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	3
Chapterr 1. PREPARATION OF THE CALCULATION MODEL	4
1.1 Shell modelling using ANSYS Workbench	4
1.2 Finite element mesh construction	7
Chapter 2. MODELLING HEAT EXCHANGERS	10
2.1 Running the ANSYS Fluent program and its features	10
2.2 Setting up calculation parameters	13
2.3 Setting Boundary Conditions	17
2.4 Result Processing	27
Conclusion	32
List of references	33

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 \rightarrow All Programs \rightarrow ANSYS

14.5 \rightarrow 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).

Λ к	avitator - Workbench	-						
File	e View Tools Units Extensions H	lelp						
1	New 📔 Open 房 Save 🔣 Save As	🚮 Im	port.		∉φ Reconnect	建 Refresh	Project	🕖 Update Project
Tool	оох 🝷 🗸 🗙	Project	t Sche	emat	tic			
	Analysis Systems							
	Design Assessment							
٢	Electric		•		А			
J.	Explicit Dynamics		1	C	Fluid Flow (Flue	ent)		
C	Fluid Flow - Blow Molding (Polyflow)		2	Θ	Geometry	? 🖌		
G	Fluid Flow-Extrusion(Polyflow)		3		Mesh			
G	Fluid Flow (CFX)			*				
G	Fluid Flow (Fluent)				Setup	2 4		
C	Fluid Flow (Polyflow)		5	6	Solution	P 🖌		
\sim	HarmonicResponse		6	6	Results	P 🖌		
\sim	Hydro dynamic Diffraction				Fluid Flow (Flue	ent)		
\simeq	Hydrodynamic Time Response				1101011011(110	ency		
	IC Engine							
Σ	Linear Buckling							
00	Magnetostatic							
"I"	Modal							
" ! "	Modal (Samcef)							
ald	Random Vibration							
aide	Response Spectrum							
7776	Rigid Dynamics							
777	Static Structural							
777	Static Structural (Samcef)							
	Steady-State Thermal							

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 \rightarrow 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).

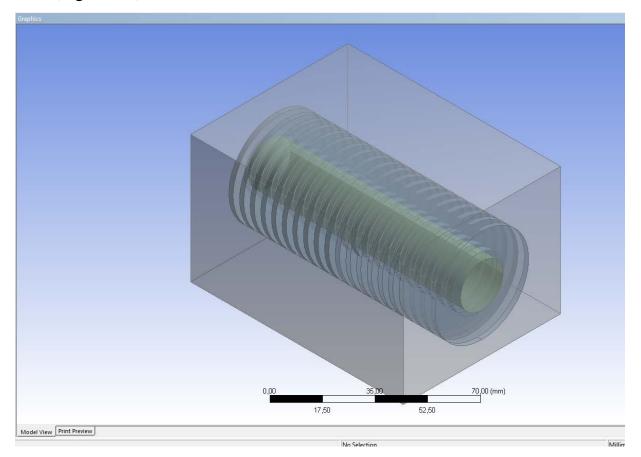


Figure 1.3 – Appearance of the heat exchanger section

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).

🗊 A: Fluid Flow (Fluent) - DesignModeler	
File Create Concept Tools View Help	
] 🖉 🛃 🛃 📫] 💬 Undo 📿 Redo Select: 🌇 🏷 🔹 💽 💽 🔂 🏹	
] Ⅲ - 	
📙 XYPlane 🛛 🔻 🗼 None 🚽 ಶ 🗍 🧚 Generate 🎲 Share Topology 😰	Param
📙 🛅 Thin/Surface 💊 Blend 🔻 🦠 Chamfer 🏘 Slice 🗍 🚸 Point 📳 Conversion	
🛛 BladeEditor: 🎇 Import BGD 🛛 🔠 Load BGD 🛛 🚍 FlowPath 🥒 Blade 💋 Splitter 🚽 Vi	staTFE
<u> </u> ଡ॒⊼ଃ≣(<u></u> ള)፼	p
Tree Outline	4 G
XYPlane XZPlane YZPlane So Measure Selection Hide Body Hide All Other Bodies Suppress Body Generate Bo Rename	

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.

