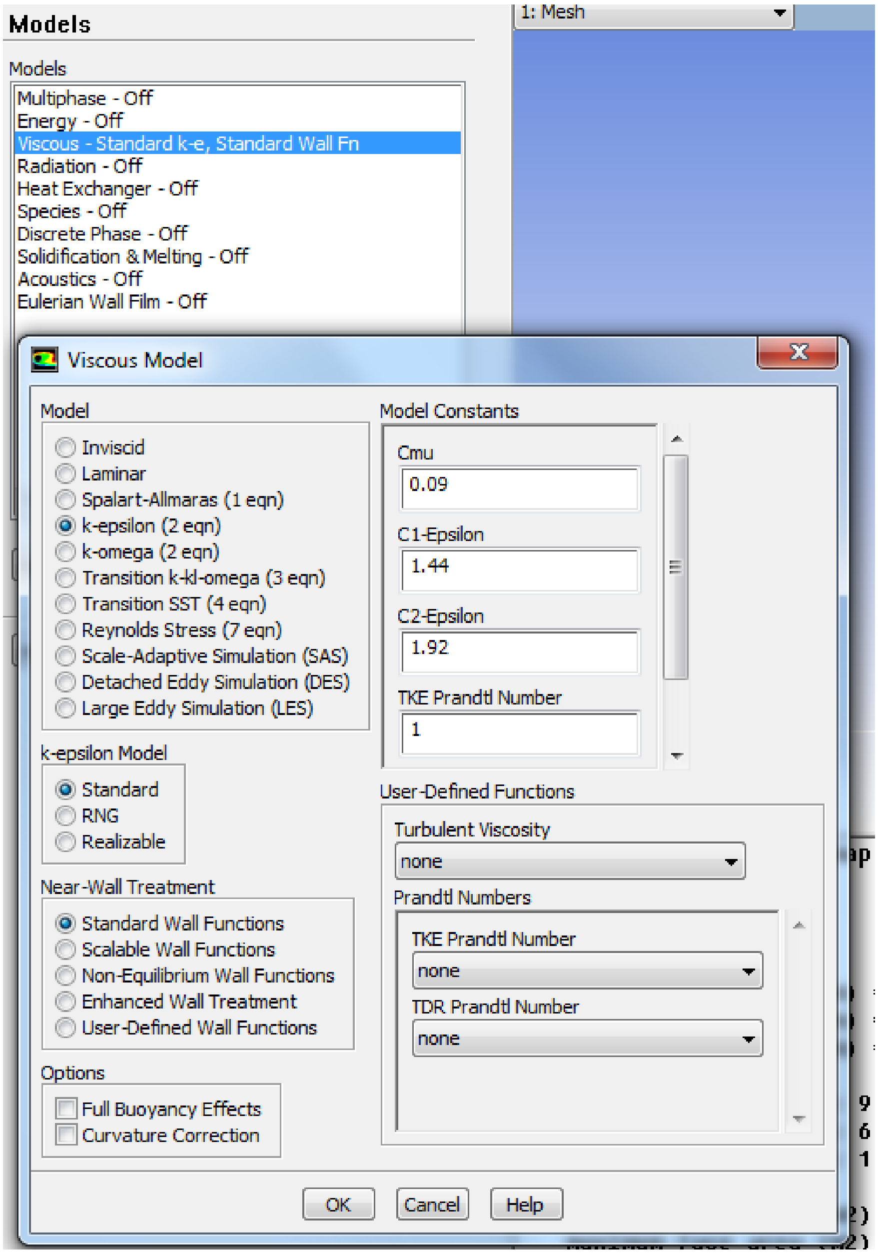


Figure 2.6 – Adjusting the turbulence parameters (Viscous)

Connecting the Energy Equation

When solving this problem, heat exchange and heat transfer must be taken into account. To do this, you need to connect the energy equation to the solution using the command:

Define → Models → Energy ...

In the window that appears, tick the Energy Equation box and click OK (Figure 2.7).

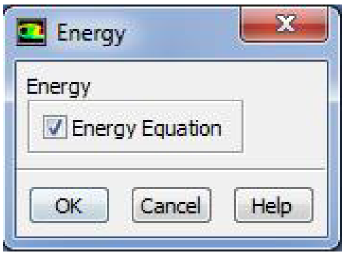


Figure 2.7 – Energy equation inclusion menu (Energy)

Setting the properties of the working fluid and reference pressure

You can set the properties of the working fluid in the Materials menu (Figure 2.8), which is called by the command:

MM: Define→ Materials

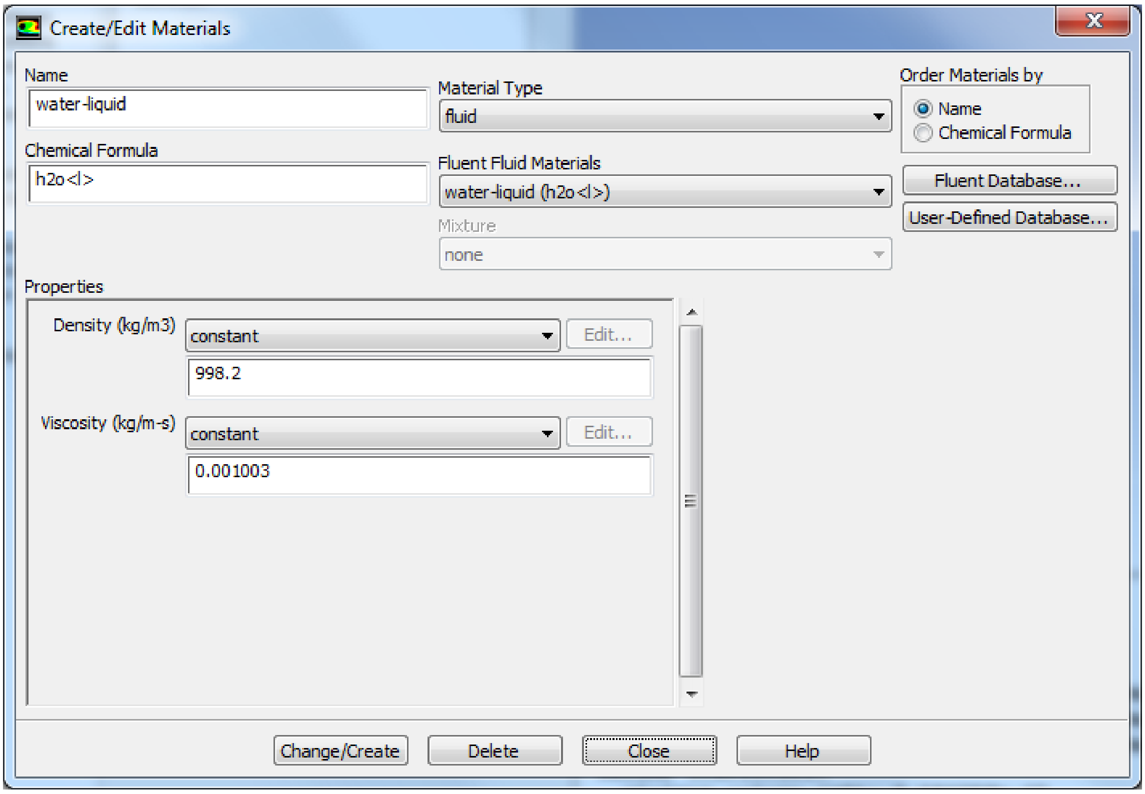


Figure 2.8 – Materials menu (Materials)

In this problem, water and air are used as a working fluid. Air is set by default. In order to add water as working fluid, in the Materials menu it is necessary to press the Fluent Data Bases button, and then select the necessary water-liquid material from the database in the appeared Fluent Fluid Materials list.

