VŠB - Technical University of Ostrava Faculty of Mechanical Engineering

MODELING OF HEAT, MASS AND MOMENTUM

Tutorials

Milada Kozubková Marian Bojko Veronika Mořkovská Patrik Marcalík

Ostrava 2020

Annotation

The aim is to briefly acquaint the student with the basic concepts of mass, momentum and heat transfer in applications to heat exchangers. The following is an illustrative example containing defining the problem, the physical properties of the flowing media, and the boundary conditions. With the help of Ansys - Fluent software, geometry preparation, calculation network creation, calculation and evaluation of results and their comparison with analytical solution are realized. The following is a series of examples to solve the procedure described above. After studying the module, the student should be able to describe the problem of the exchanger, build a physical and mathematical model, prepare the problem for numerical calculation, and perform and calculate the calculation. Then evaluate the numerical calculation with the analytical solution.

Table of contents

1	1	TRANSMISSION OF MASS, MOMENTUM AND HEAT4
2	2	HEAT EXCHANGERS
	2.1	2.1 Heat output
	The	heat output can be determined as the value calculated in Fluent
	2.2	Pressure loss
	2.3	Dimensionless criteria9
3	F	OURIER'S LAW - HEATING IN THE BAR13
	3.1	ANSYS Workbench14
	3.2	ANSYS DesignModeler14
	3.3	ANSYS Meshing20
	3.4	ANSYS Fluent
	3.5	Varianty výpočtů46
4	L	AMINARY FLOW - WATER FLOW BETWEEN PLATES51
	4.1	Creating geometry and mesh52
	4.2	Calculation in Fluent54
5	Т	URBULENT - WATER FLOW BETWEEN THE PLATES65
	5.1	5.1 Geometry and mesh66
	5.2	ANSYS Fluent67
6	S	AMPLE EXAMPLE SOLUTION - CO-CURRENT EXCHANGER74
	6.1	6.1 Mathematical model and theoretical-empirical estimation of the problem 75
	6.2	6.2 Geometry creation76
	6.3	6.3 Creation of mesh79
	6.4	ANSYS FLUENT
7	Н	EAT DISTRIBUTION OF CONDUCTIONS AND AIR CONVECTIONS109
	7.1	Mathematical model and theoretical-empirical estimation of the problem110
	7.2	Geometry and mesh creation111
	7.3	Calculating the Gravity Problem112
	7.4	Výsledky115

1 1 TRANSMISSION OF MASS, MOMENTUM AND HEAT

The basic laws of conservation of mass, momentum and energy are described by integral or partial differential equations with boundary and initial conditions that significantly influence the result of the solution. In a general conservative form, the form of the equations is as follows:

$$\iiint_{V} \frac{\partial(\rho\zeta)}{\partial t} dV + \iiint_{S} \left(\rho\zeta \vec{u} \cdot \vec{n} \right) dS = \iiint_{S} \left[\alpha_{\zeta} \nabla\zeta \right] dS + \iiint_{V} S_{\zeta} dV$$
(1.1)

accumulation + convection = diffusion + source

where ζ is a general variable and the terms in the equation are progressively convective (related to the velocity vector \vec{u}), diffusion and source term, therefore the equation is also called convection - diffusion equation.

This equation can be expressed in differential form (more common in hydromechanics and thermomechanics textbooks)):

$$\frac{\partial(\rho\zeta)}{\partial t} + \nabla \cdot \left(\rho \vec{u} \zeta\right) = \nabla \cdot \left[\alpha_{\zeta} \nabla \zeta\right] + S_{\zeta}$$
(1.2)
accumulation + convection = diffusion + source

If ζ represents temperature, admixture or other scalar quantity, then it is a second order linear equation, if ζ represents the velocity component, it is a nonlinear equation.

The task of finding a solution to equation (1.2) satisfying both boundary and initial conditions is called a mixed problem. If the boundary conditions are zero, they are called homogeneous boundary conditions, similarly if the initial conditions are zero, they are called homogeneous initial conditions. Instead of boundary conditions, other types of conditions, also called boundary conditions, may be given. Consideration of boundary and initial conditions for temperature is valid for the general variable. An analytical solution of such systems is possible only in significantly simplified applications. Therefore, the emphasis is currently on the numerical solution and in order to specify its possibilities.

Numerical modeling allows to solve various problems, eg.:

- planar two-dimensional flow, axially symmetrical flow, general threedimensional flow
- stationary, non-stationary and transient flow
- · laminar and turbulent flow in simple and complex geometries
- compressible and incompressible flow
- heat transfer, natural and mixed convection, radiation
- chemical transfer including chemical reactions, combustion
- multiphase flow, free-flow flow, solid-particle flow and bubbles
- flow through porous environment, etc.

Powerful CFD (Computational Fluid Dynamics) software systems, such as Ansys-Fluent, Ansys-CFX, OpenFoam, Star CCM +, etc. are available for this purpose.

With the development of computer technology, the requirements for its users are changing, especially in the field of designing. Recently, the knowledge leading to the right choice of the computational model, computational methods and interpretation of results has gained a significant advantage over the mathematical and programming aspects of the problem. It remains dedicated to top mathematics and programming specialists and problem-oriented software specialists.

The calculation method is based on the finite volume method. The user should know their nature to the extent necessary for reliable use in standard cases. In Fluent program it is necessary to know what shapes of finite volumes will be used, which implies the choice of mesh density, which approximation schemes it will be suitable to use, dynamics to have an idea of the time dependence of individual quantities and the resulting time step size, etc. it is necessary to understand the general diction of the manuals, because without this tool it is not possible to seriously process the assignment. Equally important is the evaluation of results, which is particularly difficult for three-dimensional tasks. It is optimal to have at least approximate values of calculated quantities, ideal is to compare the results with the experiment. This textbook should provide guidance on how to deal with the above problems.

2 HEAT EXCHANGERS

Heat exchangers are devices that ensure the transfer of internal thermal energy (enthalpy) between two or more fluids, between a solid surface and a fluid, or between particles and a fluid, in their interaction without the external work and heat supplied.





Fig. 2.1 - Scheme of fluid and heat flows through the exchanger (counterflow exchanger)

The fluids may generally be mono-constituent, or may be a mixture, both monophasic and binary. Typical applications are two-fluid heaters and coolers, where both fluids are separated by a solid wall, and evaporators in thermal and nuclear power plants. Typical exchangers can be divided into several groups

• Tubular, tubular, spiral (co - current, counter - current and cross - flow exchangers),

- Honeycomb exchangers,
- Plate heat exchangers.

The basic design parameters for heat exchanger description are heat output and pressure drop, which will be defined for simplicity according to the diagram in Fig. 2.1.

2.1 2.1 Heat output

The energy analysis is based on a calorimetric equation that describes the heat exchange of two or more objects. Thus, heat passes through the solid wall of the exchanger and subsequently also through the fluid and is then affected by the flow.

Heat conduction through a solid wall, ie heat output, is described by the following equation

$$P = \lambda \frac{t_{h,S} - t_{c,S}}{d} S$$
(2.1)

whre λ is the thermal conductivity coefficient [W·m⁻¹·K⁻¹], $t_{h,s}$ is the wall temperature, hot side, $t_{c,s}$ is the fixed wall temperature, cold side, *S* is the heat exchange surface [m²], *d* is a characteristic dimension [m]. However, there is a velocity and temperature boundary layer close to the wall. The temperature boundary layer is related to a heat transfer coefficient that defines how intensely the heat is transferred from the fluid to the solid wall or vice versa. The heat transfer equation for hot and cold walls is given by the following equations

$$P = \alpha_c (t_{c,s} - t_c) S$$

$$P = \alpha_h (t_{h,s} - t_h) S$$
(2.2)

where α_h is the heat transfer coefficient on the hot fluid side, α_c is the heat transfer coefficient on the cold fluid side, t_h is the temperature curve in the cooled liquid, t_c is the temperature curve in the heated liquid. Next, a quantity is called a **heat transfer coefficient**

$$k = \frac{1}{\frac{1}{\alpha_h} + \frac{d}{\lambda} + \frac{1}{\alpha_c}}$$
(2.3)

After the introduction of the heat transfer, the power equation is transformed into:

$$P = k(t_h - t_c)S \tag{2.4}$$

By analyzing the previous relation, it is possible to determine parameters that influence the heat exchanger performance. If the intention is to maximize performance, then the following conditions must be considered

1. The wall thickness should be as small as possible (this is the reason for the thin walls in the exchangers)

2. the thermal conductivity of the solid wall should be as large as possible (that is why materials with high thermal conductivity, aluminum, copper etc. are used)

3. the heat exchange surface should be as large as possible (that is why there are a large number of fins, honeycombs, small tubes below.)

4. the heat transfer coefficient should be as large as possible, its value can be partially influenced by the fluid velocity, but with increasing velocity they increase with the square of pressure loss.

When flowing through the pipe system, there is a significant change in temperature, then the heat output would be greatly overestimated when using a temperature difference $\Delta T = T_s - T_{ref}$. As the fluid moves through the pipe system, the wall temperature decreases and thus the temperature difference. Therefore, the so-called logarithmic temperature difference is used

$$\Delta T_{lm} = \frac{(T_s - T_l) - (T_s - T_o)}{\ln\left(\frac{(T_s - T_l)}{(T_s - T_o)}\right)}$$
(2.5)

where T_{I}, T_{O} are the inlet and outlet temperatures of the flowing medium. The outlet temperature that is needed to determine ΔT_{Im} can be estimated from the relationship

$$\frac{T_s - T_o}{T_s - T_i} = \exp\left(-\frac{\pi d N \overline{\alpha}}{\rho v N_T S_T C_p}\right) \Longrightarrow T_o = -\left(\exp\left(-\frac{\pi d N \overline{\alpha}}{\rho v N_T S_T C_p}\right) (T_s - T_i) - T_s\right)$$

where N is the total number of tubes in the system and is the number of pipes in the vertical plane, V is an estimate of the flow rate. Therefore ΔT_{lm} is known. Of course, when using numerical calculation, temperature values are determined as average values at the inlet and outlet edges.

The heat output per unit length of the pipe can be calculated from the relation

$$P = \mathcal{N}(\pi d \overline{\alpha} \Delta T_{lm}) \tag{2.6}$$

The heat output can be determined as the value calculated in Fluent.

2.2 Pressure loss

The power to be supplied to the fluid to flow through the exchanger in a given amount can be determined by the pressure drop from the following relationship:

$$P = \frac{Q_m \Delta \rho}{\rho}$$

$$P \approx \frac{1}{2} \frac{\eta}{\rho^2} \frac{4l}{d_h} f(\text{Re}) \qquad \text{for laminar flow} \qquad (2.7)$$

$$P \approx \frac{0.046}{2} \frac{\eta^{0.2}}{\rho^2} \frac{4l}{d_h} \frac{Q_m^{2.8}}{S_0^{1.8} d_h^{0.2}} \qquad \text{for turbulent flow}$$

/ is the length over which heat transfer occurs, d_h is the hydraulic diameter and S_0 is the minimum flow area of the exchanger.

In general, the pressure drop of the exchanger depends on the following parameters:

1. frictional losses associated with fluid flow around heat exchange surfaces and thus frictional (viscous) forces

2. torque effect, which is related to the change in density in the exchanger flow

3. Compression and expansion of fluid by flowing bodies (heat transfer surfaces))

4. geometric parameters of the exchanger (for large vertical exchanger it is necessary to include also the static pressure exerted by gravity, for gases this loss is neglected.

When flowing through a pipe system, the pressure drop is dependent on the loss coefficient of the pipe system and empirically determined.

$$\Delta \rho = N_{L} \zeta \left(\frac{\rho u_{\text{max}}^{2}}{2} \right) \text{ resp. } \Delta \rho = N_{L} \zeta \left(\frac{8Q_{m}^{2}}{\rho \pi^{2} d^{4}} \right)$$
(2.8)

The loss coefficient is specific to the different pipe arrangements. In the arrangement of the tubes in succession, it is defined as follows:

$$\zeta = \gamma \left(N_{L} \frac{S_{L}}{S_{T}} A + B \right)$$
(2.9)
Where $A = 0.028 \left(\frac{S_{T}}{2a} \right)^{2} a = \frac{S_{T} - d}{2} B = \left(\frac{S_{T}}{2a} - 1 \right)^{2}$

In cross-sectional arrangement of tubes, it is similarly defined:

$$\zeta = \gamma \left(0.7 + 0.8 \left(N_L \frac{S_L}{S_T} A + B \right) \right)$$

$$kde \ A = 0.028 \left(\frac{S_T}{2a} \right)^2 \ a = \frac{S_T - d}{2} \ B = \left(\frac{S_T}{2a} - 1 \right)^2$$
(2.10)

Coefficient γ depends on Reynolds number. For values higher than 40000 it is equal to one and for values lower than 40000 it is estimated from empirical measurements and is shown in Fig. 2.2.





As can be seen, the solution flow around such a pipe system is dependent on a series of empirically determined coefficients, the specification of which is not the object of this subject. In Fluent, the pressure drop is obtained directly using the average pressure values at the inlet and outlet edges. It is also possible to determine the backward coefficient, so it can be the result of the calculation.

$$\zeta = \frac{p_{1tot} - p_{2tot}}{p_{2dyn}}$$
(2.11)

2.3 Dimensionless criteria

When preparing the mathematical model it is necessary to decide on the type of flow and to compare the numerical solution with the analytical solution, therefore it is necessary to define dimensionless parameters such as:

Reynolds number (**Re**), which is determined from boundary and physical conditions to specify laminar or turbulent flow. Its value characterizes the flow in the transition region between laminar and turbulent flow [3].

$$Re = \frac{ud_h}{v}$$
(2.12)

where the so-called hydraulic diameter represents the diameter of the pipe when flowing in the pipe, while the pipe diameter also flows around the pipe, the mean velocity of the flowing medium. When flowing in a pipe, if Re <2320 is a laminar flow (particles move in layers). At higher Re> 2320 it is a turbulent flow (the particles swirl) [4].

The Prandtl number is only dependent on the material properties of the fluid. It refers to boundary layer thicknesses, reference speed and temperature.

$$\Pr = \frac{\rho \, c_{\rho} v}{\lambda} = \frac{v}{a} \tag{2.13}$$

For air, its value can be assumed to be constant 0.7.

The Fourier number is the ratio of heat conduction to its accumulation in a solid body

$$Fo = \frac{\lambda \tau}{c_{\rho} \rho d_{h}^{2}}$$
(2.14)

 τ is the time constant.

The Nusselt number expresses the effect of flow on the heat flow through the wall, and depends on the geometric reference parameter (which is well definable).

$$Nu = \frac{\alpha d_h}{\lambda}$$
(2.15)

Heat transfer coefficient α includes thermal conductivity λ solid walls that separate both fluids and a heat transfer coefficient $\alpha_{1,2}$ for the interface between the solid wall and the two fluids. However, this coefficient is dependent on both the material properties of the flowing fluid and the flow pattern around the solid wall.

The second definition of the Nusselt number contains better measurable quantities such as heat output P, characteristic dimension d_h , surface S, to which the heat transfer, the temperature gradient between the wall temperature and the reference ambient temperature is determined $\Delta T = T_s - T_{ref}$. The temperature gradient can also be specified as the mean log difference.

$$Nu = \frac{P d_h}{S \Delta T \lambda}$$
(2.16)

The heat transfer coefficient can be determined on the basis of a number of empirical relationships, and in practice the similarity theory is most often used. If we know the value of the Nusselt number we can determine the heat transfer coefficient. The Nusselt number is generally a function of other similarity criteria

$$Nu = f(Re, Pr, Fo)$$
(2.17)

In the case of forced convection, the value of the Nusselt number is determined by the value of the number.