Details of Body	
Body	gas
Volume	
Surface Area	
Faces	125
Edges	252
Vertices	168
Fluid/Solid	Fluid
Shared Topology Method	Fluid
Geometry Type	Solid
	Solid

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).

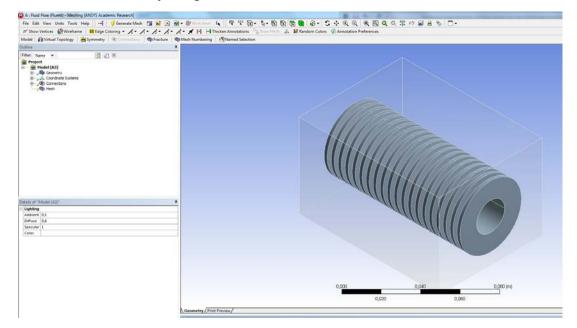


Figure 1.6 – The working window of the Meshing module and the appearance of the loaded geometric model of the heat exchanger section.

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 \rightarrow Sizing (Figure 1.7).

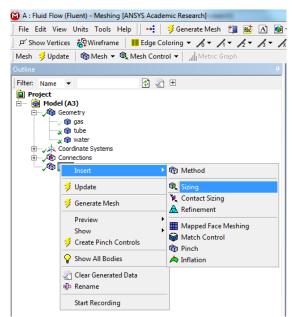


Figure 1.7 – Sizing grid size adjustment menu

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

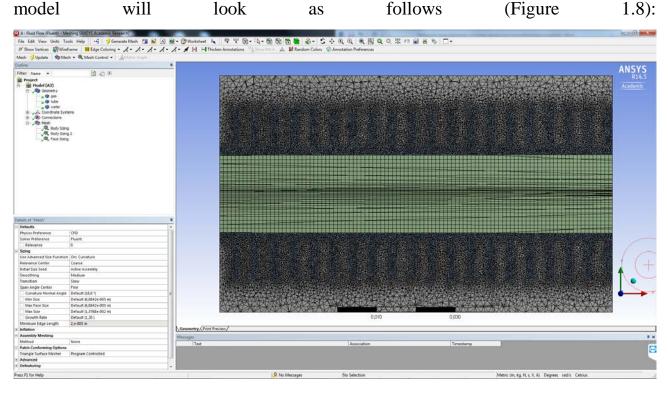


Figure 1.8 – Appearance of the mesh

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.

Selection Name
jinlet
Apply selected geometry
 Apply geometry items of same:
Size
🗌 Туре
Location X
Location Y
Location Z
Apply To Corresponding Mesh Nodes
OK Cancel

Figure 1.9 – Appearance of the surface name selection window

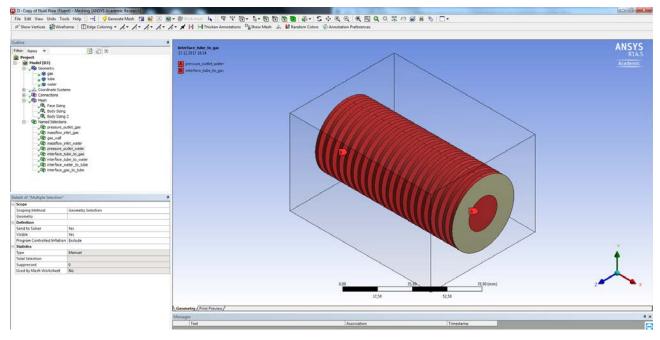


Figure 1.10 – Appearance of the Surface Name Selection Window After that, the Meshing module can be closed.

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).

Fluent Launcher (Setting Edit Only)	
ANSYS	Fluent Launcher
Dimension 2D 3D Display Options Display Mesh After Reading Embed Graphics Windows	Options Double Precision Meshing Mode Use Job Scheduler Use Remote Linux Nodes
 Workbench Color Scheme Do not show this panel again Show More Options 	Processing Options Serial Parallel (Local Machine) Number of Processes
	ancel <u>H</u> elp v

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).