The peculiarity of the ANSYS Fluent program is that the pressure received and set in the calculation is redundant. That is, in order to obtain the true value of pressure, it is necessary to add to it the so-called "reference pressure". By default, its quality is normal atmospheric pressure - 101325 Pa. If 0 is taken as the reference pressure, the calculation results and the initial data will be set in absolute values. You can change the value of "reference pressure" in the menu (Figure 2.9), which appears as a result of the command:

MM: Define→ Operating Conditions

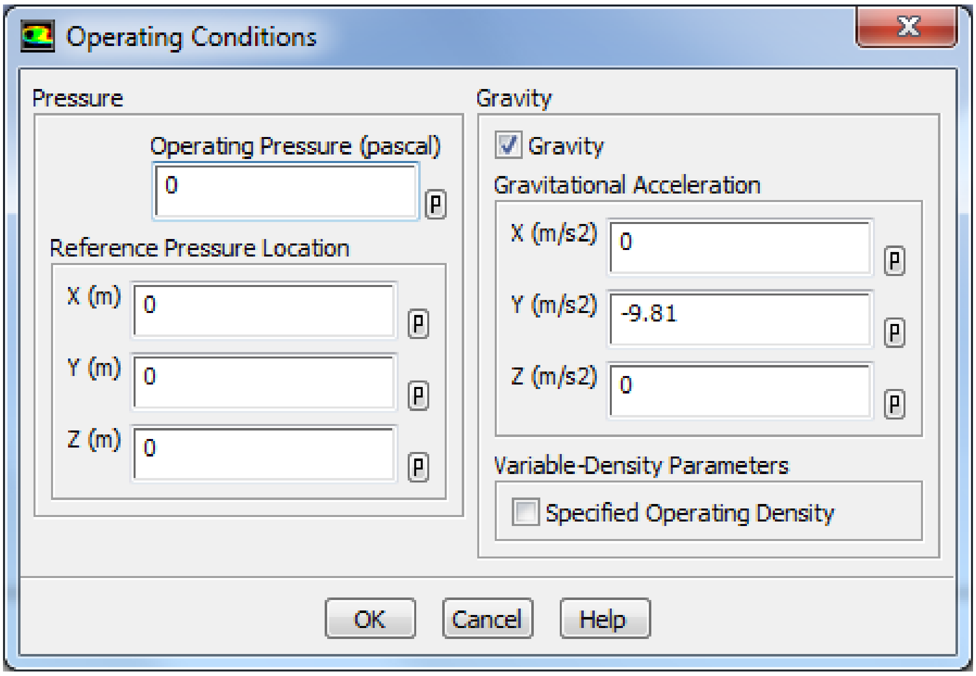


Figure 2.9 – Operating Conditions

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

## 2.3 Setting Boundary Conditions

The menu for setting boundary conditions (Figure 2.10) is called by the command:

MM: Define→ Boundary Conditions

The Zone field contains a list of all the boundary conditions defined in Meshing. If you select the name of one of them, for example, massflow_inlet_water, then the type of the boundary condition will be specified in the Type window. If necessary, the type of boundary conditions can be changed in this window. To start setting boundary conditions, you need to select the required boundary condition in the Zone window, make sure that the type of the boundary condition is specified in the Type window, then click Edit ...

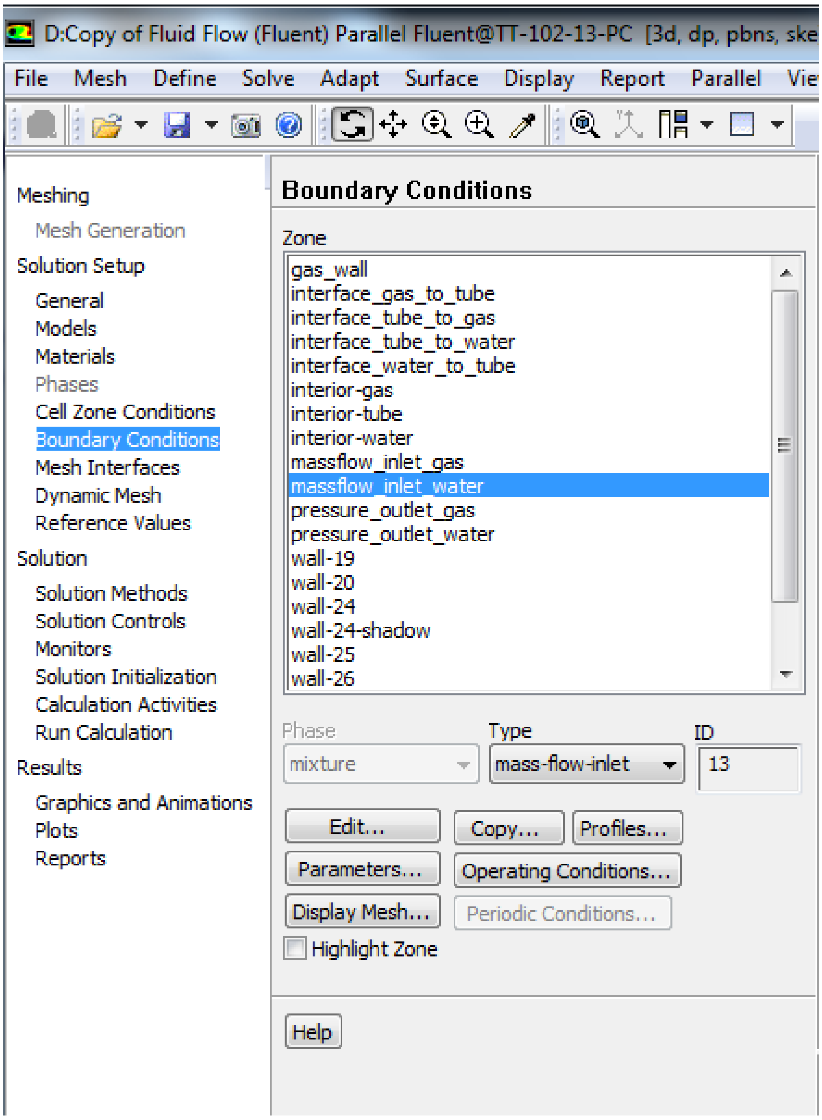


Figure 2.10 – Boundary Conditions

The name and type of the boundary condition are presented in Table 1.

Table 1. – Boundary conditions

| № | Boundary condition name | Boundary condition type |
|---|---|---|
| 1 | massflow_inlet_gas | pressure-inlet |

| | | |
|---|---|---|
| 2 | massflow_inlet_water | mass-flow-inlet |
| 3 | pressure_outlet_gas | pressure- outlet |
| 4 | pressure_outlet_water | pressure- outlet |
| 5 | interface_gas_to_tube | interface |
| 6 | interface_tube_to_gas | interface |
| 7 | interface_tube_to_water | interface |
| 8 | interface_water_to_tube | interface |

After that, the following actions are performed:

The parameters for the input boundary condition massflow_inlet_gas are set, in accordance with Figure 2.11:

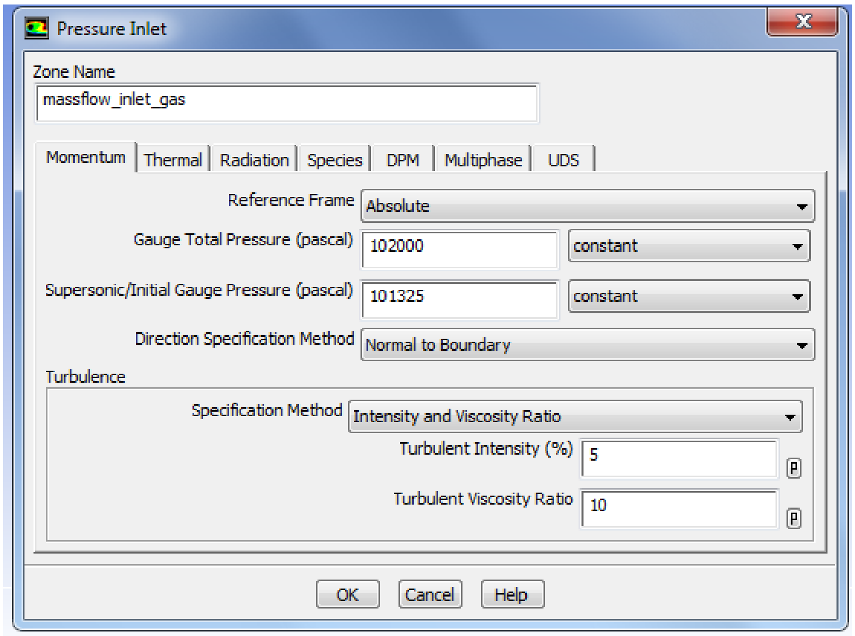


Figure 2.11 – Setting the boundary conditions for the inlet section of the gas path (massflow_inlet_gas)

The gas temperature is set in the Thermal tab and is 587 $^0$K (Figure 2.12).

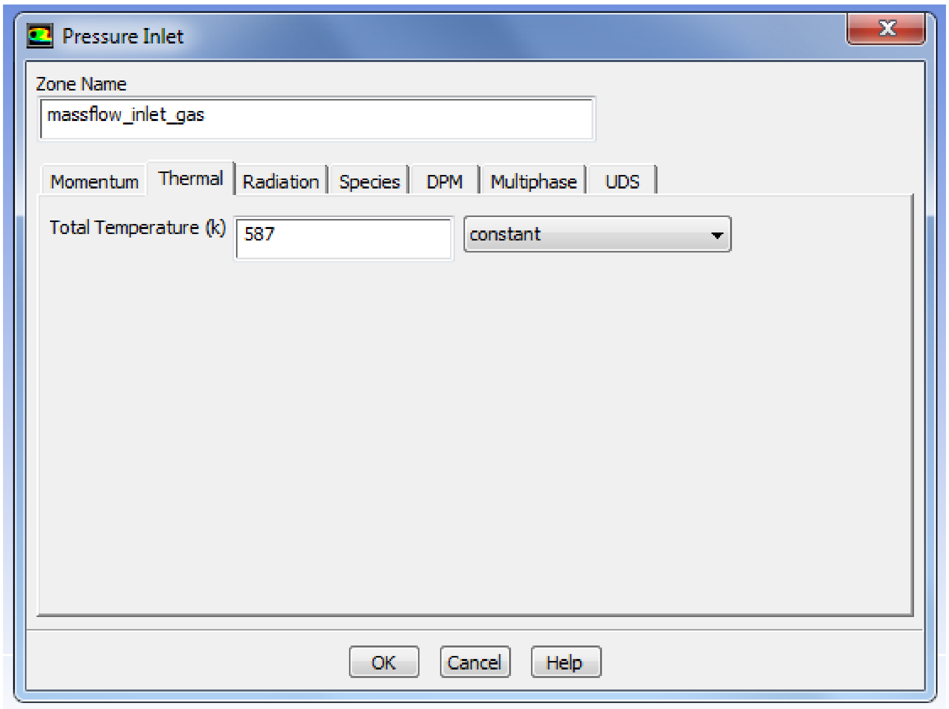


Figure 2.12 – Setting the gas temperature at the inlet

The parameters for the output boundary condition pressure_outlet_gas are set in accordance with Figure 2.13.

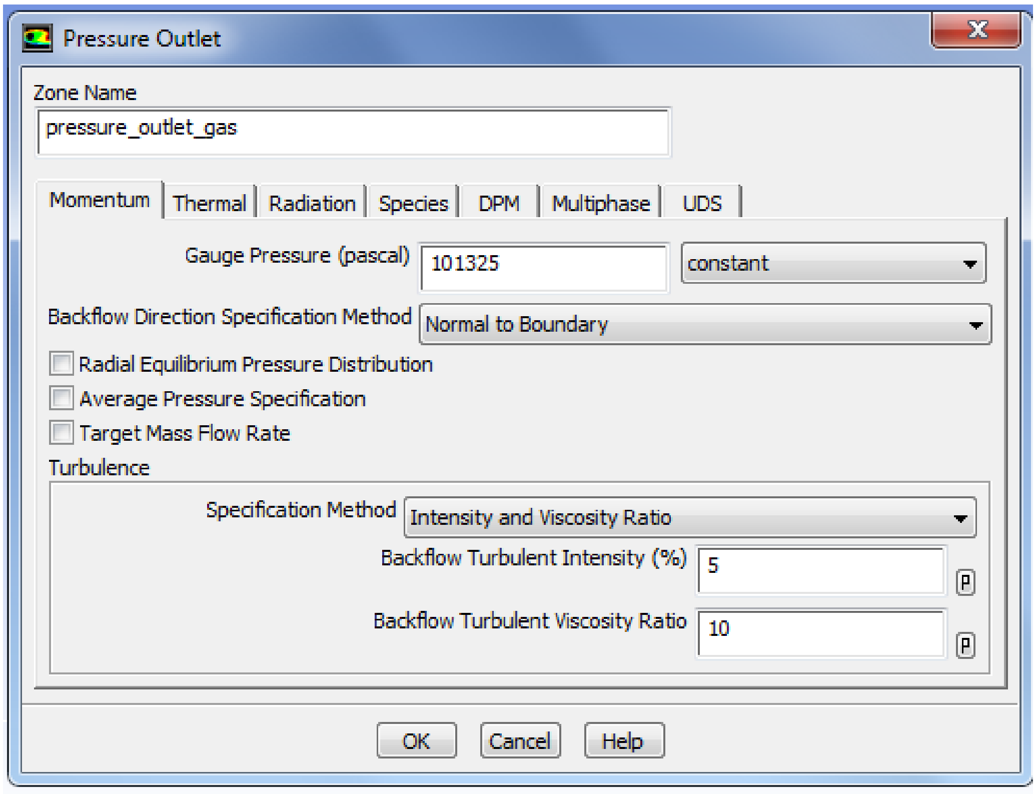


Figure 2.13 – Adjustment of the boundary conditions for the outlet section of the gas path (pressure_outlet_gas)

The parameters for the input boundary condition massflow_inlet_water are set in accordance with Figure 2.14.