Tab. 2.1 Forced convection

laminar flow around the	Nu = 0,664 Re _L ^{1/2} Pr ^{1/3}		$0,6 \le \Pr$				
	$\operatorname{Re}_{L} = \frac{uL}{v}, \ 10^{4} \le \operatorname{Re}_{L}$. plate length					
laminar flow around the	$Nu = 0,908 \operatorname{Re}_{L}^{1/2} \operatorname{Pr}^{1/3}$		0,6 ≤ Pr				
	$\operatorname{Re}_{L} = \frac{uL}{v}, \ 10^{4} \le \operatorname{Re}_{L}$	$\leq 5.10^{5}$, L	. plate length				
turbulent flow around the	Nu _x = 0.0405 Re ^{4/5} _L Pr	.1/3 ($0,6 \le \Pr \le 60$				
plate, 7818 constant	$5.10^5 \le \text{Re}_L \le 10^8$						
laminar flow in the tube	Nu=4.36 pro <i>q</i> =cor	nst. for wal	l				
	Nu=3.66 pro <i>T</i> =cor	nst. for wal	I				
turbulent flow in the pipe	$Nu = 0,023 \text{ Re}^{0.8} \text{ Pr}^{m}$	9	m=0.3 for co	oling			
	$3.10^4 \le \text{Re}_L \le 10^6$		m=0.4 for he	ating			
laminar, transitional and	$\mathrm{Nu} = C_1 \mathrm{Re}^{C_{\tilde{e}}} \mathrm{Pr}^{0.38}$						
turbulent transverse pipe wrap	Re	C1	C2				
	0,4 ÷ 4	0,989	0,330				
	4 ÷ 40	0,911	0,385				
	40 ÷ 4 000	0,683	0,466				
	4 000 ÷ 40 000	0,193	0,618				
	40 000 ÷ 400 000	0,0266	0,805				
laminar, transition and	$\operatorname{Nu}_{\mathrm{D}} = C_1 \operatorname{Re}_{D,\max}^m$	рі	ю	N_L $\rangle 10$,			
turbulent wrapping of the tube bundle, N_L is the	$2000 \le \operatorname{Re}_{D,\max}^{m} \le 4000$	Pr = 0.7	, constants	C₁ a <i>m</i>			
number of tubes	are given in the table	е					
	S_L – horizontal pipe spacing, S_T – vertical pipe spacing						

direct system	S⊤/D=	1.25	S⊤/D=	1.50	S⊤/D=	2.00	St/D=	3.00
S∟/D	C1	m	C1	т	C1	т	C1	т
1.25	0.348	0.592	0.275	0.608	0.100	0.704	0.063	0.752
1.50	0.367	0.586	0.250	0.620	0.101	0.702	0.068	0.744
2.00	0.418	0.570	0.299	0.602	0.229	0.632	0.198	0.648
3.00	0.290	0.601	0.357	0.584	0.374	0.581	0.286	0.608
system cross	St/D=	1.25	St/D	= 1.50	St/D=	= 2.00	St/D	= 3.00
S∟/D	C	m m	С	² 1 <i>m</i>	С	1 m	C	C1 m
1.000			0.49	7 0.558				
1.125					0.478	8 0.565	0.51	8 0.560
1.250	0.518	0.556	0.50	5 0.554	0.519	9 0.556	0.52	2 0.562
1.500	0.451	0.568	0.46	0.562	0.452	2 0.568	0.48	8 0.568
2.000	0.404	0.572	0.410	6 0.568	0.482	2 0.556	0.44	9 0.570
2 000								
3.000	0.310	0.592	0.350	6 0.580	0.448	3 0.562	0.48	2 0.574

A number of relationships can be found in the literature to determine the value of the Nusselt number. These equations are predominantly empirically determined and have limited validity in certain specific cases. In the previous text, only a very brief selection of the most commonly used relationships was given.

3 FOURIER'S LAW - HEATING IN THE BAR

Example

Solve the temperature distribution in the bar of given length (Fig. 3.1) in **ANSYS Fluent** program. The task is to create geometry, computational mesh (mesh onward), define physical model, physical properties of material, boundary and initial conditions, mathematical model in the programs **DesignModeler**, **ANSYS Meshing** and **ANSYS**

Fluent. The next step is to realize the numerical calculation and evaluate the calculated quantities.



Fig. 3.1 - Bar of defined length

The dimensions of the area are shown in Tab. 3.1 and physical properties of individual materials in Tab. 3.2.

Tab. 3.1 Geometry area

area length / [m]	0,5
area diameter D [m]	0,08

Tab. 3.2 Physical properties of the material (steel, aluminum, copper, wood) at 300 K

material	wood	steel	aluminium	copper
density ρ [kg·m ⁻³]	700	8030	2719	8978
specific heat capacity c_{ρ} [J·kg ⁻¹ ·K ⁻¹]	2310	502,48	871	381
thermal conductivity λ [W·m ⁻¹ ·K ⁻¹]	0,173	16,27	202,4	387,6

The boundary conditions are defined on the left wall (see Figure 3.1) by temperature T_0 and on the right wall (**"right wall"**) temperature T_l (Tab. 3.3). Outer wall (**"outer wall"**) or tube sheath is considered to be insulated $q = 0 W/m^2$.

Tab. 3.3 Boundary conditions

left wall	right wall	outer wall
$T_0 = 50^{\circ}C$	$T_l = -10^{\circ}C$	$q = 0 W/m^2$

Mathematical model

There is no flow in this task, so the flow with zero velocity is fictitious, ie as a laminar flow.

3.1 ANSYS Workbench

Start the program in **Start / All Programs / ANSYS 2019 R3 / Workbench 2019 R3**. After running the program in the menu toolbar on the left side of the window, double-click **Fluid Flow (Fluent)**, as shown in Figure 3.2. For example, name the newly created panel as bar (never use diacritics and mathematical symbols). Now save the entire project **File / Save as** to any directory under any name, again do not use diacritics and mathematical symbols.



Fig. 3.2 - Working environment of **ANSYS Workbench 2019 R3** with Fluid flow block.

3.2 ANSYS DesignModeler

In the first stage, you must create the geometry in **DesignModeler**. Right-click on "**Geometry**" and select "**New DesignModeler Geometry**" (see Figure 3.3). The working environment of **DesignModeler** is shown in Fig. 3.4..

啦 Tyc - Workbench



×





Fig. 3.4 - Program DesignModeler

Creating geometry

After launching **DesignModeler**, set the appropriate units, in your case it is meter - the "**Units - Meter**" pull-down menu. In this case, the model represents a simple cylinder of defined dimensions. The procedure for creating a 3D model is to create a simple cylinder shape using the "**Create - Primitives - Cylinder**" drop-down menu (Fig. 3.5).

Select the coordinate plane of the cylinder base (**XYPlane**), change the position of the center of the base (**Origin**), the axis length (**Axis**) and the radius according to Fig. 3.6.

DM A:	tyc -	Design	Modele	r													
File	Cre	ate C	oncept	Tools	Units	View	Help										
🔄	⊁	New P	lane			ю	Select	t: *13	\$ -	R (R R		3-		E]	S-
]		Extrud	e				# #										
XYPI	命	Revolv	/e			-	题]	誟 Ger	nerate	W S	Share	Topolo	gy	💦 Pa	aramet	ers	
Blade	6	Sweep				GD	<> Loa	ad NDF	1	Flow	Path	🥖 Bla	de	🥩 S	plitter	4	Vista [®]
		Skin/L Thin/S	oft Surface					•						•	9.1	1	1 🧭
Tree O ⊡…√		Fixed F Variabl Vertex Chami Pattern Body (Body 1 Boolea Slice Delete	Radius B le Radiu Blend fer n Dperatio Fransfor an	lend s Blenc n matior	1	•											
		Point															
		Primit	ives				Sphere Box Parallele Cone Prism Pyramic Torus Bend	epiped r									

Fig. 3.5 - Creating cylinder geometry

S	ketching Modeling										
D	Details View										
	Details of Cylinder1										
	Cylinder	Cylinder1									
	Base Plane	XYPlane									
	Operation	Add Material									
	Origin Definition	Coordinates									
	FD3, Origin X Coordinate	0 m									
	FD4, Origin Y Coordinate	0 m 0 m									
	FD5, Origin Z Coordinate										
	Axis Definition	Components									
	FD6, Axis X Component	0,5 m									
	FD7, Axis Y Component	0 m									
	FD8, Axis Z Component	0 m									
	FD10, Radius (>0)	0,08 m									
	As Thin/Surface?	No									

Fig. 3.6 - Setting the cylinder dimensions



Fig. 3.7 - Resulting geometry

Naming boundary conditions

Because the model is three-dimensional, the boundaries will be areas of the area (**cylinder**). In the first stage, the selection mode is changed to **Face** (see Figure 3.8).



Fig. 3.8 - Selection of type of face selection mode ("Face")

An example of naming the right_wall boundary condition as specified is shown in Figure 3.9, Figure 3.10. The desired area is highlighted and the "**Named selection**" menu is selected. In the second phase naming **NamedSel2** is done as "**right_wall**" and "**Generate**", see Fig. 3.10.



Fig. 3.9 - Area selection for naming the boundary condition ("**Named** Selection")

Tree Outline ☐	e e r1 Sel2 Body	4
Sketching Modeling	J	
Details View		Ф.
Details of NamedSel	2	
Named Selection	NamedSel2	
Geometry	Apply	Cancel
Propagate Selection	Yes	
Export Selection	Yes	
Include In Legend	Yes	

Fig. 3.10 - Naming the boundary condition

The boundary condition is newly displayed in the command tree under the newly created item "**right_wall**", see Fig. 3.11.



–…,√🚱 A: tyc

→ XYPlane → ZXPlane → YZPlane → Cylinder1 → ŵ right_wall → ŵ left_wall → ŵ 1 Part, 1 Body



Fig. 3.11 - Representation of the boundary condition "right_wall"

If there are multiple areas of the same meaning in the area (eg inlets for the pipe system in the exchanger), then all of them can be selected (using Ctrl) and named with one name.

Follow the same procedure to define and name the remaining boundary conditions ("**left_wall, outer_wall**"), which are shown in Figure 3.12.



Fig. 3.12 - Naming boundary conditions

Now the model geometry is complete and ready for the computer network creation in **ANSYS Meshing**. You can save the entire project from **DesignModeler** using the "**File / Save Project**" command and close the program. Go back to Workbench. You can save the entire project from the Workbench at any time with the "**File / Save**" command. If the geometry is created without errors, then the Geometry item has a green tick (Fig. 3.13).

File View Tools Units Extensions Jobs Help Import Import <t< th=""><th>🚾 Tyc - Workbench</th><th></th><th></th><th>-</th><th></th><th>×</th></t<>	🚾 Tyc - Workbench			-		×				
Import Import Import Impor	File View Tools Units Extensions Jobs Help									
	T 2 R R Project									
Toobox Analysis Systems Coupled Field Static Coupled Field Static Coupled Field Transient Design Assessment Eigenvalue Buckling Explicit Dynamics Fluid Flow-Eltruk flow(Fluent) Setup Solution Solution 	🕌 Import 🖗 Reconnect 🔮 Refresh Project 🍠 Update Project 📲 ACT Start Page									
□ Analysis Systems □ Coupled Field Static □ Design Assessment □ Eligenvalue Buckling • Eligenvalue Buckling <td>Toolbox 👻 🕂 🗙 Pr</td> <td>roject Schematic</td> <td></td> <td></td> <td></td> <td>чх</td>	Toolbox 👻 🕂 🗙 Pr	roject Schematic				чх				
 Filid Flow (Fluent) Filid Flow (Polynamic Response Hydrodynamic Response I C Engine (Fluent) Magnetostatic Spouštění ANSYS Meshingu 	Analysis Systems Analysis Systems Coupled Field Static Coupled Field Static Coupled Field Transient Design Assessment Eigenvalue Buckling Electric Explicit Dynamics Fluid Flow - Blow Molding (Polyflow) Fluid Flow - Blow Molding (Polyflow) Fluid Flow (Fourt) Fluid Flow (Polyflow) Fluid Flow (Polyflow) Harmonic Acoustics Harmonic Acoustics Harmonic Response Hydrodynamic Response IC Engine (Fluent) Magnetostatic Modal	I Image: Fluid Flow (Fluent) Image: Pluid Flow (Flow (Fluent) <td>Geometrie OK ANSYS Meshingu</td> <td></td> <td></td> <td></td>	Geometrie OK ANSYS Meshingu							

Fig. 3.13 - Workbench environment after creating geometry without errors

You can then proceed to the creation of a mesh in the **ANSYS Meshing** program, which runs from the **Workbench** environment similar to the **DesignModeler** program, see Figure 3.13.

3.3 ANSYS Meshing

In the project, double-click on the "**Mesh**" item to start the **ANSYS Meshing** program, which allows the generated components to be meshed (Figure 3.14). This may take several minutes depending on the model's complexity.



Fig. 3.14 - Environment of the ANSYS Meshing program

After you start the program and load the components, you have several options to create a mesh. From a simple scheme, basically just double-clicking "**Mesh**" (right click

and select the "Generate mesh" command (very simple automatic mesh according to preset parameters and for most of the cases unsatisfactory, (Fig. 3.15) up to a user-defined mesh shape



Fig. 3.15 - Creating a simple automatic network

In this application, the computational area is cylindrical, so regular hexahedron elements are used as the elements to create a so-called computational mesh compaction towards the **outer wall**. I.e. a combination of an automatic mesh with a user-defined mesh is used.

In this example, you will use three operations to create a mesh:

- • Automatic element size adjustment
- • Defining wall compaction parameters
- • Define the Sweep method

Automatic setting of mesh element type

Click on "**Mesh**" in the "**Outline**" panel of Figure 3.16 to get information about the meshing parameters in the "**Details of Mesh**" panel. There are many items in this panel. Click "**Sizing**" to get predefined element size information. You can change these values as you like. The values are given in units (meter), if they are given in millimeters, for example, it is necessary to change the units (in the "**Units**" drop-down menu). Redefine the element size in "**Element Size, Max Size, Defeature Size, Curvature Min Size**".

Dieplay		1
Display		ľ
Defaults		
Physics Preference	CFD	
Solver Preference	Fluent	
Element Order	Linear	
Element Size	1,e-002 m	
Export Format	Standard	
Export Preview Surface Mesh	No	
Sizing	^	1
Use Adaptive Sizing	No	1
Growth Rate	Default (1,2)	1
Max Size	1,e-002 m	1
Mesh Defeaturing	Yes	
Defeature Size	1,e-002 m	1
Capture Curvature	Yes	
Curvature Min Size	1,e-002 m	1
Curvature Normal Angle	Default (18,°)	1
Capture Proximity	No	1
Bounding Box Diagonal	0,54882 m	1
Average Surface Area	9,7006e-002 m ²	1
Minimum Edge Length	0,50265 m	1
Quality		1
Inflation		1
Assembly Meshing		1
Advanced		1

Fig. 3.16 - Details of Mesh

For cylindrical bodies resp. The Sweep method is used for crosslinking. Use the "**Method**" function in the "**Mesh / Insert / Method**" menu and the "**sweep**" method, which is suitable for the cylinder geometry, see Fig. 3.17.

	7	Context									
File	Home	Mesh	Display	Selec	tion	Auto	mation				
Duplicate	⊷Cut □Copy Paste Outline	× Delete Q Find ₽ <mark>₽</mark> Tree ▼	Generate Mesh	Lam Ar Coo Com	ed Se rdinai ment	election te System Inser	® Imag ⊈ Sectio Section Anno t	es▼ on Plane tation	<mark>₩ft</mark> Units	Worksheet	Keyframe Animation
Outline 👓							×××× ₽	□×	Q (२ 📦 📦	۵
Name Projec ⊡	ct* Iodel (A3) @ Geome [] Materia 32 Coordi	 Search Ou try als nate Systems 	utline 🗸 .	•							
\$		Insert		•	\$4	Method			1		
	1	Update			10	Sizing _F					-
	7	Generate N	lesh		Ų	Contact	Method	Control	the slave	ithen and	
		Preview Show		Þ	▲	Refinen Face Me		mesh ty meshes	pes used on scope	to generate ed entities.	
	7	Create Pino	h Controls		Q	Mesh C	(i) Pres	s F1 for h	eln.		
		Group All S	imilar Child	ren		Match (_
	٠	Clear Gene	rated Data			Pinch					
	alp	Rename		F2		Inflation	1				
		Start Recor	ding			Mesh Ni	unt				
					** ®	Contact Contact Node M	Match Gro Match erge Grou	pup	-		
					8; •••	Node M Node M	erge ove	_			

Giant. 3.17 - Inserting a method

Select the volume "Geometry". We must define "Source Face" in the "Details of Sweep Method" table. In "Src / Trg Selection" select "Manual Source". The source left_wall. The number of elements by length can be specified in "Type" ("number of division" and "sweep num divs"). The number of elements along the length of the region is inserted (eg number of division = 100). The settings and the resulting area are shown in FIG. 3.18.

De	etails of "Sweep Meth	nod" - Method 🗢 🖛 🕂 🗖 🗙	
	Scope	Commenter Calenting	-
-	Scoping Method	Geometry Selection	-
	Geometry	1 Body	
	Suppressed	No	
ŀ	Method	Sween	-
ŀ	Algorithm	Program Controlled	
ŀ	Element Order	Use Global Setting	
ŀ	Src/Trg Selection	Manual Source and Target	
ŀ	Source	1 Face	
	Target	No Selection	
	Free Face Mesh Type	Quad/Tri	
ŀ	Туре	Number of Divisions	
Ì	Sweep Num Divs	Default	
Ī	Element Option	Solid	
	Advanced		
	Sweep Bias Type	No Bias	
D	- 11 6 "C M1		
=	Scope Scoping Method	Geometry Selection	
ŀ	Geometry	1 Body	
al	Definition	*	
	Suppressed	No	
ļ	Method	Sweep	
	Algorithm	Program Controlled	
Ī	Element Order	Use Global Setting	
	Src/Trg Selection	Manual Source and Target	
	Source	1 Face	
ļ	Target	1 Face	
	Free Face Mesh Type	Quad/Tri	
ļ	Туре	Number of Divisions	
-	Sweep Num Divs	100	
_ -	Element Option	20110	
퀴		No Pipe	
Ľ	Sweep bias type	NO DIG2	

Giant. 3.18 - Setting parameters for the Sweep method

Then we can generate the mesh by clicking on the "Generate" command. The resulting mesh is shown in FIG. 3.19.



Giant. 3.19 - The resulting mesh

It can be seen that the net is not densified by the wall, which is useful in the case of turbulent flow. Therefore, the **Inflation** method is used to repair the mesh.