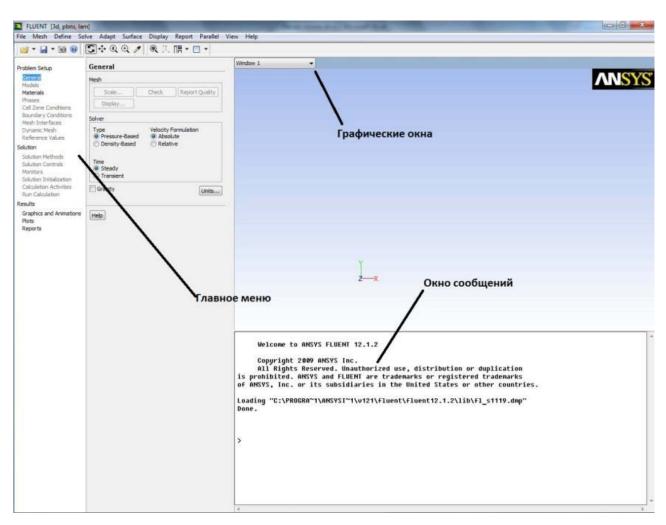


Figure 2.2 – ANSYS Fluent working window

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 \rightarrow 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.

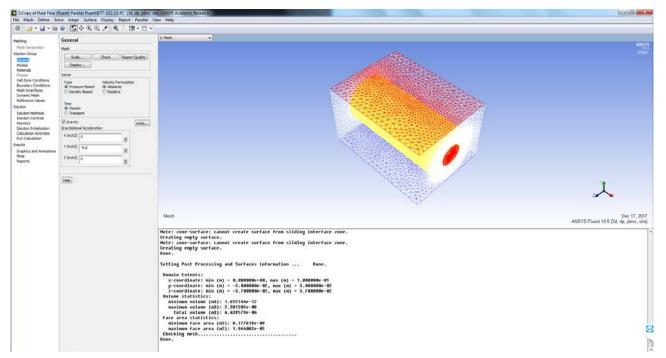


Figure 2.3 – Mesh Display

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 \rightarrow 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.

Mesh Display			×
Options E Vodes Edges Faces Partitions	idge Type	Surfaces gas_wall interface_gas_to_tube interface_tube_to_gas interface_tube_to_water	
Shrink Factor	eature Angle	interface_water_to_tube interior-24 interior-9 interior-gas	-
Surface Name Patt	ern Match	New Surface Surface Types	
Outline Interior	r	axis clip-surf exhaust-fan fan	•
Dis	play Color	rs Close Help	

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.

A:Fluid Flow (Fluent) Pa	arallel Fluent@TT-102-1-PC [3d, pbns, lam] [ANSYS (
<u>F</u> ile <u>M</u> esh D <u>e</u> fine <u>S</u> ol	ve <u>A</u> dapt S <u>u</u> rface <u>D</u> isplay <u>R</u> eport Para <u>l</u> lel
i 📖 i 📂 🖌 🖌 🖬	❷ 🖫 ↔ ④ ⊕ 🥕 🔍 ♥ 🔍 🖷 ▾ 🗆 ▾
Meshing Mesh Generation Solution Setup General Models Materials Phases Cell Zone Conditions Boundary Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution Solution Methods Solution Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation Results Graphics and Animations Plots Reports	Image: Imag
	Help

Figure 2.5 – The solver setting menu (Solver)

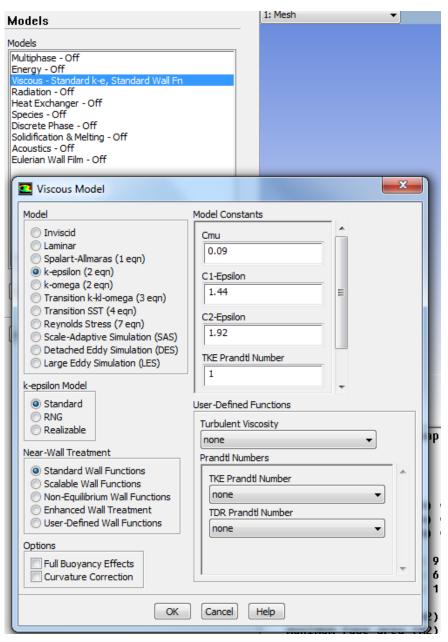
In the Time field, it is selected whether the solution is stationary Steady or nonstationary Transient. That is, whether the flow parameters will depend on the time or not. The problem under consideration is three-dimensional stationary. In addition, in this problem, gravity must be taken into account.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