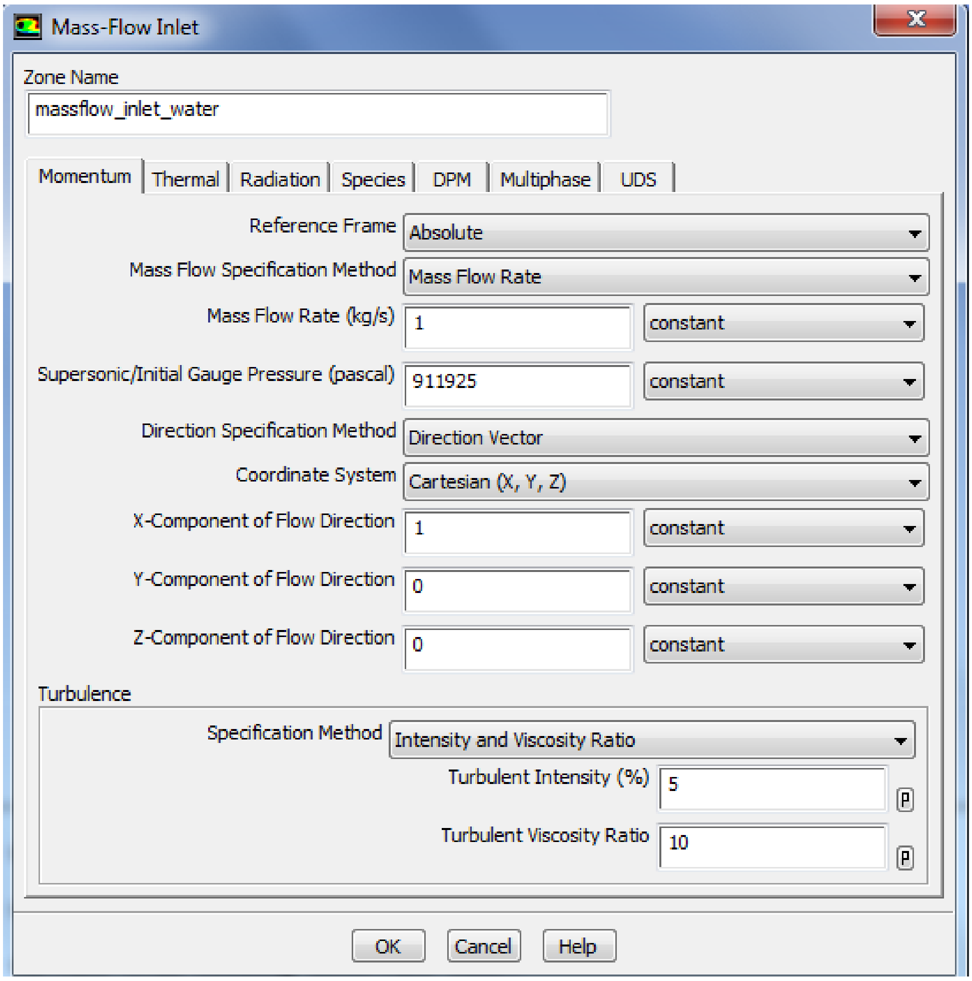


Figure 2.14 – Setting the boundary conditions for the input section of the water path (massflow_inlet_water)

The gas temperature is set in the Thermal tab and is 293 $^{0}$K.

The parameters for the input boundary condition pressure_outlet_water are set in accordance with Figure 2.15.

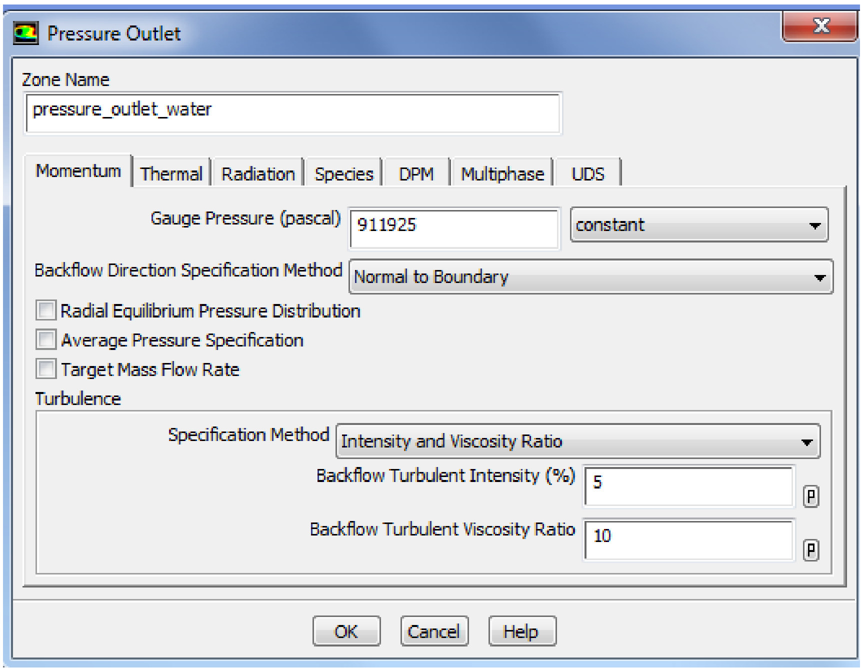


Figure 2.15 – Setting of the boundary conditions for the waterway outlet section (pressure_outlet_water)

Adjusting permeability zones between individual grid areas

ANSYS version 14.5 automatically creates permeability zones, after which they are displayed in the Mesh Interfaces window (Figure 2.16):

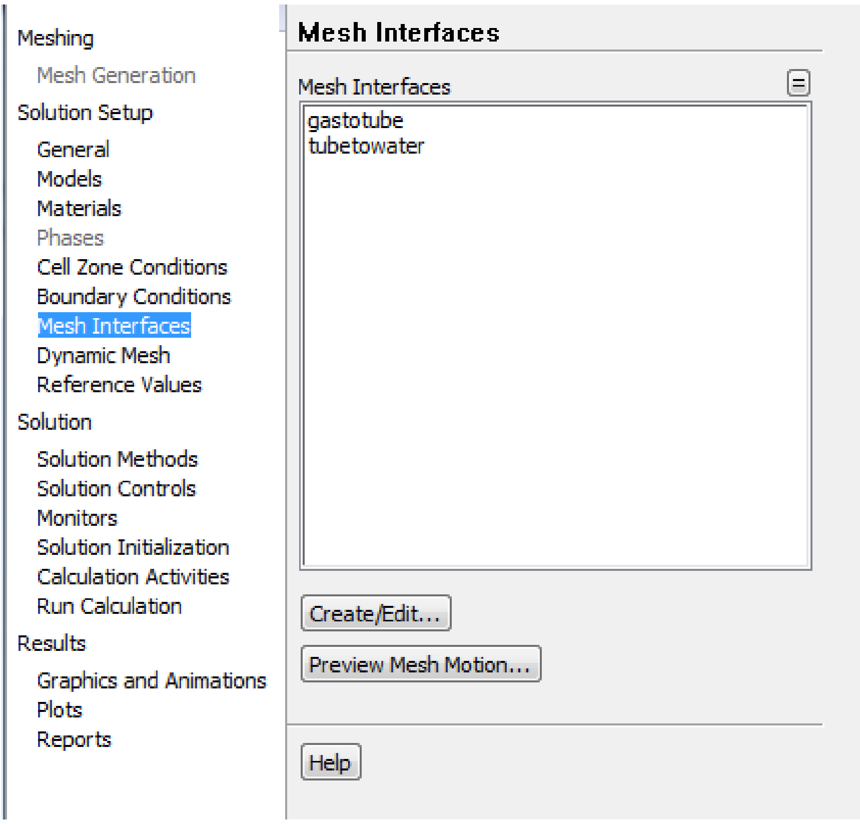


Figure 2.16 – Display of created surfaces

If you are working in an earlier version, these zones must be created. A total of 2 permeability zones are created: from the outlet branch to the cochlea and from the cochlea to the tube.

To do this, click Create / Edit in the Mesh Interfaces window. To create a zone of permeability (Figure 2.17), in the Mesh interface, you need to specify gastotube, select Interface_gas_to_tube as Interface Zone 1, interface_tube_to_ gas as an Interface Zone 1, and then click OK. Similarly,  tubetowater is created, as Interface Zone 1 - interface_tube_to_water, as Interface Zone 2 - interface_ water_to_tube.