Defining Inflation parameters

Inflation is defined for the "**Source Face**" in the "**Mesh / Insert / Inflation**" menu. You can access this menu with the right mouse button (Fig. 3.20).



Fig. 3.20 - Selecting the "Inflation"

In general, the following parameters must be specified to create Inflation:

- the geometry (2D or 3D areas) where Inflation will be generated
- boundary at which Inflation will be created (in 2D it is the edge (line), in 3D it is the surface)
- Inflation parameters, ie reduction of the first cell at the border, number of Inflation layers (cells), growth factor characterizing the gradual increase in cell size,

The characteristics of the parameters **defining Inflation** are shown in Figure 3.21. The yellow highlighted "**No Selection**" must be selected from the model geometry. "**Geometry / No Selection**" represents the selection of the area (area or volume) where the Inflation will be. By changing the "**Geometry Selection**" you can select by region name. First, click on the "**No Selection**" window (go to "**Apply**", see Figure 3.22). Then select the area by clicking on the model (a green background will appear). Click "**Apply**" to confirm the result.

Details of "Inflation" - Inflation								
-	3 Scope							
	Scoping Method	Geometry Selection						
	Geometry	No Selection						
-	Definition							
	Suppressed	No						
	Boundary Scoping Method	Geometry Selection						
	Boundary	No Selection						
	Inflation Option	Smooth Transition						
	Transition Ratio	Default (0,272)						
	Maximum Layers	5						
	Growth Rate	1,2						
	Inflation Algorithm	Pre						

Fig. 3.21 - Inflation characteristics

Details of "Inflation" - Inflat	ion and a second se	
Scope		
Scoping Method	Geometry Selection	
Geometry	Apply	Cancel
- Definition		
Suppressed	No	
Boundary Scoping Metho	d Geometry Selection	
Boundary	No Selection	
Inflation Option	Smooth Transition	
Transition Ratio	Default (0.272)	
Maximum Layers	5	
Growth Rate	1.2	
Inflation Algorithm	Pre	

Fig. 3.22 - Selection of the area in which Inflation will be created

Follow the same procedure to define the boundary to which **Inflation** is defined. Define the edge in the "**Boundary**" item (Fig. 3.23). First, click in the "**No Selection**" field. Then select the edge of the model. Then click on the border and select "**Apply**". The result is shown in Fig. 3.23.

E	etails of "Inflation" - Inflatio Scope	n	
	Scoping Method	Geometry Selection	
	Geometry	1 Face	
Э	Definition		
	Suppressed	No	
	Boundary Scoping Method	Geometry Selection	
	Boundary	1 Edge	
	Inflation Option	Smooth Transition	
	Transition Ratio	Default (0.272)	
	Maximum Layers	5	
	Growth Rate	1.2	
ĺ	Inflation Algorithm	Pre	

Fig. 3.23 - Edge selection to define Inflation

Then define the Inflation parameters (Fig. 3.23)

- factor characterizing gradual cell size reduction 0,272
- number of layers (cells) of Inflation 5
- growth factor 1,2

Then we can generate the network by clicking on the "**Generate**" command. The resulting mesh, including densification, is shown in FIG. 3.24.



Giant. 3.24 - The resulting computing network

Save the project in ANSYS Meshing with the command "File / Save Project".

3.4 ANSYS Fluent

Before starting **ANSYS Fluent**, it is necessary to check whether the "**Geometry**" and "**Mesh**" items have a green tick. If this is not the case, then the "**Geometry**" or "**Mesh**" must be updated using the "**Update**" command. In this case it is necessary to perform the "**Update**" for the "**Mesh**" by the right mouse button (Fig. 3.25).



Fig. 3.25 - Illustration of the Update mesh check mark

The resulting project in the Workbench environment is shown in Figure 3.26.



Fig. 3.26 - Resulting panel project

The **ANSYS Fluent** program is started by double-clicking "**Setup**". When Fluent is started, the area dimension (**3D**) is verified and the calculation will be performed with the usual or double precision ("**Double Precision**"). Define "**Double Precision**" (Fig. 3.27). It is also advisable to set the parallel calculation in "**Processing Options / Parallel**" for larger numbers of cells. The number of cores is eg 4.

•	A Cited and A	
1	Fluid Flow (Fluent)	Double Precision
	Geometry	
	Mesh	
	🙀 Setup 😼 🖌	Fluent Launcher 2019 R3 (Settin D ×
5	Solution 🖓 🖌	
6	😥 Results 🛛 😨 🖌	ANSYS Fluent Launch
		Display Options Meshing Mode Image: Display Mesh After Reading Display Mesh After Reading Do not show this panel again Processing Option ACT Option Image: Serial Load ACT Parallel

Fig. 3.27 - Starting ANSYS Fluent 2019 R3

Then the **ANSYS Fluent** program opens (Fig. 3.28).

A:tyc Fluent@ntb_mech_koz30 [3d, dp, pbn	s, lam] [ANSYS Academic Teaching Introductory]			
🖉 🎕 🌐 2 🦸 🐧 🛱 🖺				
<u>F</u> ile Domain Phy	rsics User-Defined Solution Re	sults View P	Parallel Design 🔺	
Mesh Display i Info , Check, Quality ,	Scale Scale ↓ Transform ↓ ↓ Make Polyhedra ↓ ★ Make Polyhedra ★ Adjacency ★ Adjacency	Append Jack Mesh Replace Mesh Replace Zone	Mesh Models Adapt Image: Dynamic Mesh Image: Dynamic Mesh Image: Dynamic Mesh Image: Dynamic Mesh Image: Dynamic Mesh Image: Dynamic Mesh Image: Dynamic Mesh Image: Dynamic Mesh Image: Dynamic Mesh Image: Dynamic Mesh Image: Dynamic Mesh Image: Dynamic Mesh Image: Dynamic Mesh Image: Dynamic Mesh Image: Dynamic Mesh Image: Dynamic Mesh Image: Dynamic Mesh Image: Dynamic Mesh Image: Dynamic Mesh Image: Dynamic Mesh Image: Dynamic Mesh Image: Dynamic Mesh Image: Dynamic Mesh Image: Dynamic Mesh Image: Dynamic Mesh Image: Dynamic Mesh Image: Dynamic Mesh Image: Dynamic Mesh Image: Dynamic Mesh Image: Dynamic Mesh Image: Dynamic Mesh Image: Dynamic Mesh Image: Dynamic Mesh Image: Dynamic Mesh Image: Dynamic Mesh Image: Dynamic Mesh Image: Dynamic Mesh Image: Dynamic Mesh Image: Dynamic Mesh Image: Dynamic Mesh Image: Dynamic Mesh Image: Dynamic Mesh Image: Dynamic Mesh Image: Dynamic Mesh Image: Dynamic Mesh Image: Dynamic Mesh Image: Dynamic Mesh Image: Dynamic Mesh Im	en + Create & Manage
Outline View	Task Page		-	
Filter Text	General			
 Setup B General Models Models M Materials E Cell Zone Conditions Dynamic Mesh Reference Values Keference Values Methods Controls Report Definitions Controls Report Definitions Cell Registers Initialization Cell Acquisition Results Surfaces 	Mesh Scale Check Report Quality Display Units Solver Type Velocity Formulation Pressure-Based Density-Based Time Steady Transient Gravity			

Fig. 3.28 - Basic environment of the program ANSYS Fluent

In the first phase it is necessary to check the mesh by displaying all boundaries (**boundary conditions**) and the whole area by the command "**Domain / Mesh / Display**" (Fig. 3.29). Selecting all items in the "**Surfaces**" window will display the boundary conditions.



Fig. 3.29 - Mesh and boundary conditions check

It is also necessary to check the mesh size units with the command "**Domain / Mesh / Scale**" (Fig. 3.30). If the calculation area is created in other dimensions (mm, cm, ...), you can use the "**Scaling**" and "**Specify Scaling Factors**" commands to convert the dimensions into basic units of meters (m).

<u>F</u> ile Domain	n Phy	sics (User-Defin	ed	Solution	R	esults	View	v
	Mesh					Zones			Inter
🕀 Display		Scale	6	Combin	ne 🚽 🗮	Delete	+ Append	-	👯 Me
🚺 Info 🔔 👹		📣 Transform	, , Ç	Separa	ate 🚽 🖽	Deactivate	Replace M	1esh	Ove
Units Check	, Quality 🚽	🔶 Make Polyl	hedra 🦂	Adjace	ency	Activate	Replace Z	one	
Outline View		Task Page						×	-
Filter Text		General					(2	-
Setup		Mesh						L	<u> </u>
Ceneral General								-	Q+
Ø Models		Scale.		леск	Report Qua	ality			
Ar Materials Ar Materials Ar Materials	ditions	Display	U	nits					
Boundary Con	Scale Mesh							×	Q
Ø Dynamic Mesh	Domain Extents					Scaling			
📃 🖹 Reference Valu	Vmin (m) 0		Vmay (m)	0.5		Scaling			
(*) 🖾 Reference Fran		00005		0.5		Conve	rt Units	1	6 .
J× Named Expres	Ymin (m) -0.079	99985	Ymax (m)	0.08		⊖ Specir	y Scaling Factor	s	
% Methods	Zmin (m) -0.079	84678	Zmax (m)	0.07985	624	Mesh Was	Created In		A
🕺 Controls						<select></select>	-		*
🔯 Report Definiti	View Length Unit	In				Scaling Fa	ctors		
Monitors	m	•				X 1			
Cell Registers						Y 1			
(+) 🛱 Calculation Acti						Ζ 1			Ľ.
Run Calculatio						01			
Results						Scale	Unscale		E
Surfaces			_					e	
🔹 🥗 Graphics			C	lose F	lelp				6
PIOTS Animations					_				<u>-</u>
Annations Reports									_
Parameters & Custo	omization							l	

Fig. 3.30 - Checking the unit of dimensions

The next check concerns the number of cells in the mesh using the "**Domain / Mesh / Info / Size**" command. Subsequently, a row will appear in the text window (**Console**) with information about the number of cells (**Cells**), areas (**Faces**) and nodes (**Nodes**) of the mesh, see Figure 3.31.

```
writing right_wall (type wall) (mixture) ... Done.
writing outer_wall (type wall) (mixture) ... Done.
writing left_wall (type wall) (mixture) ... Done.
writing zones map name-id ... Done.
Mesh Size
Level Cells Faces Nodes Partitions
0 75500 229355 78477 1
l cell zone, 4 face zones.
```

Fig. 3.31 - Display of the number of cells, areas and nodes

The following is a check of the existence of negative volumes in the mesh with the command "**Domain / Mesh / Check / Perform Mesh Check**" (Fig. 3.32), which can occur in complicated geometries, in which case it is necessary to recreate the mesh.

```
Console
Domain Extents:
    x-coordinate: min (m) = 0.000000e+00, max (m) = 5.000000e-01
    y-coordinate: min (m) = -7.999985e-02, max (m) = 8.000000e-02
    z-coordinate: min (m) = -7.984678e-02, max (m) = 7.985624e-02
Volume statistics:
    minimum volume (m3): 1.785957e-08
    maximum volume (m3): 2.940157e-07
        total volume (m3): 1.002666e-02
Face area statistics:
    minimum face area (m2): 3.571914e-06
    maximum face area (m2): 5.880315e-05
Checking mesh......
Done.
```

Fig. 3.32 - mesh check for negative volumes

If all data is correct, proceed from left to right and top to bottom in the menu. Many of the essential commands that appear in the tab menu are also in the left drop-down panel (Figure 3.33).



Fig. 3.33 - Menu of commands defining a mathematical model

The first commands from the "**Solver**" menu ("**Physics / Solver**") define the solver type, "**Time-Steady**" for a time-independent solution. Next, define "**Type-Pressure-Based**", "**Velocity Formulation-Absolute**". The setting of the "**Solver**" commands is shown in Figure 3.34. It is also possible to define the external force (eg gravity) by accelerating the Gravity in any direction and change the physical units from the SI system to another set of units or only units of selected quantities.



Fig. 3.34 - Commands from the "Solver"

Other commands are from the menu "**Models**" ("**Physics / Models**"), which defines the physical nature of the task according to very illustrative menu, ie "**Multiphase**", "**Energy**", "**Species**", "**Discrete Phase**" respectively. "**Viscous**", where laminar flow, turbulent flow can be defined using various turbulent models, and a special case of the ideal liquid flow "**Inviscid**" can be solved (Fig. 3.35)).

<u>F</u> ile	Domain		Physics		User-Defined	S	olution	Res	ults	V
General	Solver Operating Co	nditio	ons	nergy	Radiation	N nger	4odels Multiph Č, Specie	1ase 15	네 Structu 디니 Acousti	ire ics
Outline View		Т	ask Page		<u> </u>				ooo more	+
Filter Text	eral lels Multiphase (Off) Energy (Off) Viscous (Laminar) Radiation (Off) Heat Exchanger (O Species (Off) Discrete Phase (Off) Solidification & Me Acoustics (Off) Structure (Off) Eulerian Wall Film (Electric Potortial (C	ff)) ltii	Models Multiphase Energy - Of Viscous - La Radiation - Heat Exchar Species - O Discrete Phi Solidificatio Acoustics - Structure - (Eulerian Wa Electric Pote	- Off f minar Off n & N Off Dff II Film ntial	r • Off Off Melting - Off n - Off - Off					

Fig. 3.35 - Characteristics of the "Models"

This problem solves the problem of heat transfer, ie define only the energy equation "**Energy**". There is no flow in the task, so there is a fictitious solution of zero velocity flow as a laminar "**Laminar**" (Fig. 3.36).

Task Page 🛞	🖸 Viscous Model 🛛 🗙
Task Page Models Multiphase - Off Energy - On Viscous - Laminar Radiation - Off Heat Exchanger - Off Species - Off Discrete Phase - Off Solidification & Melting - Off Acoustics - Off Structure - Off Eulerian Wall Film - Off Electric Potential - Off	Viscous Model Model Inviscid Laminar Spalart-Allmaras (1 eqn) k-epsilon (2 eqn) K-omega (2 eqn) Transition SST (4 eqn) Reynolds Stress (7 eqn) Scale-Adaptive Simulation (SAS) Detached Eddy Simulation (DES) Large Eddy Simulation (LES) Options Viscous Heating Low-Pressure Boundary Slip
Edit	OK Cancel Help

Fig. 3.36 - Setting the mathematical model of the solved problem

Defining rod material

Fluid type defines flowing medium (water, air,...). The "**Solid**" type defines a solid material (steel, copper,...). In this example, we define the heat transfer in a solid (material "**Solid**"), the type of material is steel. Define the material using the "**Physics** / **Materials / Create / Edit Materials**" command, see Figure 3.37.

Resul	ts Vi	iew I	Parallel	Desig	jn 🔺		
ж	Af Structure Acoustics More ↓	Materials	Phase List/Sho Theract Add Pha	es w All ions ase	Zones Cell Zones Boundaries Profiles	Model Specific Discrete Phase DTRM Rays Shell Conduction	
۵	Create/Edit M	1aterials					×
ĺ	Name air			Material fluid	I Туре	*	Order Materials by Name Charginal Seconds
	Chemical Formul	la		Fluent F	luid Materials	*	Fluent Database
	r	Properties		none		User-Defined Database	
		D	ensity (kg/m3)	constant			▼ Edit ▲
		Cp (Specific	Heat) (j/kg-k)	constant			▼ Edit
	100 Thermal Conductivity (w/m-k) co		1006.43 constant		▼ Edit		
	0.0		0.0242			V Edit	
		VISC	Cha	inge/Crea	te Delete C	llose Help	¥

Fig. 3.37 Characteristics of the "Materials"

Select "**Solid**" in the **ANSYS Fluent** database ("**Fluent Database**") and change the "**Material Type**" menu to "**Solid**". Next, select "**steel**" in the "**Fluent Solid Materials**" menu. Confirm the move to the mathematical model using the "**Copy**" command, see Figure 3.38. Physical properties (Density, Specific Heat, Thermal Conductivity,...) are visible at the bottom of the menu and may vary according to the solver's requirements.