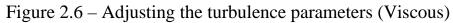
Choice of the model of turbulence

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 \rightarrow Models \rightarrow 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 \rightarrow Models \rightarrow Energy ...
```

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

Energy	
Energy Equation	
OK Cancel	Help

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:

ame		Material Type	Order Materials by
water-liquid		fluid	 Name
nemical Formula n2o <l></l>		Fluent Fluid Materials water-liquid (h2o <l>) Mixture none</l>	Chemical Formula Fluent Database User-Defined Database
operties			
Density (kg/m3)	constant 998.2	Edit	
Viscosity (kg/m-s)	constant 0.001003	Edit	
		*	

MM: Define \rightarrow 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

Operating Conditions	×
Pressure Operating Pressure (pascal) Reference Pressure Location X (m) V (m) C Y (m) C P Z (m) P P	Gravity Gravity Gravitational Acceleration X (m/s2) 0 Y (m/s2) -9.81 Z (m/s2) 0 Variable-Density Parameters Specified Operating Density
ОКС	Cancel Help

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 ...

	luent) Parallel Fluent@TT-102-13-PC [3d, dp, pbns, ske
File Mesh Define Sol	lve Adapt Surface Display Report Parallel Vie
i 🔳 i 📂 - 🖬 - 🚳	@ \$\$ ↔ Q & ↗ @ 洗 II ▼ 🗆 ▼
Meshing	Boundary Conditions
Mesh Generation	Zone
Solution Setup	gas wall
General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution Solution Methods Solution Controls Monitors Solution Initialization	gas_wall Image: Constraint of the system interface_gas_to_tube interface_tube_to_gas interface_tube_to_water interface_water_to_tube interior-gas interior-tube interior-water massflow_inlet_gas massflow_inlet_water pressure_outlet_gas pressure_outlet_gas pressure_outlet_water wall-19 wall-20 wall-24 wall-25 wall-25 wall-26
Calculation Activities Run Calculation	Phase Type ID
Run Calculation Results	Phase Type ID mixture Type ID
Graphics and Animations Plots Reports	Edit Copy Profiles Parameters Operating Conditions Display Mesh Periodic Conditions Highlight Zone

Figure 2.10 – Boundary Conditions

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

Table 1. – Boundary conditions

N⁰	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:

Pressure Inlet	X		
Zone Name			
massflow_inlet_gas			
Momentum Thermal Radiation Species DPM Multiphase U	IDS		
Reference Frame Absolute	•		
Gauge Total Pressure (pascal)	constant 🔻		
Supersonic/Initial Gauge Pressure (pascal) 101325	constant 🗸		
Direction Specification Method Normal to Boundary			
Turbulence			
Specification Method Intensity and Viscosity Ratio			
Turbulent Intensity (%) 5			
Turbulent Viscosity Ratio 10			
OK Cancel Help			

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 ^oK (Figure 2.12).

Pressure Inlet	x
Zone Name	
massflow_inlet_gas	
Momentum Thermal Radiation Species DPM Multiphase UDS	
Total Temperature (k) 587 constant	
OK Cancel Help	

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.

Pressure Outlet		
Zone Name		
pressure_outlet_gas		
Momentum Thermal Radiation Species DPM Multiphase UDS		
Gauge Pressure (pascal) 101325 constant		