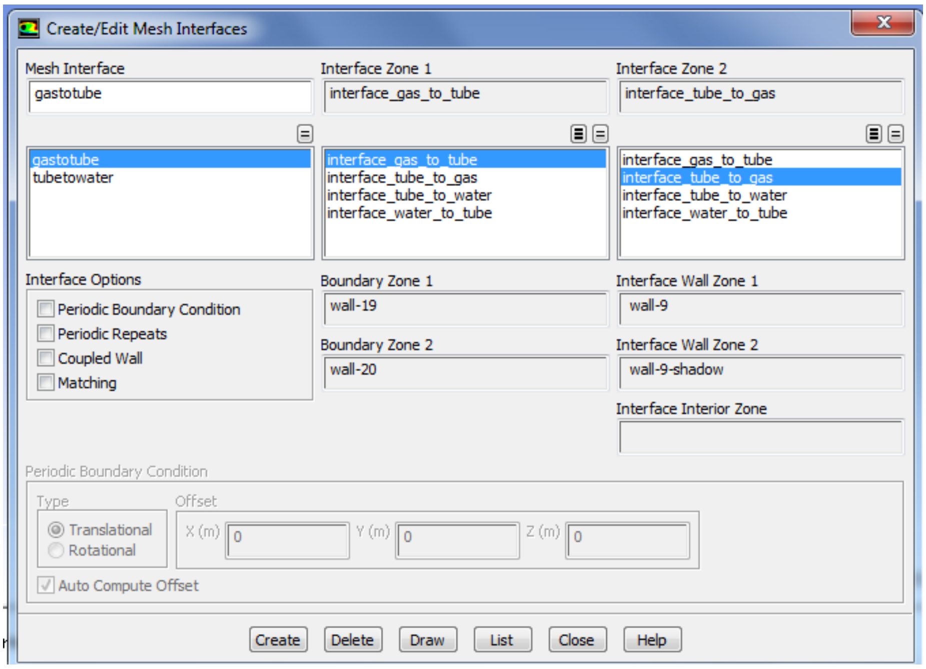


Figure 2.17 – Setting up the created permeability zones

Setting up the process for solving the problem

Set calculation parameters

The menu for setting calculation parameters is accessed using the command:

MM: Solve → Methods

As a result, the Solution Methods menu appears on the screen (Figure 2.18).

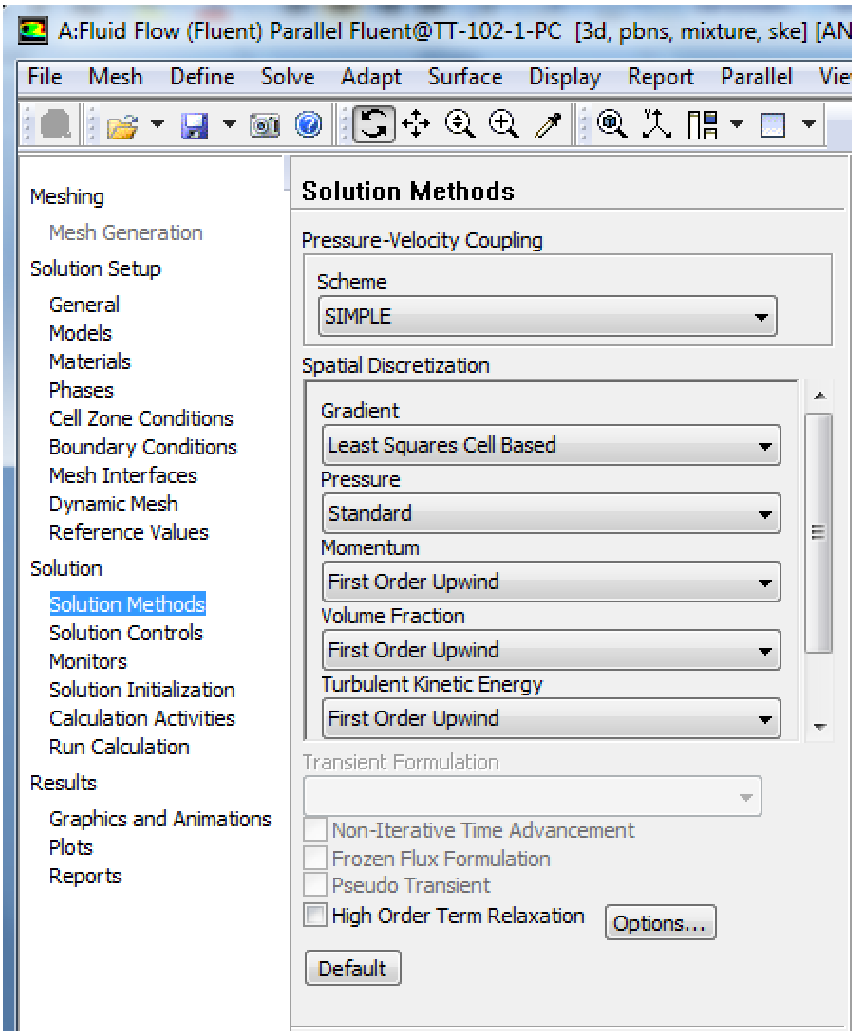


Figure 2.18 – Configuring calculation parameters (Solution Methods)

This menu consists of three main groups of configurable parameters:

Spatial Discretization;

Pressure-Velocity Coupling;

Transient Formulation.

In the Spatial Discretization zone, the discretization schemes of the corresponding equations are determined, that is, the algorithm for constructing a discrete analog of the differential equation at the node of the finite element grid is described.

To obtain exact solutions, it is necessary to use second and higher order sampling. However, at the first iterations such a solution is unstable.

Therefore, the first few dozen iterations should be carried out in the first order order (First Order), then increase it.

In the Pressure-Velocity Coupling zone, an algorithm for solving the equation of motion and continuity coupling is chosen to correctly determine the pressure and velocity fields. To solve the flow problems in interblade channels, it is advisable to choose the Coupled algorithm (it is similar to the splitting algorithm).

Set initial values of calculation parameters (initialization of the calculation process).

When solving problems of gas dynamics by numerical methods, it is necessary to establish the initial values of the parameters in the computational domain before starting the solution. The initial conditions setup menu (Figure 2.19) is called by the command:

MM: Solve→ Solution Initialization

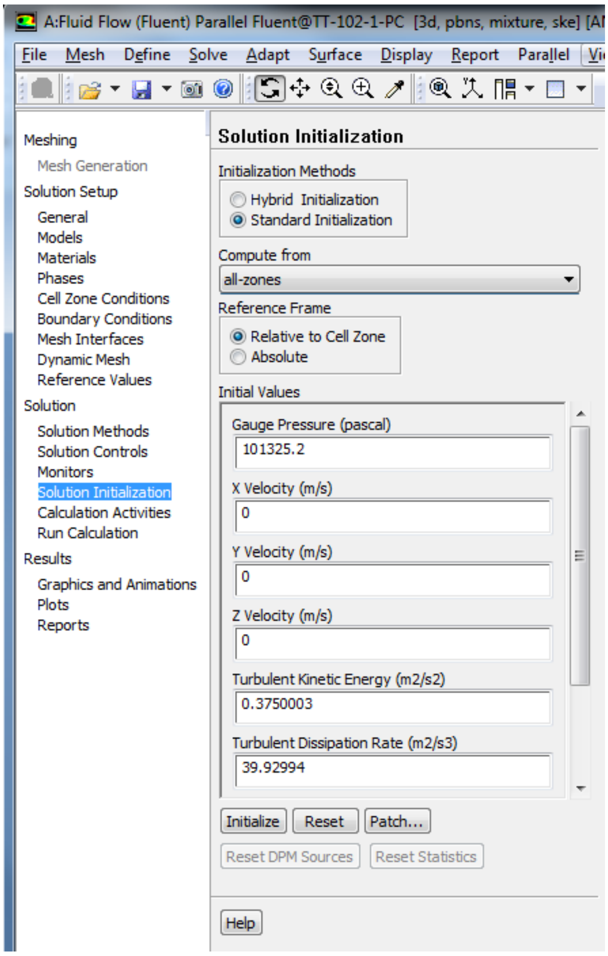


Figure 2.19 – Initialization of the process of calculation (Solution Initialization)

In the field of this menu, Compute From, you must select an 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.