Name Mater			Orde	er Materials by	
el	solid		• •	Name	
emical Formula	Fluent Solid Mat	erials	0	Chemical Formul	а
	steel		•	Fluent Databa	
	Mixture			Fluent Databas	se
	none			er-Defined Data	abas
Properties					
🗾 Fluent Database Mate	erials				×
Fluent Solid Materials	[1/13]		Material Type solid		•
gypsum (caso4_2h20) nickel (ni)))	*	Order Materi	als by	
steel			O Chemical	Formula	
Copy Materials from Properties	Case Delete				
Copy Materials from Properties D	Delete			• View	4
Copy Materials from Properties D	Delete Density (kg/m3) constant 8030			View	*
Copy Materials from Properties D Cp (Specific	Delete Density (kg/m3) constant 8030 c Heat) (j/kg-k) constant			View View	•
Copy Materials from Properties D Cp (Specific	Delete Density (kg/m3) constant 8030 c Heat) (j/kg-k) constant 502.48			View	•
Copy Materials from Properties D Cp (Specific Thermal Condu	Delete Density (kg/m3) constant 8030 c Heat) (j/kg-k) constant 502.48 uctivity (w/m-k) constant			View View View View	*
Copy Materials from Properties D Cp (Specific Thermal Condu	Delete Density (kg/m3) constant 8030 c Heat) (j/kg-k) constant 502.48 uctivity (w/m-k) constant 16.27			 View View View 	*
Copy Materials from Properties D Cp (Specific Thermal Conductivit	Delete Density (kg/m3) constant 8030 c Heat) (j/kg-k) constant 502.48 uctivity (w/m-k) constant 16.27 ty (siemens/m) constant			 View View View View View 	*
Copy Materials from Properties D Cp (Specific Thermal Conductivity	Delete Density (kg/m3) constant 8030 c Heat) (j/kg-k) constant 502.48 uctivity (w/m-k) constant 16.27 ty (siemens/m) constant 8330000			 View View View View 	•

Fig. 3.38 - Steel selection from ANSYS Fluent database

As a result, the "steel" material is moved to the "Materials" item (Fig. 3.39)

Task Page	×
Materials	?
Materials	
Fluid	
air	
Solid	
steel	
aluminum	

Fig. 3.39 "Steel" material in the "Materials"

The final assignment of the steel material to the area is done by the command "**Physics / Zones / Cell Zone Conditions**", see Figure 3.40. First select "**Type**" "**solid**". Then select "**Material Name**" - "**Steel**" and confirm with OK (Fig. 3.40).
Task Page	۵		
Cell Zone Conditions	(?)	÷	
Zone Filter Text		- <u></u>	
solid		Q	
Solid			XXXAALALMXXIIII
Zone Name			
solid			
Material Name steel	Edit		
Frame Motion Source	ce Terms		
Mesh Motion Fixed	Values		Ś
Reference Frame	Mesh Motion	Source Terms	Fixed Values
Rotation-Axis Origin		Rotation-Axis Direction	
X (m) 0		- X0	•
Y (m) 0		▼ Y0	•
Z (m) 0		▼ Z1	•
	ОК	Cancel	
Phase Type	ID		
mixture 💌 solid	▼ 2		

Fig. 3.40 - Characteristics of the Cell Zone Conditions command

Defining boundary conditions

We define the boundary conditions using the menu "Physics / Zones / Boundary Conditions", see Figure 3.41.



Fig. 3.41- Boundary Conditions and types of boundary conditions

The default type of boundary condition in **ANSYS Fluent** is wall. If we name a certain boundary condition in the **ANSYS Meshing** program according to the **ANSYS Fluent** conventions, a specific type will be assigned to this condition. The types of boundary conditions can be defined according to the menu, see Figure 3.41. E.g. the axis condition is assigned the axis condition type ("**axis**"). Furthermore, for naming inlet, the type of velocity inlet of the flowing medium is assigned to the area, and for the naming outlet is assigned the type of output (**pressure outlet**) of the flowing medium from the area, etc..

Specification of boundary conditions

• left wall – type "wall" - "Edit" ($T_0 = 50 \circ C = 323.15K$), see Figure 3.42.

Task Page	a 🚺 Wall	×
Boundary Conditions	Zone Name left_wall Adjacent Cell Zone	
Zone Filter Text	solid	
interior-solid	Momentum Thermal Radiation Species DPM Multiphase UDS Wall Film Potential	Structure
left_wall outer wall	Thermal Conditions	
right_wall	O Heat Flux Temperature (k) 323.15	
	Temperature Wall Thickness (m)	-
	Convection	I
	Mixed I Layer	Edit
	O via System Coupling	
	💛 via Mapped Interface	
	Material Name	
	aluminum 💌 Edit	
	OK Cancel Help	

Fig. 3.42 - Defining boundary condition "left_wall"

• **right_wall** – type **"wall" - "Edit"** $T_l = -10^{\circ}C = 263.15K$, see Figure 3.43.

Task Page	🖻 📑 Wall	×
Boundary Conditions	Zone Name right_wall Adjacent Cell Zone	
Zone (Filter Text	solid	
interior-solid	Momentum Thermal Radiation Species DPM Multiphase UDS Wall Film F	Potential Structure
left_wall outer_wall	Thermal Conditions	
right_wall	O Heat Flux Temperature (k) 263.15	
	Temperature Wall Thickness (m)	
	O Convection	
	Radiation Heat Generation Rate (w/m3)	•
	Mixed Shell Conduction 1 Layer	Edit
	🔿 via System Coupling	
	🔿 via Mapped Interface	
	Material Name aluminum Edit OK Cancel Help	

Fig. 3.43 - Defining the boundary condition "right_wall"

• outer_wall – type "wall" - "Edit" $q = 0 W/m^2$, see Figure 3.44.

Task Page 🛞	🖸 Wall	×
Boundary Conditions	Zone Name outer_wall Adjacent Cell Zone solid	
interior-solid left_wall outer_wall right_wall	Momentum Thermal Radiation Species DPM Multiphase UDS Wall Film Potential Thermal Conditions 	Structure

obr. 3.1 – Definování okrajové podmínky "outer_wall"

Initialization

Subsequently, the Standard Initialization of the computational area is performed; Define initial conditions for the entire region using the "**Solution / Initialization / Method**" command. Define by default "**Method-Standard**" initialization using the "**Options**" menu, see Figure 3.45. In this task we define only temperature. Define the mean temperature value T = 293.15K. Confirm initialization with the "**Initialize**" button (Figure 3.45).

Outline View	Task Page	×
Filter Text	Solution Initialization	(?)
⊖ Setup	Initialization Methods	
 ✓ General ✓ Ø Models (+) ₱ Materials 	 Hybrid Initialization Standard Initialization 	
Cell Zone Conditions	Compute from	
Boundary Conditions	-	Potvrzení
📀 🖽 Internal	Reference Frame	inicializace
(*) 📑 Wall Ø Dynamic Mesh 🔁 Reference Values	Relative to Cell Zone Absolute	/
📀 🔽 Reference Frames	Initial Values	
Named Expressions	Gauge Pressure (pascal)	
% Methods	0	
🔀 Controls	X Velocity (m/s)	
Report Definitions	0	
(*) Q Monitors	Y Velocity (m/s)	
Ditialization	0	
	Z Velocity (m/s)	
Run Calculation	0	
Results	Temperature (k)	
 Graphics Image: Plots 	293.15	
 Animations Reports Parameters & Customization 	Initialize Reset Patch	
	Reset DPM Sources Reset Statistics	
	VOF Check	

Fig. 3.45 - Initialization of the computation area

Calculation

After initialization, the iteration calculation is started with the "**Solution / Run Calculation**" command, see Figure 3.46. Number of Iterations must be specified. The predefined value is 0. Enter a value quite high, eg 1000, when it is assumed that convergence will be achieved.



Figure 3.46 - Run "Canculation command"

The iteration calculation is then started with the "**Calculate**" button. Convergence can be monitored both graphically and numerically (Fig. 3.47). Since we do not solve the flow, the components of speed and continuity are not calculated. Only the temperature (**energy**) is calculated and as soon as the desired accuracy ("**Results / Residuals**") is reached, the calculation is terminated by a note that the solution is converged, see Figure 3.47



Fig. 3.47 - Progression of convergence

Evaluation of calculation

First we need to create a longitudinal section of the geometry. We display the geometry using the command "**Domain / Display**" to find out in which axis we have to create the section. To create a section, use the command "**Domain / Surface / Create / Iso-Surface**". Select "**Mesh**" as the "**Surface of Constant**" and select the z-axis (see Figure 3.48). Click Compute to find the coordinates in the z-axis. Since we want to have a cross section in the middle of the cylinder, enter a value in Iso-Values that is in the middle of these coordinates. Click **Create** to confirm the slice.



Giant. 3.48 - Creating a section (Iso-Surface)

For the sake of clarity, evaluation options are presented, ie filled temperature isocurves, other variables are meaningless, even if they are offered, such as pressure, speed, etc. The isocurves are evaluated in the created longitudinal section. The setting of temperature isocurve plots is shown in Fig. 3.49.

Contours		×
Options	Contours of	
✓ Filled	Temperature	•
✓ Node Values	Static Temperature	•
Global Range	Min (k) Max (k)	
Clin to Range	263.15 323.15	
Draw Profiles	Surfaces Filter Text)
Coloring Banded	left_wall outer_wall rez-podelny right_wall	
Smooth Levels Setup 20 1 1		
	New Surface	

Fig. 3.49 - Setting the drawing of temperature isolines by command "Contours"

The result of the temperature isocurve evaluation is shown in Figure 3.50, where a linear temperature drop from 323.15K to 263.15K can be seen. This is in line with the analytical solution (a line connecting the temperature boundary values)).



Fig. 3.50 - Temperature distribution in the whole area [K]

In addition, the temperature distribution along the area can be evaluated using the "**Results / Plots / XY Plot**" command, see Figure 3.51. Select "**Temperature / Static Temperature**" in the "**Y Axis Function**" menu and select the longitudinal section of the area in "**Surfaces**". In the "**Plot Direction**", enter the correct direction in which the cylinder geometry lies.



Fig. 3.51 - Characteristics of "XY Plot command"

A plot of the temperature distribution along the length of the area is shown in Figure 3.52.



Fig. 3.52 - Temperature distribution along the length of the areas

It is very interesting to evaluate the amount of heat passing through the left wall and right wall. The evaluation is performed by the command "**Results / Reports / Fluxes**", see Figure 3.53. Select "Total Heat Transfer Rate" in the "**Options**" menu and select

left wall and right wall in the "**Boundaries**" menu. The resulting values are listed under "**Results**" and Tab. 3.4.

Flux Reports		×
Options		
O Mass Flow Rate	Boundaries Filter Text 🔂 🗟 🐺	Results
 Total Heat Transfer Rate 	interior-solid	
Radiation Heat Transfer Rate	left_wall	45.26611387269372
	outer_wall	
	right_wall	-44.39726345435234
Save Output Parameter		Net Results (w)
		0.8688504
	Compute Write Close Help	

Fig. 3.53 - "Fluxes"

Tab. 3.4 – Heat passing through the wall

Heat passing through the wall Q [W]	steel
left wall	45.27
right wall	-44.40

.

The heat transfer through the wall elements in units $[W \cdot m^{-2}]$ can also be evaluated in detail at each wall location. In this simple case, it is constant because the temperature distribution is linear in the z direction and the mesh is lengthwise with a constant step, so there is a single slope (the temperature derivative is a flow), but in general geometry it will not. To do this, use the "**Results / XY Plot**" command, see Figure 3.54. In the "**Plot Direction**" menu, define X = 0, Y = 1, Z = 0, in the "**Y Axis Function**" menu, select "**Wall Fluxes / Total Surface Heat Flux**" and in "**Surfaces**" select **left_wall** and **right_wall**. The graph shows a value line inside the surface and several values that are on the border and modified in connection with the outer wall boundary condition"



Fig. 3.54 - Distribution of heat flow through walls "left wall and right wall"

3.5 Varianty výpočtů

In other variants of numerical calculations, first define a different material (Table 3.5) of the calculation area (bars). Perform numerical calculations and compare the results as shown in the example.

Tab. 3.5 - Physical properties of material (steel, aluminum, copper, wood))

material	Wood	Steel	Aluminium	copper
density $ ho$ [kg·m ⁻³]	700	8030	2719	8978
specific heat capacity C_{ρ} [J·kg ⁻¹ ·K ⁻¹]	2310	502.48	871	381
thermal conductivity λ [W·m ⁻¹ ·K ⁻¹]	0.173	16.27	202.4	387.6

Then define the variations of the different temperature boundary conditions on the **left** wall and **right wall** as shown in Tab. 3.6.

	OKRAJOVÉ PODMÍNKY				
Varianta	left wall	right wall	right wall	right wal	I
	<i>T</i> ₀ [° <i>C</i>]	$T_l [°C]$	$q_l[W.m^{-2}]$	$\alpha[W.m^{-2}.K^{-1}]$	$T_{\infty}[^{\circ}C]$
A	50	-10			
В	-20	100			
С	50		162700		
D	50		0		
E	50			1000	-10

Tab. 3.5 – Variations of boundary conditions on left wall and right wall

Where T_0 is the temperature at **"left wall"**

 T_l is the temperature at **"right wall"**

- q_l is the specific heat flux at **"right wall"**
- T_{∞} is the ambient temperature
- α is the heat transfer coefficient to **"right wall**"
- Prepare a solution area consisting of three rods of different diameters, when defining the geometry, always use the movement of the coordinate system to the end of the rod ("Create / New Plane"). The boundary conditions are the same. When sweeping, use the sweep method on the first and third rods, the second rod relative to a different diameter is transmitted only by inflation

Geometry creation:

tails View		1
Details of Pipe22		
Cylinder	Pipe22	
Base Plane	XYPlane	
Operation	Add Frozen	
Origin Definition	Coordinates	
FD3, Origin X Coordinate	0 m	
FD4, Origin Y Coordinate	0 m	
FD5, Origin Z Coordinate	0 m	
Axis Definition	Components	
FD6, Axis X Component	0.3 m	
FD7, Axis Y Component	0 m	
FD8, Axis Z Component	0 m	
FD10, Radius (>0)	0.01 m	
As Thin/Surface?	No	

Details of EndPipe22		
Plane	EndPipe22	
Sketches	0	
Туре	From Face	
Subtype	Outline Plane	
Base Face	Selected	
Use Arc Centers for Origin?	Yes	
Transform 1 (RMB)	None	
Reverse Normal/Z-Axis?	No	
Flip XY-Axes?	No	
Export Coordinate System?	No	

Details View

~		т
-	Details of Pipe32	
	Cylinder	Pipe32
	Base Plane	EndPipe22
	Operation	Add Frozen
	Origin Definition	Coordinates
	FD3, Origin X Coordinate	0 m
	FD4, Origin Y Coordinate	0 m
	FD5, Origin Z Coordinate	0 m
Axis Definition		Components
	FD6, Axis X Component	0 m
	FD7, Axis Y Component	0 m
	FD8, Axis Z Component	0.05 m
	FD10, Radius (>0)	0.015 m
	As Thin/Surface?	No

.

Sketching Modeling					
etails View	4				
Details of EndPipe32					
Plane	EndPipe32				
Sketches	0				
Туре	From Face				
Subtype	Outline Plane				
Base Face	Selected				
Use Arc Centers for Origin?	Yes				
Transform 1 (RMB)	None				
Reverse Normal/Z-Axis?	No				
Flip XY-Axes?	No				
Export Coordinate System?	No				



tails View	1
Details of PipeOut22	
Cylinder	PipeOut22
Base Plane	EndPipe32
Operation	Add Frozen
Origin Definition	Coordinates
FD3, Origin X Coordinate	0 m
FD4, Origin Y Coordinate	0 m
FD5, Origin Z Coordinate	0 m
Axis Definition	Components
FD6, Axis X Component	0 m
FD7, Axis Y Component	0 m
FD8, Axis Z Component	0.2 m
FD10, Radius (>0)	0.01 m
As Thin/Surface?	No



Boundary conditions:



Meshing:

Inflation to the **right_wall**, the border is a circle, sweep is applied only to the third tube (the next tube changes the diameter, so it is not possible to continue with the sweep)



Inflation on the left_wall, the border is a circle, the sweep is applied only to the first pipe



Inflation on the volume of the second tube, the boundary is the area of the tube



The resulting mesh



The next calculation is based on the previous task.

4 LAMINARY FLOW - WATER FLOW BETWEEN PLATES

Example

Solve water flow between two infinitely large plates, see Figure 4.1. The physical model is given by the shape of the area, the type of flow and the hydraulic flow parameters. Define the numerical calculation in ANSYS Fluent. Use DesignModeler and **ANSYS Meshing** to create the computational area (geometry) and computational mesh.



Fig. 4.1 - Area diagram

The water flows into the area at the speed of $0.05 \ m.s^{-1}$ and exits into the atmosphere where the relative pressure is 0 Pa. The problem is given as a 3D model and represents the flow in a cuboid area of given length, thickness and width, see Tab. 4.1. The physical properties of the flowing medium are given in Tab. 4.2.

area length / [m]	0.5	
height of the area s [m]	0.02	
area width b [m]	0.1	

Tab.	4.2 -	Physical	properties	of water
------	-------	----------	------------	----------

Tab. 4.1 – Geometry area

density of water ρ [kg.m ⁻³]	998
dynamic viscosity η [kg.(m.s) ⁻¹]	0.001003

Boundary conditions

VELOCITY INLET is defined on the "inlet" and a static pressure condition (**PRESSURE-OUTLET**) on the "**outlet**". On the walls ("**top wall, bottom wall**") there is a boundary condition of the type WALL where zero flow velocity is assumed (it is predefined). A side condition is defined by a boundary condition of the **SYMMETRY** type (infinitely large boards). The marginal conditions are given in Tab. 4.3.

inlet – medium speed U_s [m.s ⁻¹]	0.05
outlet – static pressure p [Pa]	0

Mathematical model

The selection of the mathematical model will be solved in the next chapters, now it will be left predefined (laminar flow model).

The laminarity criterion is Reynolds number:

 $\operatorname{Re} = \frac{u.d}{v} = \frac{0.05 \cdot 0.02}{1.10^{-6}} = 1000$

The flow is therefore laminar.