Backflow Direction Specification Method Normal to Boundary		
Radial Equilibrium Pressure Distribution		
Average Pressure Specification		
Target Mass Flow Rate		
Turbulence		
Specification Method Intensity and Viscosity Ratio		
Backflow Turbulent Intensity (%) 5		
Backflow Turbulent Viscosity Ratio		
OK Cancel Help		

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.

Mass-Flow Inlet		X	
Zone Name			
massflow_inlet_water			
Momentum Thermal Radiation Species DPM Multiphase UDS			
Reference Frame	Absolute	•	
Mass Flow Specification Method	Mass Flow Rate		
Mass Flow Rate (kg/s)	1	constant 👻	
Supersonic/Initial Gauge Pressure (pascal)	911925	constant 🔻	
Direction Specification Method Direction Vector		•	
Coordinate System	Cartesian (X, Y, Z)	•	
X-Component of Flow Direction	1	constant 👻	
Y-Component of Flow Direction	0	constant 🔻	
Z-Component of Flow Direction	0	constant 💌	
Turbulence			
Specification Method Intensity and Viscosity Ratio			
Turbulent Intensity (%) 5			
Turbulent Viscosity Ratio 10			
OK Cancel Help			

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 ^oK.

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

Pressure Outlet
Zone Name
pressure_outlet_water
Momentum Thermal Radiation Species DPM Multiphase UDS
Gauge Pressure (pascal) 911925 constant
Backflow Direction Specification Method Normal to Boundary
Radial Equilibrium Pressure Distribution
Average Pressure Specification
Target Mass Flow Rate
Turbulence
Specification Method Intensity and Viscosity Ratio
Backflow Turbulent Intensity (%) 5
Backflow Turbulent Viscosity Ratio 10
OK Cancel Help

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):

Meshing	Mesh Interfaces
Mesh Generation	Mesh Interfaces
Solution Setup General Models Materials Phases Cell Zone Conditions Boundary Conditions <u>Mesh Interfaces</u> Dynamic Mesh Reference Values	gastotube tubetowater
Solution Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation Results Graphics and Animations Plots Reports	Create/Edit Preview Mesh Motion Help

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.

Create/Edit Mesh Interfaces		— ×			
Mesh Interface	Interface Zone 1	Interface Zone 2			
gastotube	interface_gas_to_tube	interface_tube_to_gas			
gastotube	interface_gas_to_tube	interface_gas_to_tube			
tubetowater	interface_tube_to_gas	interface_tube_to_gas			
	interface_tube_to_water	interface_tube_to_water			
	interface_water_to_tube	interface_water_to_tube			
Interface Options	Boundary Zone 1	Interface Wall Zone 1			
Periodic Boundary Condition	wall-19	wall-9			
Periodic Repeats	Boundary Zone 2	Interface Wall Zone 2			
Coupled Wall Matching	wall-20	wall-9-shadow			
		Interface Interior Zone			
Periodic Boundary Condition					
Type Offset					
Image: Translational Rotational X (m) 0 Y (m) 0 Z (m) 0					
Auto Compute Offset	☑ Auto Compute Offset				
Create Delete Draw List Close Help					

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 \rightarrow 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.

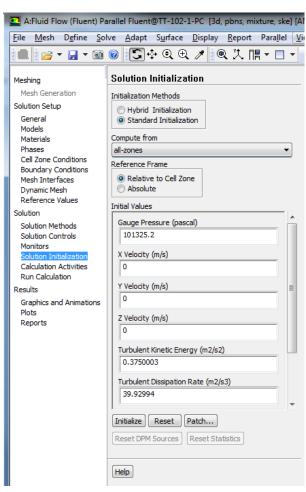
To obtain exact solutions, it is necessary to use second and higher order sampling. However, at the first iterations such a solution is unstable.