Configuring the display of the solution process.

In order to display discrepancies in the calculation process, and also to set a criterion for stopping the solution, it is necessary to call up the Residual Monitors menu (Figure 2.20) with the command:

MM: Solve → Monitors→ Residual Monitors

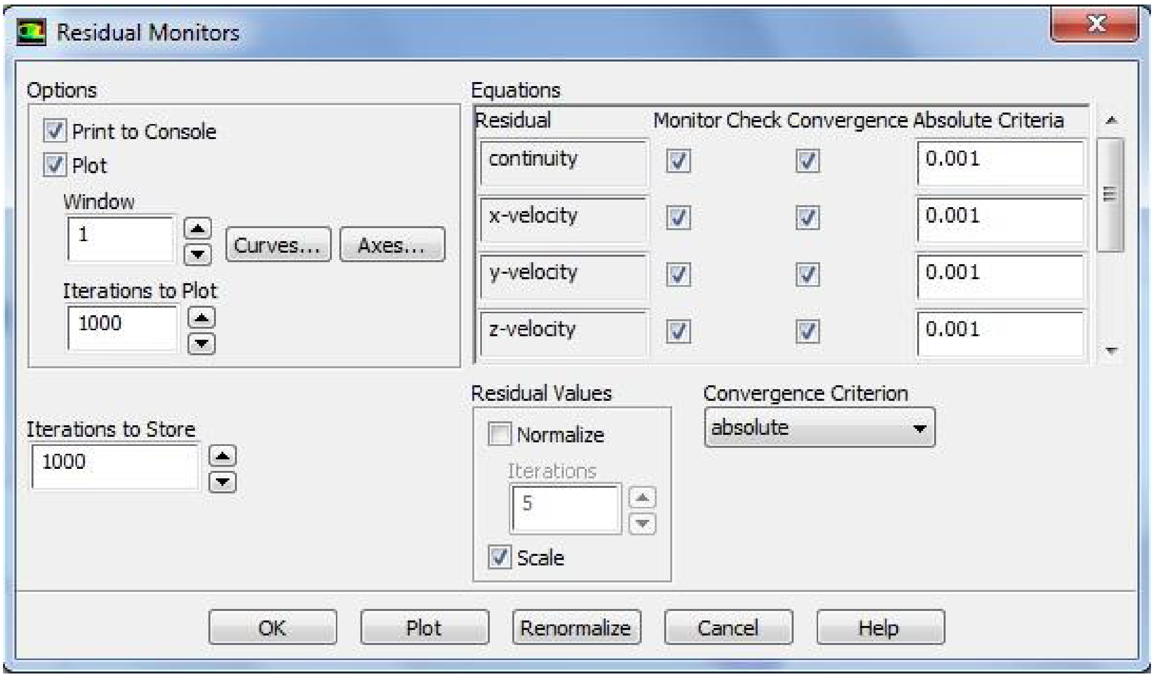


Figure 2.20 – Configuring the display of the solution process (Residual Monitors)

In the Option field, check the Plot and Print boxes. This will cause the residuals for all equations to be printed in the Print message window and displayed as graphs in the Plot graphics window.

In the fields of Residual, above each other, all solved equations are listed, and opposite to each of them, limiting discrepancies are established. The problem is considered solved when the residuals for all equations are less than the set values. In this case, the calculation process will be automatically stopped. It is believed that to obtain an exact solution it is sufficient to achieve the 10-3 residuals for all equations. To save the calculation model and all the settings made by the solver, you need to call the command:

MM: File → Save Project.

Run the calculation for 1000 iterations using the following actions (Figure 2.21):

MM: Solve → Run Calculation

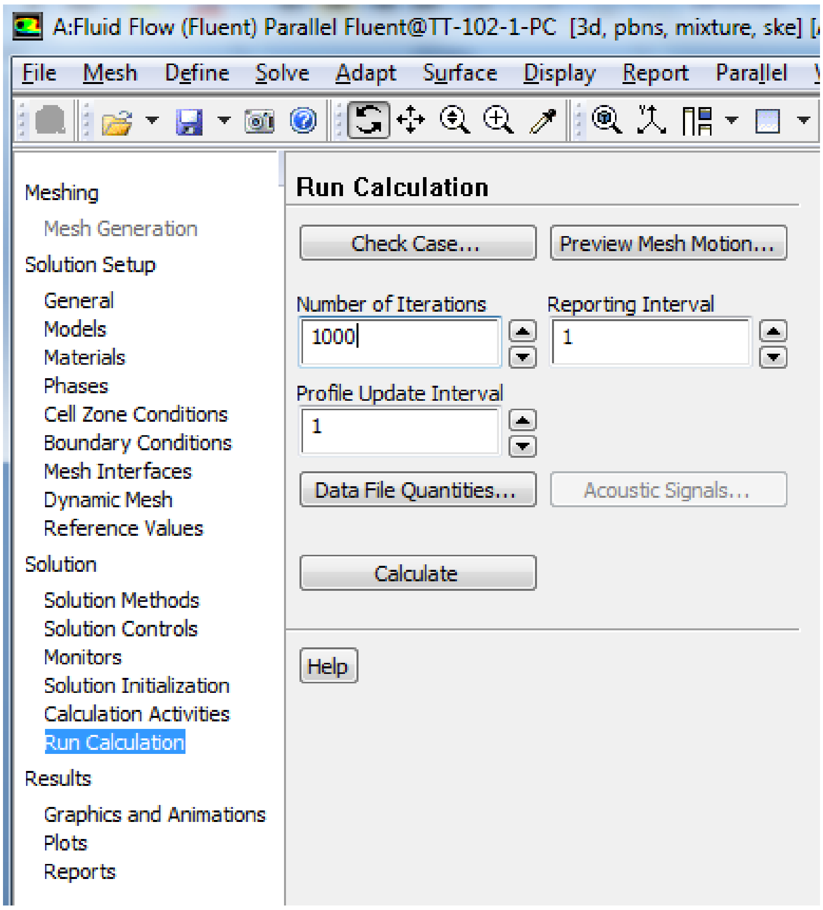


Figure 2.21 – Running the Calculation Process

If the condition of convergence is reached, then the process of counting will stop itself and the message "solution is converged" appears. If a given number of iterations is not enough to obtain a solution, then the decision process can be continued. The solution may not converge in this range of iterations. Then it is necessary to increase their number to 10,000. For faster convergence, it is possible to reduce the relaxation factors to 0.1.

## 2.4 Results Processing

At this stage, it is necessary to display the static pressure contours in the investigated places. To view the parameters distribution fields, run the following command:

MM: Display → Contours

As a result, the Contours menu appears (Figure 2.22). To distribute the parameters displayed in the form of fields, you must tick the Filled box. Otherwise, the parameter distributions will be displayed as isolines. The parameter whose change you want to display is selected in the Contours of field. It consists of two drop-down lists. In the upper group, the group to which the desired parameter belongs (for example, pressure) is selected. The lower list specifies which group parameter is to be determined (for example, static pressure). This is a typical and often used procedure for selecting the displayed parameter in the ANSYS Fluent program. Similarly, the parameter is determined when determining the mean integration values. The Surfaces field defines the surfaces on which the parameter distribution fields are constructed. In case you want to display parameters throughout the calculation area, the interior-solid surface is selected.