4.1 Creating geometry and mesh

In the **Workbench**, select "**Fluid Flow / Fluent**" and drag it to the work window. Right-click on "**Geometry**" and select "**New DesignModeler Geometry**". Create the cuboid geometry with given dimensions using the "**Create / Primitives / Box**" command (Fig. 4.2). Confirm the box by clicking on "**Generate**".



Giant. 4.2 - Creating cuboid geometry

In the next phase, name the boundary conditions as described in Figure 4.1. To rename boundary conditions, use the "**Named Selection**" command. The resulting marking and naming of all boundary conditions is shown in Fig. 4.3.



Fig. 4.3 - Marking of boundary conditions

Networking is done in **ANSYS Meshing**. Since it is a flow between the boards, it is necessary to insert on the inlet Inflation surface to both walls (**top wall, bottom wall**). Create Inflation for both edges at the same time as in chap. 3.3 using the "**Meshing / Inflation**" command (the parameters are shown in Figure 4.4). Then use the "**Sweep**" method according to chap. 3.4. The parameters are shown in Fig. 4.4. Then generate a new mesh with the "**Generate Mesh**" command. The resulting form of the mesh including meshing parameters is shown in Fig. 4.5.

			D)etails of "Sweep Method" - Method 🛛 🕶 🖛 🖬 🗙			
			-	∃ Scope			
D	etails of "Inflation" - Inflatio	n 🗤 🕂 🗖 🗖 🗙		Scoping Method	Geometry Selection		
				Geometry	1 Body		
	scope		_ =	Definition			
	Scoping Method	Geometry Selection	_	Suppressed	No		
	Geometry	1 Face		Method	Sweep		
	Definition		_	Algorithm	Program Controlled		
	Commenced No.		-	Element Order Use Global Setting			
	Suppressed	NO		Src/Trg Selection	Manual Source		
	Boundary Scoping Method	Geometry Selection	_	Source	1 Face		
	Boundary	2 Edges		Target	Program Controlled		
	Inflation Option	Smooth Transition	-	Free Face Mesh Type	All Quad		
	Transition Batio	0.2		Туре	Number of Divisions		
		10	-	Sweep Num Divs	200		
	Maximum Layers	10	_	Element Option	Solid		
	Growth Rate 1.2		Ξ	Advanced			
	Inflation Algorithm	Pre		Sweep Bias Type	No Bias		

Fig. 4.4 - Inflation parameters and Sweep method

0	utline		
	Name	✓ Search Outline ✓	-
D	etails of "Mesl	sh"	
F	Display		
	Display Style		Use Geometry Setting
-	Defaults		
	Physics Prefer	rence	CFD
	Solver Prefere	ence	Fluent
	Element Orde	er	Linear
	Element Si	Size	3.0 mm
	Export Format	at	Standard
	Export Preview	ew Surface Mesh	No
-	Sizing		
	Use Adaptive	e Sizing	No
	Growth Ra	late	Default (1.2)
	Max Size		3.0 mm
	Mesh Defeatu	turing	Yes
	Defeature	e Size	3. mm
	Capture Curva	vature	Yes
	Curvature	Min Size	3.0 mm
	Curvature	Normal Angle	Default (18.0°)
	Capture Proxi	kimity	N0
	Bounding Bo	ox Diagonal	510.29 mm
	Average Surfa	race Area	20667 mm*
	Minimum Edg	ge Length	20.0 mm
±	Quality		
	Lise Automati	tic Inflation	None
	Use Automati	tic Inflation	None Smooth Transition
	Transition	uon Ratio	
	Maximum		5
	Growth Pa	late	12
		orithm	Pre
	View Advance	red Options	No
	Assembly Me	eshina	
	Method	2	None
Ξ	Advanced		
	Number of CF	PUs for Parallel Part Meshing	Program Controlled
	Straight Sideo	ed Elements	-
	Rigid Body Be	ehavior	Dimensionally Reduced
	Triangle Surfa	face Mesher	Program Controlled
	Topology Che	iecking	Yes
	Pinch Toleran	nce	Default (2.7 mm)
	Generate Pino	nch on Refresh	No
-	Statistics		
	Nodes		148512
	Elements		137775
D	etails Section	an Planer	

Fig. 4.5 - The resulting form of a computer network for flow between boards

4.2 Calculation in Fluent

After creating the mesh, go back to the **Workbench**, Fig. 4.6. Before running the **ANSYS Fluent** program, it is necessary to update the mesh by the command "**Update**" for the item "**Mesh**" with the right mouse button (a green check should have appeared). The ANSYS Fluent program is started by double-clicking "**Setup**". Do not forget to set the calculation with a higher order of precision "**Double precision**" and parallel calculation using "**Processing Options / Parallel**".

🔤 laminarní proudeni mezi deskami - Workb	inch	-		×			
File View Tools Units Extensions Jobs Help							
🔁 🚰 🛃 🚺 Project							
Import 🗟 🖗 Reconnect 😰 Refresh Projec	t 🕖 Update Project 📲 ACT Start Page						
Toolbox 🔻 🗜 🗙	Project Schematic			φ χ			
Analysis Systems							
🕞 Coupled Field Static							
🕞 Coupled Field Transient	▼ A						
🗹 Design Assessment	1 💽 Fluid Flow (Fluent)						
Eigenvalue Buckling	2 Geometry						
(i) Electric	3 Mesh						
🔝 Explicit Dynamics							
🔇 Fluid Flow - Blow Molding (Polyflow)	4 We setup C						
S Fluid Flow-Extrusion (Polyflow)	5 🗑 Solution 😨 🖌						
S Fluid Flow (CFX)	6 😡 Results 😨 🖌						
🔇 Fluid Flow (Fluent)	laminarni proudeni mezi deskami						
🔇 Fluid Flow (Polyflow)							
Harmonic Acoustics							

Fig. 4.6 - ANSYS Workbench after update

After running **ANSYS Fluent**, check the calculation area dimensions and boundary conditions as in the previous task (Chapter 3.4)

If all data is correct, proceed in the job setup in **ANSYS Fluent:**

- Physics Solver General / Solver-Type (Pressure-Based) Solver Command
- Command to set time-independent Physics General / Solver -Time (Steady) solution
- Physics General / Solver Gravity (no) command
- Command to set Physics General / Solver-Units SI
- Command to set the Physics Models Viscous Model Laminar laminar model

Definition of physical properties of fluid

- Command to copy water from Physics database Materials-Create / Edit Materials - Fluent Database Materials (select "Material Type" water-liquid and copy with Copy command)
- Physics Zones-Cell Zones Conditions command (define Zone (solid) and select material water-liquid). The zone must be of Fluid type

Defining boundary conditions

- Defining boundary conditions bottom wall type wall (define fixed stationary wall, default setting)
- inlet type of velocity inlet (define speed according to Table 4.3)
- outlet type of pressure outlet (define the static pressure according to Table 4.3)
- top wall type wall (define fixed fixed wall, default setting)
- symmetry1, symmetry2 ¬– type of symmetry

Initialization

Subsequently, the current field is initialized; Define initial conditions for the entire area using the "**Solution-Initialization-Method** (**Standard / Options**)" command. The values are defined based on the input boundary condition "**Compute from Inlet**", see Figure 4.7.

Outline View	Task Page	×
Filter Text	Solution Initialization	?
 Setup General Models Materials Cell Zone Conditions Dynamic Mesh Reference Values X. Reference Frames Named Expressions Solution Methods Controls Report Definitions Cell Registers Initialization Calculation Activities Run Calculation Results Surfaces Yenganics Reports 	Initialization Methods Hybrid Initialization Standard Initialization Compute from Initet Reference Frame Relative to Cell Zone Absolute Initial Values Gauge Pressure (pascal) X Velocity (m/s) 0.05 Y Velocity (m/s) 0 Z Velocity (m/s) 0 	
	Initialize Reset Patch Reset DPM Sources Reset Statistics	
	VOF Check	

Fig. 4.7 - Initialization based on input boundary condition

Before starting the calculation, set the stabilization diagrams for the calculation of the individual variables with the "**Solution / Methods**" command, see Fig. 4.8 with regard to the stability of the numerical calculation.

Task Page	
Solution Methods	?
Pressure-Velocity Coupling	
Scheme	
Coupled	_
Spatial Discretization	
Gradient	
Least Squares Cell Based	_
Pressure	
Second Order	-
Momentum	
Second Order Upwind	•
Transient Formulation	
•	
Non-Iterative Time Advancement	
Frozen Flux Formulation	
✓ Pseudo Transient	
Warped-Face Gradient Correction	
High Order Term Relaxation Options	
Structure Transient Formulation	
*	
Default	

Fig. 4.8 - Setting of stability schemes

Then you run the iterative calculation "**Solution-Run Calculation**". You must specify the number of "**Number of Iterations**" iterations. The predefined value is 0. Enter a fairly high value, eg 1000, when it is assumed that convergence will be achieved, see Figure 4.9. Convergence can be monitored both graphically and numerically.

Outline View	Task Page
Filter Text	Run Calculation
 Setup General ♥ Models ♥ Materials ♥ Cell Zone Conditions ♥ Boundary Conditions ♥ Boundary Conditions ♥ Dynamic Mesh ♥ Reference Values ♥ & Reference Frames f Named Expressions 	Check Case Update Dynamic Mesh Pseudo Transient Settings Fluid Time Scale Time Step Method Time Scale Factor Automatic 1
Solution Methods Controls Report Definitions Monitors Cell Registers Cell Registers	Parameters Number of Iterations 1000 Profile Update Interval 1
Calculation Activities Calculation Activities Calculation Results Surfaces Graphics L Plots Scene Animations Dependent	Solution Processing Statistics Data Sampling for Steady Statistics Data File Quantities Solution Advancement
Reports Parameters & Customization	Calculate

Fig. 4.9 - Setting the number of iterations and starting the calculation

The residual list is activated from the menu by the "**Results / Residuals / Residuals Monitors**" commands. The course of the residuals is shown in Fig. 4.10. Residual values (relative error) for each calculated variable (pressure - continuity, velocity in x - x - velocity direction, velocity in y - y - velocity direction and velocity in z - z - velocity direction) must be less than 0.001. When this accuracy is reached, the calculation is terminated by itself.



Fig. 4.10 - The course of residuals

The next steps will be to evaluate this calculation variant. For the sake of clarity, it is possible to create auxiliary sections with given coordinates, in which eg speed vectors are displayed. This will be followed by the creation of transverse planes at distances x = 0.1m, 0.2m, 0.3m and 0.4m and a longitudinal section through the center of the z-axis (Fig. 4.11).



Fig. 4.11 - Created planes for evaluation

To create transverse planes at distances x = 0.1m, 0.2m, 0.3m and 0.4m, use the command "**Results / Surface / Create / Iso-suface**". The creation of the transverse plane at the distance x = 0.1m is shown in Fig. 4.12. In the "**Surface of Constant**" menu, select **Mesh / X-Coordinate**. Next, enter 0.1 for Iso-Values. 0.3m and 0.4m To make a longitudinal section, select **Z-Coordinate**.

Iso-Surface	
New Surface Name x-0.1m Surface of Constant Mesh X-Coordinate Min (m) Max (m) 0 0.5 Iso-Values (m) 0.1	From Surface Filter Text Fo From From From From Filter Text Fo From <
Create	ompute Close Help

Fig. 4.12 - Creating an auxiliary plane at a distance x = 0.1 m

Then, velocity vectors, velocity profiles, and filled isolines can be evaluated to illustrate. The velocity vectors are defined in each cell of the computational domain by the command "**Results / Graphics / Vectors** /", where it is possible to define the coloring of the vectors by another variable (eg temperature). In addition, "**Scale**" allows you to reduce the size of the vector and "**Skip**" to skip a certain number of vectors to make the vectors less dense.



Fig. 4.13 - Speed vectors for Scale = 0.3 and Skip =0 ($u[m.s^{-1}]$)

Next, we draw the speed profiles with the command "**Results / Graphics / Vectors** /" in each cross-section. 4.15 Then the vectors are plotted using the "**Display**" command, see Figure 4.16.

Velocity
Color by
Velocity
Velocity Magnitude
Min (m/s) Max (m/s)
0.0008550463 0.07445044
Surfaces Filter Text
New Surface 🚽

Fig. 4.14 - Definition of velocity vectors in cross-sections

Views		×
Views	Actions	Mirror Planes [0/1] 😑 🔫
bottom front	Auto Scale	symetry
isometric left right top	Previous Save Delete Read	Define Plane) Periodic Repeats
Save Name front	Write	Define
	Apply Camera Close	Help

Fig. 4.15 - Menu for defining the front view.

)		•)	•	2	
							i ·
9.18e-04	1.14e-02	2.19e-02	3.24e-02	4.28e-02	5.33e-02	6.38e-02	7.08e-02

Fig. 4.16 - Speed vectors in individual sections $(u[m.s^{-1}])$

The evaluation shows that the parabolic velocity profile is gradually formed along the length of the computational area. In order to achieve the desired shape of the velocity profile (from the previous solution), the calculation area is short. The velocity size contours are drawn using the command "**Results / Graphics / Contours**", Fig. 4.15. It is further specified whether the velocity or velocity component or other variable in the longitudinal section is plotted. Levels define the number of isosurfaces, check **Filled** in Options to display filled isocurves, otherwise they are contour lines, the result is shown in Fig. 4.18.

Contours			3
Options	Contours of		
✓ Filled	Velocity		•]
✓ Node Values	Velocity Magnit	ude	•
Global Range	Min (m/s)	Max (m/s)	
Clip to Range	0	0.06913544	
Draw Profiles	Surfaces Filter	r Text 🗾 🔁 🔫 🔫)
Coloring	bottom_wall inlet outlet		
BandedSmooth	podelny-rez symetry top wall		
Levels Setup	x-0.1m		
20 🔤 🕺 🖉	New Surface	<i>v</i>	
	Display	mpute Close Help	

Fig. 4.17 - Menu for Creating Filled Isoplases of Velocity



Fig. 4.18 - Contour velocity magnitude in the computational area $(u[m.s^{-1}])$

Similarly, the plot of static pressure isocurves of static pressure is set in Fig. 4.19



Fig. 4.19 - Filled isocurves of static pressure in the computational area p_{stat} [Pa]

The next evaluation presents speed profiles in individual sections from the inlet input to the outlet output with a step of 0.05 m along the length of the computational area. This illustration is very illustrative if it is necessary to compare the profiles of quantities at the input, output, or in other sections of the area. Rendering is performed using the command "**Results / Plots / XY Plot / Solution XY Plot**". In the Y Axis Function menu, select **Velocity - Velocity Magnitude**, and in the **X Axis** Function menu, select **Direction Vector**. Next, in the Plot Direction menu, edit **X = 0 and Y = 1 and Z = 0**. you can plot dependence on Y, see Figure 4.20 and select the appropriate slices in the Surfaces menu.

Solution XY Plot	×
Options	Plot Direction Y Axis Function
 Node Values Position on X Axis Position on Y Axis Write to File Order Points 	X 0 Velocity Y 1 Z 0 X Axis Function Direction Vector Y
File Data 🚍 Ţ 🖡	Load File Free Data Surfaces Filter Text Top_wall x-0.1m x-0.2m x-0.3m x-0.4m
Plot Ax	res) Curves) Close Help

Fig. 4.20 - Menu for creating velocity profiles

The results show the formation of a velocity profile from a constant velocity at the **inlet** input to a parabolic velocity profile at the **outlet** exit from the area (Fig. 4.21). Another way to get data is to use the **Options-Write to File** menu, which exports data to an external text file. This file is then read and edited in Excel.



Fig. 4.21 - Formation of speed profile

Another evaluation is the course of static pressure along the length of the computational area. The static pressure is evaluated in the longitudinal section of the calculation area, see Figure 4.22.



Fig. 4.22 - Static pressure profile along the length evaluated in the pipe axis (*P*_{stat} [Pa])

5 TURBULENT FLOW - WATER FLOW BETWEEN THE PLATES

Example

Solve the flow of water between two infinitely large plates (Fig. 5.1). The physical model is given by the shape of the area, the type of flow and the hydraulic flow parameters. Define the numerical calculation in ANSYS Fluent. Use DesignModeler and **ANSYS Meshing** to create the computational area (geometry) and mesh.



Fig. 5.1 - Area diagram

Water flows into the area at a speed of 1 m.s-1 and exits into the atmosphere, where the relative pressure is 0 Pa. Area dimensions shown in Tab. 5.1. The problem is given as a 3D model and represents the flow in a rectangular gap of given length and gap thickness. The physical properties of the flowing medium are given in Tab. 5.2.

area length / [m]	0.5
height of the area S [m]	0.02
area width b [m]	0.1
Tab. 5.2 – Physical properties of water	
density of water ρ [kg.m ⁻³]	998

|--|

Boundary conditions

dynamic viscosity η [kg.(m.s)⁻¹]

0.001003

VELOCITY INLET is defined on the inlet and the static pressure condition (**PRESSURE-OUTLET**) is specified at the outlet. On the walls (**top wall, bottom wall**) there is a boundary condition of the **WALL** type, where zero flow velocity (predefined) is assumed. Boundary conditions including turbulent conditions are given in Tab. 5.3.