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:

options	Equations				
Print to Console	Residual	Monitor C	heck Converge	ence Absolute Criteria	
V Plot	continuity			0.001	
Window 1 Axes	x-velocity		1	0.001	1
Iterations to Plot	y-velocity		\checkmark	0.001	
1000	z-velocity			0.001	
	Residual Values	Con	vergence Crite	rion	
terations to Store	Mormalize	abs	olute	•	
1000	Iterations 5				
	Scale				

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 \rightarrow Save Project.

Run the calculation for 1000 iterations using the following actions (Figure 2.21): MM: Solve \rightarrow Run Calculation

A:Fluid Flow (Fluent) Parallel Fluent@TT-102-1-PC [3d, pbns, mixture, ske] [,				
<u>F</u> ile <u>M</u> esh D <u>e</u> fine <u>S</u> ol	lve <u>A</u> dapt S <u>u</u> rface <u>D</u> isplay <u>R</u> eport Para <u>l</u> lel <u>\</u>			
i 📖 i 📂 🕶 🖬 👻 🚳	❷ 💽 أ € € € 🥒 🔍 🔍 🐨 – – –			
Meshing	Run Calculation			
Mesh Generation Solution Setup	Check Case Preview Mesh Motion			
General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values	Number of Iterations Reporting Interval 1000 1 Profile Update Interval 1 1 Image: Construction of the second s			
Solution Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation	Calculate			
Results Graphics and Animations Plots Reports				

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 \rightarrow 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.

Meshing	Graphics and Animations	1: Mesh	•
Mesh Generation Solution Setup General Models	Graphics Mesh Contours Vectors Pathlines	Contours Options Filled	Contours of Temperature
Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces	Partide Trads Set Up	Clip to Range	Static Temperature
Dynamic Mesh Reference Values Solution Solution Methods Solution Controls Monitors	Animations Sweep Surface Scene Animation Solution Animation Playback	Levels Setup 20 • 1 •	Surfaces
Solution Initialization Calculation Activities Run Calculation Results Graphics and Animations Plots Reports	Set Up Options Scene Views	Surface Name Pattern	New Surface Surface Types
	Lights Colormap Annotate Help	Display (Compute Close Help

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 \rightarrow 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).

Iso-Surface			
Surface of Constant Mesh Z-Coordinate Min (m) Max (m) -0.037	From Surface		
Iso-Values (m) 0 New Surface Name z-coordinate-25	From Zones		
Create Compute Manage Close Help			

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).

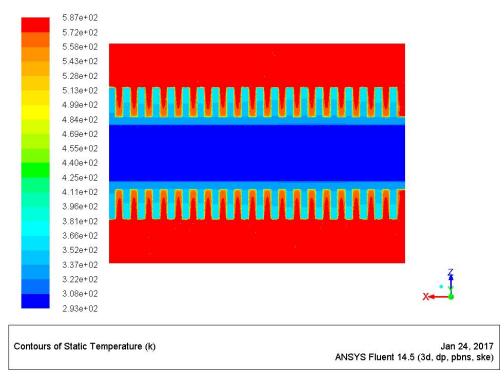


Figure 2.24 – Distribution of temperature fields in the longitudinal section of the heat exchanger

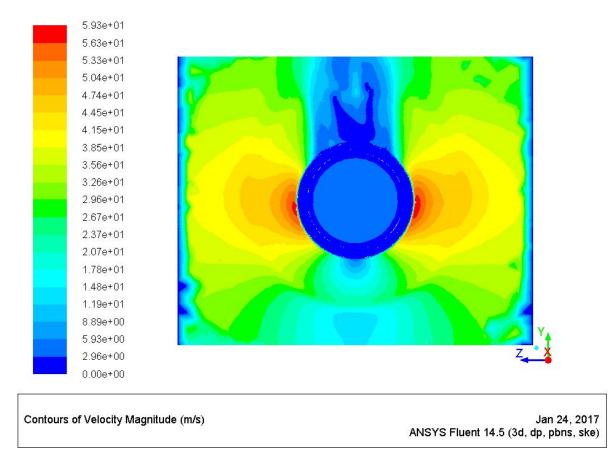


Figure 2.25 - Distribution of velocity fields in the cross section of the heat

exchanger

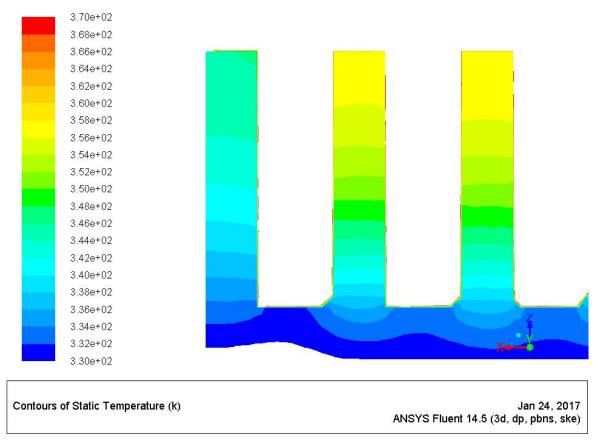


Figure 2.26 - Distribution of temperature fields in the longitudinal section of the

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.

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

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