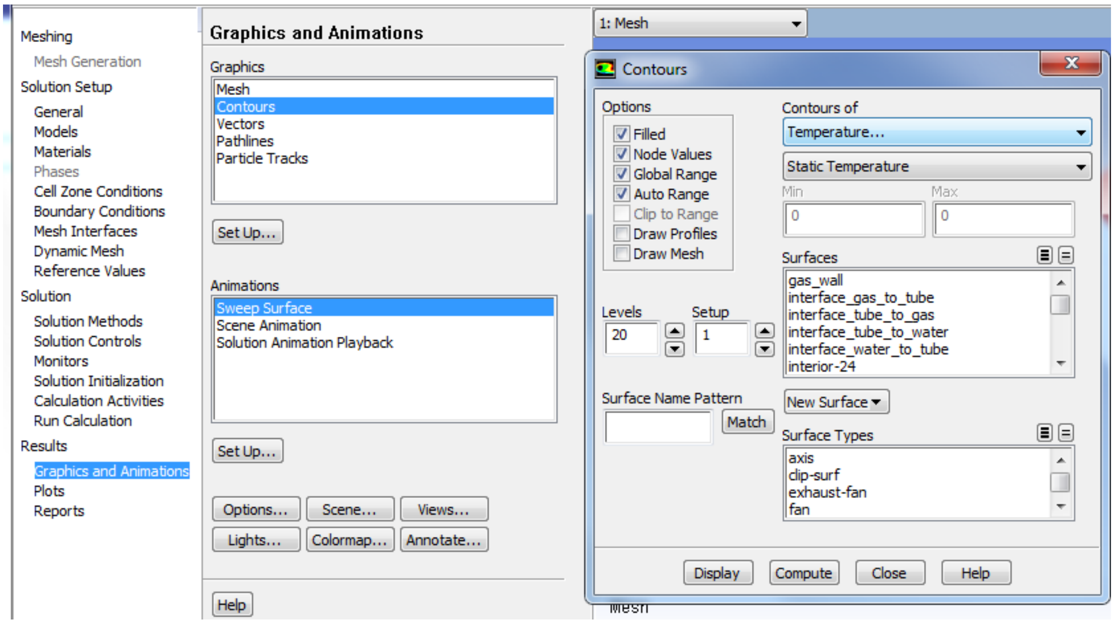


Figure 2.22 – Displaying the Contours menu

For a detailed study, in a particular section of the model, you can create an auxiliary plane on any selected axis. For this, the following actions are performed:

GM: Surface → Iso-Surface ...

In the window that appears (Figure 2.23) in the Surface of Constant field, Mesh and Z-Coordinate are selected.

In this field, you can enter the name of the new Surface Name. In this case, it remains the default.

Enter the coordinates of the section in which the study is carried out, for example Z = 0 for Iso-Values (Iso-Value).

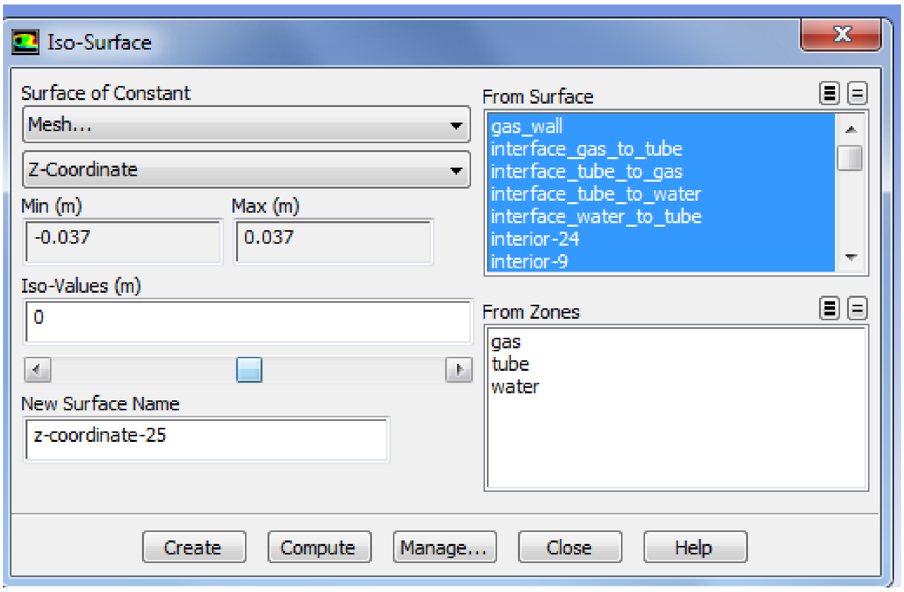


Figure 2.23 – Creating an auxiliary plane (Iso-Surface)

Then the Create button is pressed.

Similarly, other section planes are created (Figure 2.24–2.26).

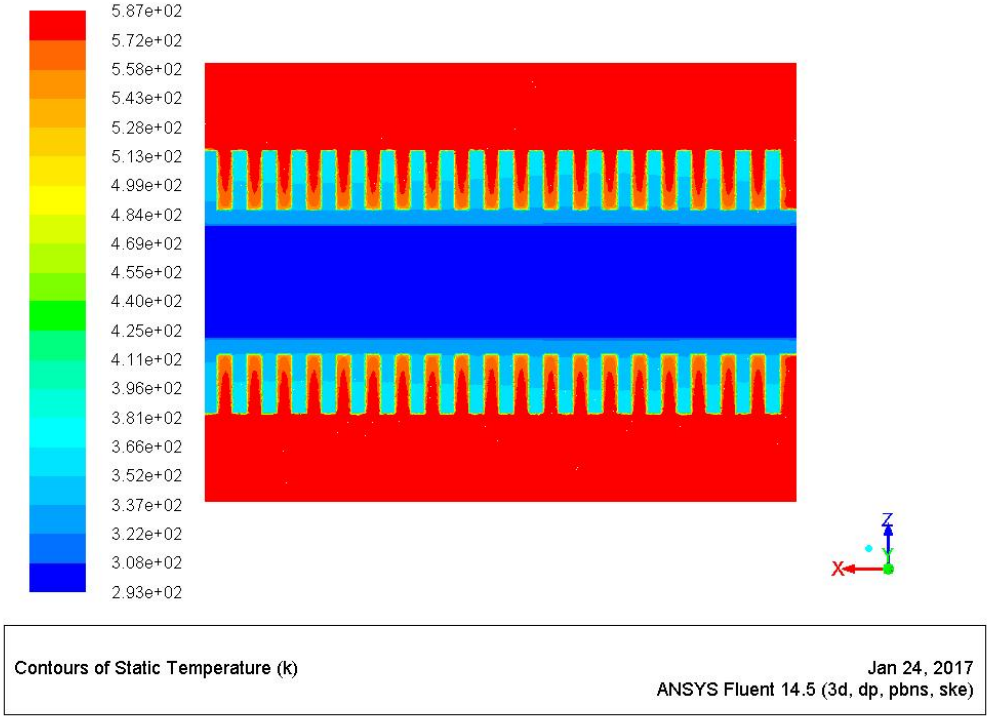


Figure 2.24 – Distribution of temperature fields in the longitudinal section of the heat exchanger