Inlet	Medium speed u_s [m.s ⁻¹]	1
	Turbulent intensity [%]	1
	Hydraulic diameter [m]	0.02
Outlet	Static pressure ρ [Pa]	0
	Turbulent backflow intensity [%]	1
	Hydraulic diameter [m]	0.02

Tab. 5.3 Boundary conditions

Mathematical model

The choice of mathematical model depends on Reynolds number.

The laminarity criterion is Reynolds number:

$$\operatorname{Re} = \frac{u.d}{v} = \frac{1.0.02}{1.10^{-6}} = 20000$$

The flow is therefore turbulent, but with a low Reynolds number, so **RNG k-\epsilon** turbulent mathematical model will be used.

5.1 5.1 Geometry and mesh

The geometry and mesh will be used from the previous example (laminar flow) by copying the entire panel in the **Workbench** environment. Copying is done by the command "**Duplicate**", which is invoked by the right mouse button, see Fig. 5.2.



Fig. 5.2 - Copying of panel by command "Duplicate"

Then rename the panel to, for example, "**turbulent flow between plates**" and run **ANSYS Fluent** with the "**Setup**" command to modify the task to turbulent flow between plates. The other setting remained from the laminar flow task, given only for repeat. Only the boundary conditions change.

5.2 ANSYS Fluent

Settings in ANSYS Fluent

• Solver Setting Command Setting Up Physics - General / Solver-Type (Pressure-Based)

• Time-dependent Solution Setting Up Physics - General / Solver-Time (Steady) Command

- Setting Up Physics Command General / Solver Gravity (no)
- Setting Up Physics General / Solver-Units SI command

Setting Up Physics - Models - Viscous Model - RNG k-epsilon, Scable Wall Functions

Definition of physical properties of fluid

• Setting Up Physics - Materials-Create / Edit Materials - Fluent Database Materials (select "Material Type" water-liquid and copy with Copy command)

• Setting Up Physics - Zones-Cell Zone Conditions (select Zone (surface_body) and select material water-liquid)

Defining boundary conditions

• bottom wall - type wall (Setting Up Physics-Boundaries - define fixed stationary wall, default setting)

- inlet type of velocity inlet (define speed according to Table 5.3)
- outlet type of pressure outlet (define the static pressure according to Table 5.3)

• top wall - type of wall (Setting Up Physics-Boundaries - define fixed stationary wall, default setting)

Initialization

Subsequently, the current field is initialized; Define initial conditions for the entire area using the "**Solving Initialization-Method (Standard / Options)**" command. The values are defined based on the input boundary condition. Then adjust the stabilization diagrams according to Fig. 4.8. The iteration calculation is then started. The resulting residues can be seen in Figure 5.3.



Fig. 5.3 - The course of residuals

In the next steps the evaluation of this calculation variant will follow in the transverse planes at distances x = 0.1m, 0.2m, 0.3m and 0.4m and in longitudinal section through the center of the z-axis area (Fig. 5.4).



Fig. 5.4 - Created planes for evaluation

Creating transverse planes is described in the previous chapter. Subsequently, velocity vectors, velocity profiles and filled isocurves can be evaluated. The velocity vectors are defined in each cell of the computational domain by the command "**Postprocessing / Graphics / Vectors**". We use a longitudinal section to evaluate the vectors. Adjust the settings to **Scale = 1** and **Skip = 1**..



Fig. 5.5 - Speed vectors for Scale = 1 and Skip = 1 ($u[m.s^{-1}]$)

Vectors	×			
Options	Vectors of			
Global Range	Velocity 💌			
✓ Auto Range	Color by			
Clip to Range	Velocity			
 Auto Scale Draw Mesh 	Velocity Magnitude			
Style arrow	Min (m/s) Max (m/s) 0.5864273 1.145277			
Scale Skip	Surfaces Filter Text 🗾 🔁 🗮 🗮			
	outlet			
Vector Options	podelny-rez			
Custom Vectors	top wall			
	x-0.1m x-0.2m			
	New Surface 🚽			
	Display Compute Close Help			

Fig. 5.6 - Menu for evaluation of velocity vectors in individual cross-sections

	3))	3	3	
							ě×,
5.86e-01	6.70e-01	7.54e-01	8.38e-01	9.22e-01	1.01e+00	1.09e+00	1.15e+00

obr. 5.1 – Vektory rychlosti v jednotlivých řezech ($u[m.s^{-1}]$)

The evaluation shows that the turbulent velocity profile is gradually formed along the length of the computational area. The longitudinal section of the velocity magnitude is shown in Fig. 5.8 and is drawn with the command "**Postprocessing / Graphics / Contours"**.



Fig. 5.8 - Contour velocities in the computational area ($u[m.s^{-1}]$)

Similarly, the isocurves of static pressure in Figure 5.9 and the effective viscosity of Figure 5.10 are plotted.



Fig. 5.9 - Filled static pressure lines in the computational area P_{stat} [Pa]



Fig. 5.10 - Filled isolines of effective viscosity

Further evaluation presents velocity profiles in individual cross-sections, see Fig. 5.11 by graph. Rendering is done using the command "**Postprocessing / Plots / XY Plot / Solution XY Plot**". Select **Velocity-Velocity Magnitude** in the **Y Axis** Function menu and **Direction Vector** in the **X Axis** Function menu. Next, in the Plot Direction menu, edit X = 0 and Y = 1. we want to plot the dependence on Y and in the Surfaces menu select the appropriate slices.



Fig. 5.11 - Formation of the speed profile

Another evaluation is the course of static pressure along the length of the computational area. The static pressure is evaluated along the axis of the calculation area, see Figure 5.12.


Fig. 5.12 The course of static pressure along the length evaluated in the pipe axis ([Pa])

6 SAMPLE EXAMPLE SOLUTION - CO-CURRENT EXCHANGER

Create a mathematical model of the co-current exchanger and perform a threedimensional (3D) numerical simulation. The flowing fluids in the exchanger are waterair combinations. The co-current exchanger model is shown in Fig. 6.1. Define individual areas and parameters according to the specified boundary conditions and graphically evaluate the results.



Fig. 6.1 - Co-current exchanger in 3D design.

Tab. 6.1 – Area dimensions

H1	0.5	m
D1	0.04	m
D2	0.08	m

In a given area, which is a co-current cooler, liquid-**water** flows in the center and air flows around. The walls are made of steel pipes of different diameters.

Tab. 6.2 – Physical properties of material (steel, water, air) at 300 K

Material	Steel	Water	Air	
density p	8030	998.2	1.225	[kg.m ⁻³]
specific heat capacity c_{ρ}	502.48	4182	1006.43	[J.kg ⁻¹ K ⁻¹]
thermal conductivity λ	16.27	0.6	0.0242	[W.m ⁻¹ K ⁻¹]
viscosity η		0.001003	0.000017894	[kg.m ⁻¹ s ⁻¹]

Tab. 6.3 – Boundary conditions

	Inlet air	Inlet water	Outlet air	Outlet water	Wall inner	Wall outer	
temperature T	300	363.15				300	[K]
velocity u	3	0.3					[m.s ⁻¹]
pressure <i>p</i>			0	0			[Pa]
intensity of turbulence <i>I</i>	1	1	1	1			[%]
hydraulic diameter <i>d</i> _h	0.02	0.04	0.02	0.04			[m]

Next, consider the thickness of the inner wall and the outer wall 003m. Wall material consider steel.

6.1 6.1 Mathematical model and theoretical-empirical estimation of the problem

In this task turbulent flow occurs, so the mathematical model of RNG is used k- ϵ . The criterion of turbulence is the Reynolds number.

Re for water flow:

$$Re_{voda} = \frac{v \cdot d_h}{v} = \frac{0.3 \cdot 0.04}{1.01e - 06} = 12000$$
(6.1)

Re for air flow:

$$Re_{vzduch} = \frac{v \cdot d_h}{v} = \frac{3 \cdot 0.02}{1.46e - 05} = 4323$$
(6.2)

The calculation of the Nusselt number and the heat transfer coefficient is based on empirical relations, which are described in detail in the literature [2]. In the next step, only the analytical calculation is performed, which will be compared with the numerical calculation. From the given parameters it is possible to calculate the above parameters of flow and heat transfer (Reynolds number is calculated from the maximum speed). The estimation of Nusselt's number is problematic and is only indicative. This estimate is followed by the calculation of the wall heat transfer coefficient determined from the Nusselt number $\stackrel{\sim}{\alpha} = \frac{\text{Nu}.\lambda}{d}$ [2].

$$\Pr = \frac{\rho \cdot c_p \cdot v}{\lambda} = \frac{998.2 \cdot 4182 \cdot 1.01e - 6}{0.6} = 6.99$$

$$Nu = 0.023 \cdot Re^{0.8} \cdot Pr^{0.3}$$
(6.3)

$$Nu = 0.023 \cdot 12000^{0,8} \cdot 6.99^{0.3} = 75.5 \tag{6.4}$$

Then the heat transfer coefficient is

$$\alpha = \frac{Nu}{d_h} \cdot \lambda = \frac{75.5}{0.04} \cdot 0.6 = 1132.6 \ W. \ m^{-2}. \ K^{-1}$$
(6.5)

Calculation of the Nusselt number for the area of air flow around a pipe:

$$Pr = \frac{\rho \cdot c_p \cdot \nu}{\lambda} = \frac{1.225 \cdot 1006.43 \cdot 1.46e - 5}{0.0242} = 0.707 \tag{6.6}$$

$$Nu = 0.023 \cdot Re^{0.8} \cdot Pr^{0.4}$$

$$Nu = 0.023 \cdot 4323^{0.8} \cdot 0.707^{0.4} = 16.79$$
(6.7)

Then the heat transfer coefficient is

$$\alpha = \frac{Nu}{d_h} \cdot \lambda = \frac{16.79}{0.02} \cdot 0.0242 = 20.3 \ W. \ m^{-2}. \ K^{-1}$$
(6.8)

6.2 6.2 Geometry creation

Run the **ANSYS 2019 R3** program as per chap. 3.1. Name the newly created panel, for example, *heat exchanger.* Then save the entire project under any name and run the **DesignModeler** geometry program.

To create the geometry use the detailed instructions in chap. 3.2, because the resulting co-current exchanger model is a 3D model similar to a 3D rod model. The co-flow exchanger model represents two areas (interior water, interior air). Fig. 6.1. These are therefore two cylinders that we must subtract from each other. You create the regions identically using **Create / Primitives / Cylinder** as in the example of the conduction heat in a rod. The final appearance of the interior water area created by the Cylinder including the dimensions is shown in Figure 6.2.



Fig. 6.2 - Creating an interior water area ("Cylinder")

The final appearance of the **interior air** created by the cylinder (**cylinder**) including the dimensions is shown in Fig. 6.3.



Fig. 6.3 - Creating an interior air area ("Cylinder")

In the case of two areas to be separate volumes, you must define the **Add Frozen** item in the **Operation tools**. This will not merge faces.

Now you need to subtract the cylinders from each other by Boolean operations using the "**Create / Boolean / Operation-Subtract**" command. Select air area as target points and water area as tool points. Select Preserve Tool Body to preserve the water area. Clicking **Generate** creates two separate volumes for the water area and the air area.

The last operation is to merge volumes into one unit, ie. New part. By merging the volumes into one unit, the continuity of the computer network between individual areas will be preserved. You can get the command by selecting both volumes in the **2 Parts**, **2 Bodies** tab and right-clicking on the "**Form New Part**" menu, see Figure 6.4.



Fig. 6.4 - Merging volumes into one unit ("Form New Part") The final form of the "**Form New Part**" command is shown in Figure 6.5.



Fig. 6.5 - The final form of the "Form New Part"

In the next phase, name the boundary conditions as described in Figure 6.1 (**inlet air**, **inlet water**, **outlet air**, **outlet water**, **wall inner**, **wall outer**). The naming of the boundary conditions is done using the "Named Selection" command with Face mode (3.2). The resulting marking and naming of all boundary conditions can be seen in Figure 6.6. In addition to the boundary conditions on the walls, the model contains two areas of interiors (volumes) that need to be defined (**interior_air** and **interior_water**).



Fig. 6.6 - Marking of boundary conditions

This completes the coil flow model in **DesignModeller**.

6.3 6.3 Creation of mesh

Now you can switch to the mesh in **ANSYS Meshing**. The procedure for starting the program is described in chap. 3.3. Use the same tools as shown in Chap. 3.3. Create a mesh with interior water and interior air boundary layers towards the wall inner and wall outer.

To create a computational network in this form, you will use network compaction (create only at the front of both cylinders) and sweep. These are therefore the same operations as used to create the computer network in the example of heat conduction in a rod. In the **Details of Mesh** panel, redefine the element size for **Element Size** to 4 mm, **Max Size** to 10 mm.

Details of "Mesh"	→ 4 □ ×	
- Display	-	
Display Style	Use Geometry Setting	
 Defaults 		
Physics Preference	CFD	
Solver Preference	Fluent	
Element Order	Linear	
Element Size	4, mm	
Export Format	Standard	
Export Preview Surface Mesh	No	
Sizing		
Use Adaptive Sizing	No	
Growth Rate	Default (1,2)	
Max Size	10, mm	
Mesh Defeaturing	Yes	
Defeature Size	Default (2,e-002 mm)	
Capture Curvature	Yes	
Curvature Min Size	Default (4,e-002 mm)	
Curvature Normal Angle	Default (18,°)	
Capture Proximity	No	
Bounding Box Diagonal	512,64 mm	
Average Surface Area	37287 mm ²	
Minimum Edge Length	125,66 mm	
- Quality		
Check Mesh Quality	Yes, Errors	

Fig. 6.7 - Defining element size

Defining Inflation Parameters

Then define the compaction mesh parameters. The number of densified layers (cells), the growth factor characterizing the gradual reduction of the cell size towards the boundary, the reduction ratio of the last cell of the densified area. Define two areas of densification into each area (interior water, interior air) towards the wall inner.

Number of boundary layer layers (cells)

- Growth factor 1,2
- Factor characterizing gradual cell size reduction 0.272

Inflation parameters towards wall inner for interior water and interior air are shown in Figure 6.8 and Figure 6.9.

D	etails of "Inflation 2" - Inflati	ion	
-	Scope		
	Scoping Method	Geometry Selection	
	Geometry	1 Face	
-	Definition	·	
	Suppressed	No	
	Boundary Scoping Method	Geometry Selection	
	Boundary	1 Edge	
	Inflation Option	Smooth Transition	
	Transition Ratio	Default (0,272)	, in the second s
	Maximum Layers	6	+
	Growth Rate	1,2	
	Inflation Algorithm	Pre	0.070 (m) Z

Fig. 6.8 - Compression parameters for the interior water area

Details of "Inflation" - Inflatio	n ::::::::::::::::::::::::::::::::::::	
- Scope		
Scoping Method	Geometry Selection	
Geometry	1 Face	
- Definition		
Suppressed	No	
Boundary Scoping Method	Geometry Selection	
Boundary	2 Edges	
Inflation Option	Smooth Transition	
Transition Ratio	Default (0,272)	
Maximum Layers	6	
Growth Rate	1,2	-
Inflation Algorithm	Pre	0.070(m)

Fig. 6.9 - Compression parameters for the interior air area

Now insert the "**Sweep**" method (for settings, see chapter 3.3) and thereby pull the surface mesh into volume.

To generate a computer network, use the **Generate Mesh** command. The resulting form of the mesh is shown in Figure 6.10.



Fig. 6.10 - The resulting form of the mesh

6.4 ANSYS FLUENT

You run the ANSYS FLUENT 2019 R3 in a similar way as in the example of heat conduction in a rod.

After successfully loading the computer network into **ANSYS Fluent 2019 R3**, check:

- units of mesh size with the command "Domain / Mesh / Scale"
- number of network cells with the command "Domain / Mesh / Info / Size"
- the existence of negative volumes in the mesh using the "Domain / Mesh / Check" command
- Mesh by displaying all boundaries (boundary conditions) and all areas with the command "Domain / Mesh / Display"

When checking the mesh with the "**Domain / Mesh / Display**" command, all boundary conditions are named as defined in **ANSYSMeshing**. Except for one newly created boundary condition wall_inner-shadow. Which represents the same boundary condition as wall_inner. A new boundary condition **wall_inner-shadow** was created (Fig. 6.11), which together with the **wall_inner** condition defines a so-called two-layer wall, where one is part of the interior water area and the other is part of the interior air area. This type of boundary condition offers the definition of additional options at the transition between the two regions.

📧 Mesh Displa	у	×		
Options Nodes	Edge Type All	Surfaces Filter Text 🔂 🖶 🗮		
 Edges Faces Partitions Overset 	 Feature Outline 	inlet_air inlet_water outlet_air outlet_vater wall_inner		
Shrink Factor F 0 Outline	Feature Angle 20 Interior	wall_inner-shadow wall_outer		
Adjacency New Surface 🚽				
Display Colors Close Help				

Fig. 6.11 - Checking boundary conditions

Use the following mathematical model settings:

- Time-stable flow
- Turbulent k-ε RNG flow model for water and air
- Without considering gravity acceleration
- Consider heat transfer (energy equation)
- Define constant physical properties of water and air (copy materials from Fluent library)

Within the "**General**" command, define "**Solver**" of the "**Pressure-Based**" type. Steady flow. Do not consider gravity acceleration. The setting of the "**General**" command is shown in Fig. 6.12.