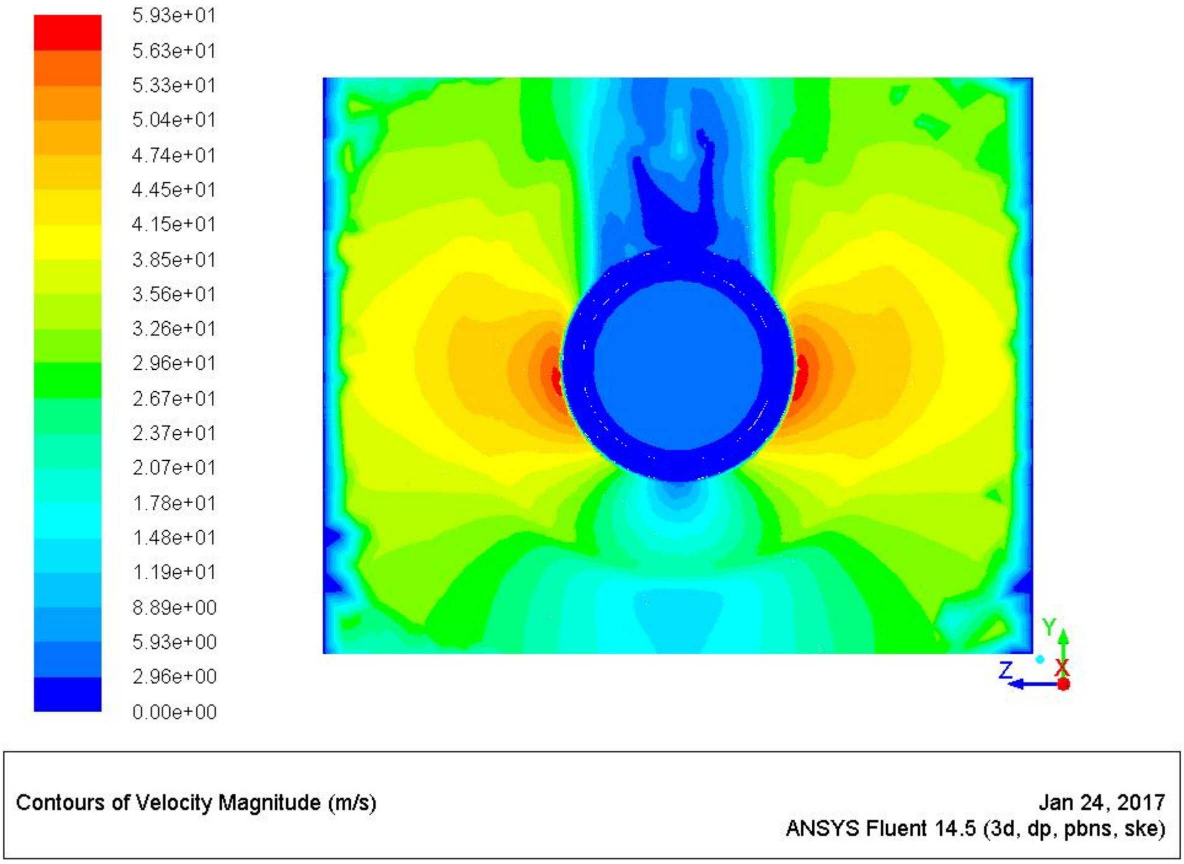


Figure 2.25 – Distribution of velocity fields in the cross section of the heat exchanger

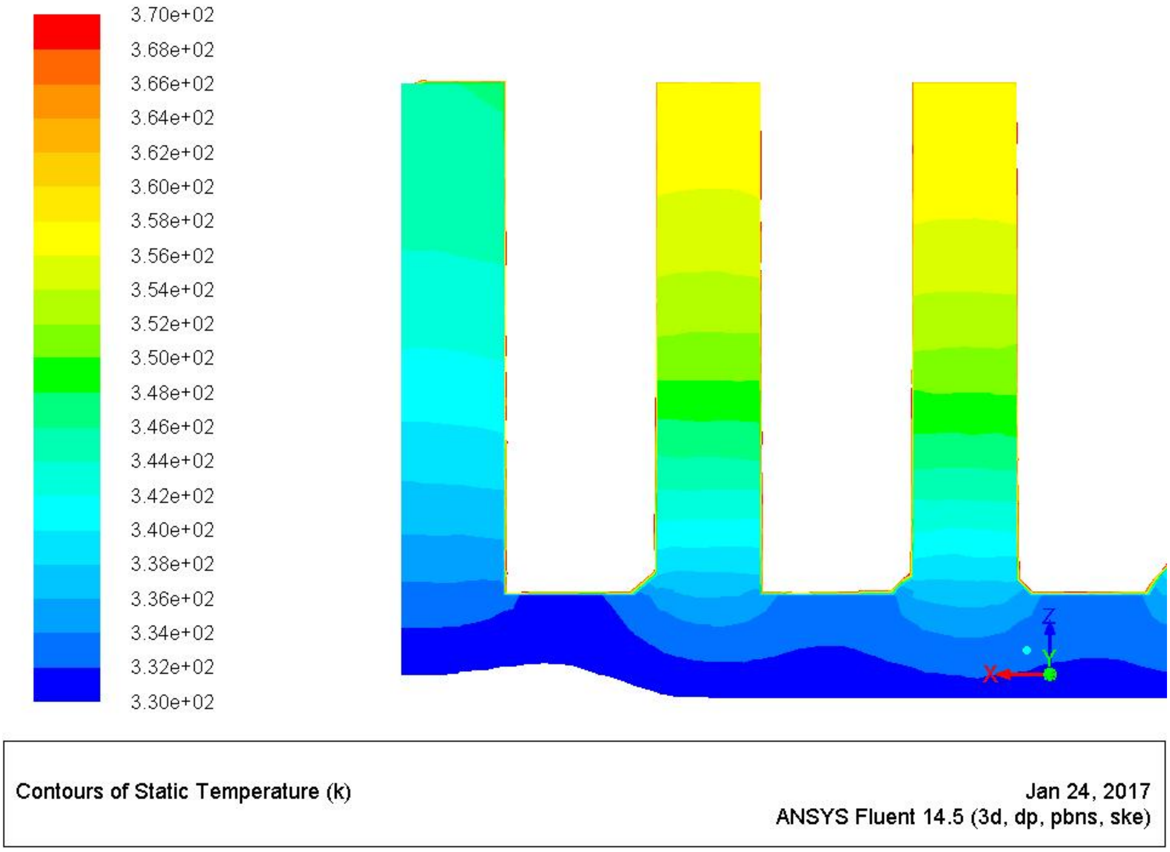


Figure 2.26 – Distribution of temperature fields in the longitudinal section of the heat exchanger fin

## Conclusion

Based on the results of the work done, the following conclusions can be drawn:

1. High informative results of numerical simulation contribute to a deeper analysis of heat transfer processes.

2. Simulation of the gas-dynamic flow structure of the turbulence model makes it possible to obtain the distributions of the basic thermodynamic parameters in the flow of the heated ball.

## List of references

1. Теплотехника [Текст]: метод. указание / Сост. В. Н. Белозерцев, В. В. Бирюк, А. П. Толстоногов.: - Самара: СГАУ, 2001. – 86 с.

2. Толстоногов, А. П. Расчет теплообменника газотурбинного двигателя замкнутого цикла [Текст]: методическое пособие / А. П. Толстоногов. – Куйбышев: КуАИ, 1984. – 16 с.

3. Нагрев и охлаждение трансформаторов [Текст] / Под редакцией Г. Е. Тарле. – М.: Энергия, 1980.

4. Гавр, Т. Г. Тепловой и гидравлический расчет теплообменных аппаратов компрессорных установок [Текст]: учебное пособие / Т.Г. Гавр, П. М. Михайлов, В. В. Рис. – Л.: Ленинградский политех. ин-т им. М. И. Калинина, 1982. – 72 с.

5. Справочник по гидравлическим сопротивлениям. 3-е изд. [Текст] / Под ред. М. О. Штейнберга. – М.: Машиностроение, 1992. – 672 с.

6. Батурин О. В. Расчет течений жидкостей и газов с помощью универсального программного комплекса. Часть 3. Работа в программе Fluent/ О. В. Батурин, И. И. Морозов, В. Н. Матвеев – Самара: Изд-во Самар. гос. аэрокосм. ун-та, 2008. - 115с.