Task Page	X
General	(?)
Mesh	
Scale Cl	heck Report Quality
Display Ur	nits
Solver	
Туре	Velocity Formulation
 Pressure-Based Density-Based 	Absolute Relative
Time	
 Steady 	
○ Transient	
Gravity	

Fig. 6.12 - Command "General"

Another command is **Models** ("**Physics / Define / Models**"), where the physical nature of the task is defined, ie **Energy** flow and the turbulent k- ϵ RNG model of Viscous flow together with the **Scable Wall Functions**, see Figure 6.13.

Viscous Model		x
Model	Model Constants	
◯ Inviscid	Cmu	
🔿 Laminar	0.0845	
O Spalart-Allmaras (1 eqn)	C1-Epsilon	
k-epsilon (2 eqn)	1.42	
🔿 k-omega (2 eqn)	C2-Epsilon	
O Transition k-kl-omega (3 eqn)	1.68	
O Transition SST (4 eqn)	Wall Prandtl Number	
O Reynolds Stress (7 eqn)	0.85	
 Scale-Adaptive Simulation (SAS) 		
O Detached Eddy Simulation (DES)		
C Large Eddy Simulation (LES)		
k-epsilon Model		
🔾 Standard		
RNG		
Realizable		
RNG Options	User-Defined Functions	
Differential Viscosity Model	Turbulent Viscosity	
Swirl Dominated Flow	none	•
Near-Wall Treatment	Prandtl Numbers	
O Standard Wall Functions	Wall Prandtl Number	
Scalable Wall Functions	none	•
O Non-Equilibrium Wall Functions		
O Enhanced Wall Treatment		
O Menter-Lechner		
O User-Defined Wall Functions		
Options		
✓ Viscous Heating		
Curvature Correction		
Production Kato-Launder		
Production Limiter		
OK Cancel Help		

Fig. 6.13 - Setting the mathematical model of the solved problem

To define the material, use the Materials command ("**Physics / Materials / Create / Edit Materials**"). Analogous to the heat conduction in a rod, select the materials: **water, air, steel** to be copied from the ANSYS Fluent database. These are fluid and solid materials, and define a constant physical property for all materials. The resulting form of material supply is shown in Figure 6.14.

Task Page	Create/Edit Materials		×	
Marked all	Name	Material Type	Order Materials by	
Materials	steel	solid	 Name 	
Materials	Chemical Formula	Fluent Solid Materials	Chemical Formula	
Fluid		steel 🔹	Fluent Database	
water-liquid		Mixture		
air		none	User-Defined Database	
steel	Properties			
aluminum	Density (kg/m3)	constant	▼ Edit	
		8030		
	Cp (Specific Heat) (j/kg-kj	constant	▼ Edit	
		502.48		
	Thermal Conductivity (w/m-k)	constant	▼ Edit	
		16.27		
	Change/Create Delete Close Help			

Fig. 6.14 - Required materials for the mathematical model

To define the flowing fluid into the area, use the "**Physics / Cell Zone Conditions**" command. In this case we have two areas (**interior water, interior air**). Define **water** in the **interior water** area, and define **air** in the **interior air** area, as shown in Figure 6.15.

Task Page	Pluid		Х
Cell Zone Conditions	Zone Name Interior_water		
Zone Filter Text interior_air interior_water	Material Name water-liquid Edit Frame Motion 3D Fan Zone Source Terms Mesh Motion Laminar Zone Fixed Values Porous Zone Fixed Values		
	Reference Frame Mesh Motion Porous Zone 3D Fan Zone Embedded LES Reaction Source Terms Fixed V	alues	Multiphase
	Rotation-Axis Origin Rotation-Axis Direction		
	x (m) 0 • X 0	•	
	Y (m) 0 Y 0	•	
	Z (m) 0 Z 1	•	
Cell Zone Conditions Zone Filter Text interior_air interior_water	Zone Name interior_air Material Name air Edit Frame Motion 3D Fan Zone Source Terms Mesh Motion Laminar Zone Fixed Values Porous Zone Reference Frame Mash Malan Boreus Zone State State Source Terms	/shuar	Mi iHidhare
		uluca I	narapinase
	Kotation-Axis Direction	•	1
	Y (m) 0 Y 0	-	
	Z (m) 0 ~ Z 1	-	
	OK Cancel Help		

Fig. 6.15 - Defining flow media to given areas

Use the "**Physics / Boundary Conditions**" command to define boundary conditions. The conditions may be of different types depending on the characteristics of the physical model. The list of conditions is apparent from Tab. 6.4.

Okrajová podmínka	Typ okrajové podmínky
inlet water	VELOCITY INLET
inlet air	VELOCITY INLET
outlet water	PRESSURE OUTLET
outlet air	PRESSURE OUTLET
wall inner	WALL
wall outer	WALL
interior water	INTERIOR
interior air	INTERIOR

Tab. 6.4 – Types of individual boundary conditions

Parameters on individual boundary conditions correspond to the input according to Tab. 6.3. The setting of boundary conditions is shown in the following figures.

Velocity	Inlet						×	🔹 Velocity I	nlet	
Zone Name								Zone Name		
inlet_water								inlet_water		
Momentum	Thermal	Radiation	Species	DPM	Multiphase	Potential	UDS	Momentum	Thermal	Radiation
Veloci	y Specificat	ion Method	Magnitude,	Normal to	Boundary		•	Temperatur	e (k) 363.1	15
	Refere	nce Frame	Absolute				•			
	Velocity	Magnitude	(m/s) 0.3				•			
Supersonio	/Initial Gau	ge Pressure	(pascal) 0				-			
	Turbulence	e								
	Specification	on Method	Intensity an	d Hydraulio	c Diameter		•			
	Turbuler	nt Intensity ((%) 1				•			
	Hydraulio	c Diameter ((m) 0.04				•			
			OK Can	cel Hel	p					ок

Fig. 6.16 - Parameters of boundary condition inlet water

📧 Velocity	Inlet						×	🗾 Velocity l	nlet		
Zone Name								Zone Name			
inlet_air								inlet_air			
Momentum	Thermal	Radiation	Species	DPM	Multiphase	Potential	UDS	Momentum	Thermal	Radiation	Spe
Veloci	ty Specificat	ion Method	Magnitude,	Normal to	o Boundary		•	Temperatur	e (k) 300		
	Refere	nce Frame	Absolute				•				
	Velocity	Magnitude	(m/s) 3				•				
Supersonio	/Initial Gau	je Pressure	(pascal) 0				-				
	Turbulence	2									
	Specificatio	on Method	Intensity an	d Hydrauli	c Diameter		•				
	Turbuler	t Intensity	(%) 1				•				
	Hydraulio	: Diameter	(m) 0.02				•				
			OK Can	cel Hel	p						ок

Fig. 6.17 - Parameters of boundary condition inlet air

Pressure Outlet					×	Pressure O	utlet		_	
Zone Name						Zone Name				
outlet_water						outlet_water				
Momentum Thermal Radiation	Species	DPM	Multiphase	Potential	UDS	Momentum	Thermal	Radiation	Species	DP
Backflow Reference Fram	ne Absolute				•	Backflow To	tal Tempera	ature (k) 363		
Gauge Pressu	re (pascal)	0			•					
Pressure Profile Multipli	er 1				•					
Backflow Direction Specification Metho	od Normal t	o Boundary			•					
Backflow Pressure Specification	on Total Pre	essure			•					
Prevent Reverse Flow										
Radial Equilibrium Pressure Distrib	oution									
Average Pressure Specification										
Target Mass Flow Rate										
Turbulence										
Specification Metho	d Intensity a	and Hydrau	lic Diameter		•					
Backflow Turbulent Intensit	y (%) 1				-					
Backflow Hydraulic Diamete	er (m) 0.04				•					
	or 6		1							
	Canc	er Help							Can	

Fig. 6.18 - Parameters of boundary condition outlet water

Pressure Outlet	× Pressure Outlet
Zone Name	Zone Name
outlet_air	outlet_air
Momentum Thermal Radiation Species DPM Multiphase Potential UD	S Momentum Thermal Radiation Species F
Backflow Reference Frame Absolute	Backflow Total Temperature (k) 300
Gauge Pressure (pascal) 0) -
Pressure Profile Multiplier 1	
Backflow Direction Specification Method Normal to Boundary	•
Backflow Pressure Specification Total Pressure	•
Prevent Reverse Flow	
Radial Equilibrium Pressure Distribution	
Average Pressure Specification	
Target Mass Flow Rate	
Turbulence	
Specification Method Intensity and Hydraulic Diameter	
Backflow Turbulent Intensity (%)	•
Backflow Hydraulic Diameter (m) 0.02	▼
OK Cancel Help	OK Cancel

Fig. 6.19 - Parameters of boundary condition outlet air

Task Page	💽 Wall									×
Boundary Conditions	Zone Name wall_inner Adjacent Cell	Zone								
Zone Filter Text	interior wate	r								
inlet_air inlet_water	Shadow Face wall_inner-sh	Zone adow								
interior-interior_air interior-interior_water outlet_air	Momentum	Thermal	Radiation	Species	DPM	Multiphase	UDS	Wall Film	Potential	Structure
outlet_water	O Heat Fl	лх		Wa	ll Thickness	(m) 0.003				
wall_inner wall_outer	Temper Coupled Material Nar steel	ne	• Edit	Heat Gene	ration Rate	(w/m3) 0 Shell	Conductio	n 1 Layer) Edit
				(OK Can	cel Help				

Fig. 6.20 - Parameters of boundary condition wall inner

Task Page	2 Wall	×
Devendence Constitutions	Zone Name	
Boundary Conditions	wall_inner-shadow	
	Adjacent Cell Zone	
Zone Filter Text	interior_air	
inlet_air inlet water	Shadow Face Zone wall_inner	
interior-interior_air	Momentum Thermal Radiation Species DPM Multiphase UDS W	all Film Potential Structure
outlet_air outlet_water wall_inner wall_inner-shadow wall_outer	Thermal Conditions Heat Flux Wall Thickness (m) 0.003 Temperature Heat Generation Rate (w/m3) 0 © Coupled Shell Conduction Material Name Edit Steel Edit	1 Layer Edit

Fig. 6.21 - Parameters of boundary condition wall inner-shadow

E Wall							X
Zone Name wall_outer Adjacent Cell Zone interior_air							
Momentum Thermal	Radiation Species	DPM	Multiphase	UDS	Wall Film	Potential	Structure
Thermal Conditions Heat Flux Temperature Convection Radiation Mixed via System Coupli via Mapped Interfe Material Name steel	V Heat Ge ace Edit	Temperature /all Thicknes: neration Rate	e (k) 300 s (m) 0.003 e (w/m3) 0 Shell	Conductio	on 1 Layer	r	Edit
		ОК Сал	Help				

Fig. 6.22 - Parameters of boundary condition wall outer

The calculation area is then initialized ("Solving-Initialization-Method (**Standard / Options**)"). defining initial conditions for the whole area. In the first step, define the initial conditions (**zero values, minimum temperature**) based on the parameters in the **inlet_air** boundary condition.

lution Initialization	?
Initialization Methods	
O Hybrid Initialization	
Standard Initialization	
Compute from	
inlet_air 🔹	
Reference Frame	
Relative to Cell ZoneAbsolute	
Initial Values	
Gauge Pressure (pascal)	-
0	
X Velocity (m/s)	
0	
Y Velocity (m/s)	
0	
Z Velocity (m/s)	
3	
Turbulent Kinetic Energy (m2/s2)	
0.00135	
Turbulent Dissipation Rate (m2/s3)	
0.00555284	
Temperature (k)	-
Initialize Reset Patch	

Fig. 6.23 - Initialization of the computation area ("Solution Initialization")

In the second step, define the initial temperature value T = 363K for the entire interior_water using the **Patch** command in the same window (Fig. 6.24) to speed up the numerical calculation.

Reference Frame			
Relative to Cell Zone Absolute	Patch		×
Initial Values Gauge Pressure (pascal) 0 X Velocity (m/s) 0 Y Velocity (m/s) 0 Z Velocity (m/s) 3 Turbulent Kinetic Energy (m2/s2) 0.00135	Reference Frame Relative to Cell Zone Absolute Variable Pressure X Velocity Y Velocity Z Velocity Temperature Turbulent Kinetic Energy Turbulent Dissipation Rate Image: State Sta	Value (k) 363 Use Field Function Field Function	Zones to Patch Filter Text
Turbulent Dissipation Rate (m2/s3) 0.00555284 Temperature (k) Initialize Reset Patch Reset DPM Sources Reset Statistics	Console Setting inlet_wate Setting inlet_air Setting wall outer	Patch Close I r (mixture) Done. (mixture) Done. (mixture) Done.	Help

Fig. 6.24 - Initialization of water flow area by command Patch

Then run the numerical calculation using the "Run Calculation" command. The first check of the calculation is the monitoring of residuals (relative errors). Once the

residual values are below 0.001 for all variables and 0.000001 for temperature, it is guaranteed that the calculation has converged numerically. How real the results are, ie whether the result is not deformed by random errors in the selection of materials or boundary conditions, is a question of evaluating all calculated quantities. The course of residuals is shown in Fig. 6.25 - The course of residuals.



Fig. 6.25 - The course of residuals

For the evaluation it is necessary to create a longitudinal section of the area using the command "**Results / Surface / Create / Iso-Surface**". In this section, subsequently evaluate the graphic outputs. The setting of the longitudinal section through the water flow area is shown in Fig. 6.26. Similarly, we create a section of the **interior-air** area, with the difference that in the From Zones item select **interior_air**.

📧 Iso-Surface				×
New Surface Nam rez-podelny-wate	er		From Surface Filter Text	
Surface of Consta Mesh	nt	•	rez-podelny rez-podelny-air rez-podelny-water	
Y-Coordinate Min (m) -0.03995027	Max (m)	•	wall_inner wall_inner-shadow wall_outer	
Iso-Values (m)			From Zones Filter Text interior_air interior_water	
		Create	ompute Close Help	

Fig. 6.26 - Creating a longitudinal section in the interior water area

The resulting section is shown in Figure 6.27.



Fig. 6.27- Longitudinal section through the center of the computing area

To evaluate the velocity vectors defined in each cell of the computation area, use the "**Results / Graphics / Vectors** /" command to adjust the value of the "**Scale**" parameter. Define a new value for the "Scale = 0.5" parameter, see Figure 6.28.



Fig. 6.28 - Velocity vectors ($u[m.s^{-1}]$)

The course of static pressure in the longitudinal section in the solved areas (interior water, interior air) can be displayed using the filled out "Results / Graphics / Contours" contours, see Fig. 6.29.



Fig. 6.29 - Static pressure contours (Pa)

The static pressure curve can also be displayed using a 2D graph with the command "**Results / Plots / XY Plot / Solution XY Plot**" in individual areas (interior water, interior air). The subsequent plot of the pressure curve with the command "**Results / Plots / XY Plot / Solution XY Plot**" in the individual areas is shown in Figure 6.30. The results can be distorted into one graph.



Fig. 6.30 - Progression of static pressure along length in area interior water and interior air

Further evaluations are the longitudinal section contour velocities using the filled in "**Results / Graphics / Contours**" contours, see Figure 6.31.

	Contours Contour Name contour-1	×	ANSYS 2019 R ACADEMI
Contour-1 Velocity Magnitude 3.356+00 2.268+00 2.248+00 1.876+00 1.876+00 1.876+00 1.876+00 1.876+00 1.000+00 (ms)	Options	Contours of Velocity Velocity Magnitude Velocity Magnitude Vielocity Magnitude Min (m/s) Max (m/s) 0 3.345917 Surfaces Filter Text Fo Odelny- Filter Text Fo Odelny- Filter Text Filte	
	s	ave/Display Compute Close Help	Å,

Fig. 6.31 - Velocity contours [m/s]

The graph of effective viscosity using filled out "*Results / Graphics / Contours*" contours is shown in Fig. 6.32.



Fig. 6.32 - Effective viscosity [kg.m⁻¹.s⁻¹]

The evaluation of the temperature field using the filled in "**Results / Graphics / Contours**" contours is shown in Figure 6.33.

	Contours Contour Name contour-1	×	AN 2 ACA
bntour-1 tatio Temperature 3 03e402 3 .57e402 3 .51e402 3 .38e402 3 .38e402 3 .32e402 3 .32e402 3 .32e402 3 .32e402 3 .32e402 3 .32e402 3 .32e402 3 .32e402 3 .32e402 3 .30e402 3 .00e402 3 .00e402	Options Filled Filled Onde Values Coloru Lines Coloring Banded Coloring Colori	Contours of Temperature Static Temperature Min (k) Max (k) 300 363.15 Surfaces Filter Text inlet_air outlet_air outlet_air outlet_air outlet_air rez-podelny-air rez-podelny-air rez-podelny-water wall_inner New Surface _	
7		Save/Display Compute Close Help	

Fig. 6.33 - Temperature field [K]

The heat flow through the **wall inner** (**wall inner-shadow**) can be evaluated by the command "**Results / Plots / XY Plot / Solution XY Plot**", see Figure 6.35. The interface wall is divided into two walls (**wall inner** and **wall inner-shadow**), one acting as an interface for water and the other as an interface for air. Their exact designation is related to the subsequent evaluation of the heat transfer coefficient and the Nusselt number. To determine exactly which wall is part of a given flow area, use the "**Physics / Zones / Boundary Conditions**" command. Subsequent editing eg wall inner it is stated that the wall is adjacent to the surrounding area (**Adjacent Cell Zone**) - **interior** _**water**, ie. with water, see Fig. 6.34. In the case of **wall_inner-shadow** it will be the opposite (the wall is adjacent to the air area)).

Task Page	Vall X					
	Zone Name					
Boundary Conditions	wall_inner					
	Adjacent Cell Zone					
Zone Filter Text	interior_water					
inlet air	Shadow Face Zone					
inlet_water	wall_inner-shadow					
interior-interior_air interior-interior_water	Momentum Thermal Radiation Species DPM Multiphase UDS Wall Film Potential Structure					
outlet_air	Wall Motion Motion					
outlet_water	Stationary Wall Relative to Adjacent Cell Zone					
wall_inner wall_inner-shadow	O Moving Wall					
wall_outer						
	O Specified Snear					
	Wall Roughness					
	Roughness Height (m) 0					
	Roughness Constant 0 5					
	OK Cancel Help					

Fig. 6.34 - Identification of the wall inner adjacent to the surrounding water

The evaluation of the **Total Surface Heat Flux** through the wall inside and inside wall with the "**Results / Plots / XY Plot / Solution XY Plot**" command is shown in Figure 6.35..



Fig. 6.35 - Heat flow (W / m^2) through the wall inner (wall inner-shadow)

By analogy, the heat flow on the walls can be evaluated using the filled out "**Results / Graphics / Contours**" (**Chyba! Nenalezen zdroj odkazů.**).



Fig. 6.36 - Heat flow (W / m^2) through the **wall inner**)

[№] Node Values [№] Total Surface Heat Flux [№] Information Global Range Min (w/m2) Max (w/m2)	
Global Range ✓ Auto Range Min (w/m2) Max (w/m2)	
V Auto Ralige	
Clip to Range 208.0936 1046.646	
1.00e+03 I Draw Mesh	
9.63e+02 outlet_water	
9.21e+02 Coloring rez-podelny-air	
8.79e+02 Banded rez-podelny-water wall inner	
8.37e+02 Smooth wall_inner-shadow	
7.95e+02 Levels Setup wall_outer	
7.53e+02 20 J 1 J	
7.11e+02	\mathbf{i}
6.69e+02 Display Compute Close Help	
6.27e+02	
5.85e+02	
5.44e+02	
5.02e+02	
4.60e+02	
4.18e+02	
3.76e+02	
3.34e+02	
2.92e+02	
2.50e+02	
2.08e+02	

Fig. 6.37 - Heat flow (W / m^2) through the wall **inner-shadow**)

Furthermore, the evaluation focuses on the **heat transfer coefficients** α and the **Nusselt number Nu** into water and air, while it is necessary to define reference values

Evaluation for water

First, define the reference values according to the inlet water with the command "**Results / Reporst / Reference Values**". Under "**Compute from**", select **inlet water**. In the "**Reference Values**" menu, specify the "**Temperature**" and the "**Lenght**" - (Tref = 363.15 K, dh = 0.04 m), see Figure 6.38.

Refe	(?)	
Com	pute from	
inle	t_water	-
	Reference Values	
	Area (m2)	1
	Density (kg/m3)	998.2
	Enthalpy (j/kg)	0
	Length (m)	0.04
	Pressure (pascal)	0
	Temperature (k)	363.15
	Velocity (m/s)	0.3
	Viscosity (kg/m-s)	0.001003
	Ratio of Specific Heats	1.4
Refe	rence Zone	
inte	rior_water	-

Fig. 6.38 - Reference values for evaluation into the water for the wall wall inner

Evaluation of the **surface heat transfer coefficient** α from the water side to the wall inner is carried out with the command "**Results / Plots / XY Plot / Solution XY Plot**", see Figure 6.39.



Fig. 6.39 - Heat transfer coefficient through the interface wall (wall inner) [*W.m*⁻².*K*⁻¹] Analogously, the heat transfer coefficients on the wall can be evaluated using the filled out "**Results / Graphics / Contours**" contours" (*obr. 6.1*).



obr. 6.1 – Heat transfer coefficient through interface wall (**wall inner**) [W.m⁻².K⁻¹] by contours

Subsequently, the **Nusselt number** on the **wall inner** can be evaluated. First check the reference values with the command "**Results / Reporst / Reference Values**" (Temperature -Tref = 363.15 K and Lenght - dh = 0.04 m. Then draw the Nusselt number with the command "**Results / Plots / XY Plot**" (**Chyba! Nenalezen zdroj o dkazů**.).



Fig. 6.41 - Nusselt number evaluated on the interface wall (wall inner)

The Nusselt number can be evaluated using the filled out "**Results / Graphics / Contours**" contours" (**Chyba! Nenalezen zdroj odkazů**.).

Fig. 6.42 - Nusselt number evaluated on the interface wall (wall inner)

Evaluation for air

Next we evaluate the **surface heat transfer coefficient** a and the wall **inner-shadow Nusselt number Nu** by the command "**Results / Plots / XY Plot / Solution XY Plot**". First, define the reference values according to the inlet air with the command "**Results / Reporst / Reference Values**". Under "**Compute from**", select **inlet air**. In the "**Reference Values**" menu, specify the "**Temperature**" and the "**Length**" (Tref = 300 K, dh = 0.02 m), as shown in Figure 6.43.

Reference Values			
Compute f	from		
inlet_air		•	
	Reference Values		
	Area (m2)	1	
	Density (kg/m3)	1.225	
	Enthalpy (j/kg)	0	
	Length (m)	0.02	
	Pressure (pascal)	0	
	Temperature (k)	300	
	Velocity (m/s)	3	
	Viscosity (kg/m-s)	1.7894e-05	
	Ratio of Specific Heats	1.4	
Reference	Zone		
interior_a	air	•	

Fig. 6.43 - Reference values for the wall wall inner-shadow



Fig. 6.44 - Coefficient of heat transfer to air for the interface (**wall inner-shadow**) $[W.m^{-2}.K^{-1}]$

The evaluation of the heat transfer coefficient using the filled in "**Results / Graphics / Contours**" contours is shown in Figure 6.45.



Fig. 6.45 - Heat transfer coefficient for air for interface (**wall inner-shadow**) [W.m⁻².K⁻¹]

Similarly, we evaluate the **Nusselt number** on the wall **inner-shadow** interface. Check the reference values with the command "**Results / Reporst / Reference Values**" (Temperature -Tref = 300 K and Lenght - dh = 0.02 m). Then draw the **Nusselt number** with the command "**Results / Plots / XY Plot / Solution XY Plot**", see Figure 6.46.



Fig. 6.46 - Nusselt number evaluated on the interface wall (**wall innershadow**)

By analogy, the Nusselt number can be evaluated using the filled out "**Results / Graphics / Contours" contours**" (**Chyba! Nenalezen zdroj odkazů**.).

Fig. 6.47 - Nusselt number evaluated on the interface wall (wall inner-shadow)

Evaluation of average values

To evaluate the average **Nusselt value** on the wall inner for the water flow area, use the "**Results / Report / Surface Integral**" command. First, define the reference values as shown in Figure 6.38. Select "**Area-Weighted Average**" in the "**Report Type**" menu. Next, in the "**Field Variable**" menu, select "**Wall Fluxes-Surface Nusselt Number**" and in "**Surface**" select "**wall inner**" (Figure 6.48). Write the resulting value in Tab. 6.5.

Surface Integrals			
Report Type	Field Variable		
Area-Weighted Average	Wall Fluxes		
Custom Vectors	Surface Nusselt Number		
Custom Vectors	Surfaces Filter Text 50 7 7		
Save Output Parameter	inlet_water outlet_air outlet_water rez-podelny rez-podelny-air rez-podelny-water wall_inner wall_inner-shadow wall_outer		
Highlight Surfaces Area-Weighted Average 82.17902			
Compute	Write Close Help		

Fig. 6.48 - Evaluation of the average value of the Nusselt number on the **wall** inner for the water flow area

Follow the same procedure for evaluating the average value of the **Nusselt number** on the **wall inner-shadow** for the air flow area, see Figure 6.49. Define the reference values as shown in Figure 6.43.

Report Type	Field Variable
Area-Weighted Average	Wall Fluxes
Custom Vectors Vectors of	Surface Heat Transfer Coef.
Custom Vectors Save Output Parameter	Surfaces Filter Text To Text Text Text Text Text Text Text Text
	Highlight Surfaces Area-Weighted Average (w/m2-k)

Fig. 6.49 - Evaluation of the average value of the **Nusselt number** on the **wall inner-shadow** for the area of **air** flow

In the same way, evaluate the mean value of the **surface heat transfer coefficient** na on the **wall inner** for the **water** flow area using the "**Postprocessing / Report / Surface Integral**" command (Figure 6.50).

Surface Integrals	×
Report Type	Field Variable
Area-Weighted Average	Wall Fluxes
Custom Vectors Vectors of	Surface Heat Transfer Coef.
Custom Vectors Save Output Parameter	Surfaces Filter Text To
Compute	Highlight Surfaces Area-Weighted Average (w/m2-k) 1232.685 Write Close Help

Fig. 6.50 - Evaluation of the mean value of the **heat transfer coefficient** α on the **wall inner** for the **water** flow area

Fig. 6.50 - Evaluation of the average value of the coefficient The same procedure applies to the evaluation of the average value of the **heat transfer coefficient** α on the **wall inner-shadow** for the **air** flow area, see Figure 6.51. Define the reference values according to Fig. 6.43 **water** flow.

Report Type	Field Variable
Area-Weighted Average	Wall Fluxes
Custom Vectors Vectors of	Surface Heat Transfer Coef.
Custom Vectors	Surfaces Filter Text To Text Text Text Text Text Text Text Text
	Highlight Surfaces Area-Weighted Average (w/m2-k) 14.58478

Fig. 6.51 - Evaluation of the mean value of the **heat transfer coefficient** α on the **wall inner-shadow** for the **air** flow area

To evaluate the heat output P, select Total Heat Transfer Rate in the "Results / Report / Fluxes" command in Options and select the wall inner and wall inner-shadow in the Boundaries menu, see Figure 6.52.

E Flux Reports		×	
Options			
O Mass Flow Rate	Boundaries Filter Text 🔂 🗾 🔫	Results	
Total Heat Transfer Rate	inlet_air		
Radiation Heat Transfer Rate	inlet_water		
	interior-interior_air interior-interior water		
	outlet_air		
	outlet_water	05 77 4702002 (1022	
	wall_inner wall inner-shadow	-85.77470398241923 85.77470398241942	
	wall_outer		
	<		
Save Output Parameter		Net Results (w)	
Save output Parameter		1.98952e-13	
Compute Write Close Help			

Fig. 6.52 - Heat flow evaluation **P**[W]

Determine the **loss coefficient** ζ based on the respective pressures defined in the equation below.

$$\zeta = \frac{p_{1tot} - p_{2tot}}{p_{2dyn}} \tag{6.9}$$

Always use the "**Results / Report / Surface Integrals**" command to evaluate the pressures at the **inlet** and **outlet** of the flowing **water** and **air**. An example of evaluating the p1tot of the total inlet air pressure is shown in Figure 6.53. Then enter the value in Tab. 6.5. Identify the remaining pressure values identically (p_{2tot} , p_{2dyn}).

Surface Integrals	x
Report Type	Field Variable
Area-Weighted Average	Pressure
Custom Vectors Vectors of	Total Pressure 💌
· · · · · · · · · · · · · · · · · · ·	Surfaces Filter Text 🔁 🗮 🗮
Custom Vectors Save Output Parameter	inlet_air inlet_water outlet_air outlet_water rez-podelny rez-podelny-air rez-podelny-water wall_inner wall_outer
	Highlight Surfaces Area-Weighted Average (pascal)
Compute	Write Close Help

obr. 6.2 – Evaluation of p_{1tot} on inlet air

Calculation of the loss coefficient ζ for the airflow area:

$$\zeta = \frac{p_{1tot} - p_{2tot}}{p_{2dyn}} = \frac{9,16 - 5,8}{5,78} = 0,58 \tag{6.10}$$

Calculation of the loss coefficient ζ for the waterflow area:

$$\zeta = \frac{p_{1tot} - p_{2tot}}{p_{2dyn}} = \frac{71,48 - 47,08}{47,003} = 0,52$$
(6.11)

	Estimate for air	Estimate for water	CFD air solutions	CFD water solutions	Units
u	3	0.3	3	0.3	[m.s ⁻¹]
Re	4108	11943	4108	11943	[1]
Nu	15,9	91,62	12.05	82,18	[1]
α	19,2	1374,3	14.58	1232,7	[W.m ⁻² .K ⁻¹]
Р			85,77	85,77	[W]
p 1tot			7,53	61,85	[Pa]
p _{2tot}			5,59	45,73	[Pa]
p 2dyn			5,59	45,67	[Pa]
ζ			0,35	0,35	[1]

Tab. 6.5 – Final comparison of average results

u velocity

Re Reynolds number

Nu Nusselt number

- *α* Heat transfer coefficient
- *P* Heat output

p_{1tot} total pressure at the inlet

p_{2tot} total pressure at the outlet

p_{2dyn} dynamic pressure at outlet

 ζ loss coefficient

Conclusion

Deviations in the solution are caused both by estimating the **Nusselt number** analytically and by the numerical solution, where it is possible to test the influence of mesh quality, models and physical properties. In particular, the analytical relations of the Nusselt number estimate do not fully correspond to the characteristics of the given co-current exchanger task. They are intended to provide basic information about the Nusselt number estimate. Nusselt numbers obtained from analytical relations and numerical calculation are in order, which can be considered satisfactory.

The accuracy of the numerical calculation depends on the quality of the mesh, which can be subsequently compressed. There are several possible adaptations, for example, a mesh can be prepared and a comparison of results can be made.
7 HEAT DISTRIBUTION OF CONDUCTIONS AND AIR CONVECTIONS

Create a mathematical model theoretically analogous to a co-current exchanger with the difference that the ambient air will be defined instead of the outer tube. Perform a 3D simulation. The fluids are combined air-to-air. The model can be seen in Fig. 6.1. Define individual areas and parameters according to the specified boundary conditions and graphically evaluate the results.



Fig. 7.1 - Geometry and boundary conditions.

Tab. 7.1 – Area	dimensions
-----------------	------------

Tube length H1	0.5	m
Pipe diameter D1	0.04	m
Block using two	(0.0 -0.1 -0.1)	m
pointson the diagonal	(0.5 0.5 0.1)	

The pipe flows in the middle of the liquid - **water**, the wall is formed by a steel pipe of a given diameter. Also consider a **wall water** thickness of 003m. Wall material consider steel.

Surrounding is **air** bounded by atmospheric pressure, a condition of pressure outlet. The **outlet bottom** is an insulated wall.

Material	Steel	Water	Air	
density ρ	8030	998.2	1.225	[kg.m ⁻³]
specific heat capacity c_{ρ}	502.48	4182	1006.43	[J.kg ⁻¹ K ⁻¹]
thermal conductivity λ	16.27	0.6	0.0242	[W.m ⁻¹ K ⁻¹]
viscosity η		0.001003	0.000017894	[kg.m ⁻¹ s ⁻¹]

Tab. 7.2 – Physical properties of material (steel, water, air) at 300K

Tab. 7.3 – Boundary conditions

	Inlet	Outlet	Wall	Outlet	Wall	
	water	water	water	air	bottom	
temperature T	363.15		coupled		q=0	[K]
velocity u	0.3					[m.s ⁻¹]
pressure p		0		0		[Pa]
intensity of turbulence <i>I</i>	1	1		1		[%]
hydraulic diameter d_h	0.04	0.04		0.5		[m]

7.1 Mathematical model and theoretical-empirical estimation of the problem

In this task turbulent flow occurs, so the mathematical model RNG k- ϵ is used. The criterion of turbulence is the Reynolds number. There is almost no air flow, eg velocity is 0.001 m/s.

Re for water flow:

$$Re_{voda} = \frac{v \cdot d_h}{v} = \frac{0.3 \cdot 0.04}{1.01e - 06} = 12000$$
(7.1)

The calculation of the Nusselt number and the heat transfer coefficient is based on empirical relations, which are described in detail in the literature [2]. In the next step, only the analytical calculation is performed, which will be compared with the numerical calculation. From the given parameters it is possible to calculate the above parameters of flow and heat transfer (Reynolds number is calculated from the maximum speed). The estimation of the Nusselt number is problematic and is only indicative. This estimate is followed by the calculation of the wall heat transfer coefficient determined

from the Nusselt number $\stackrel{\approx}{\alpha} = \frac{\mathrm{Nu.}\lambda}{d}$ [2].

Calculation of the Nusselt number for the area of water flow in a pipe:

$$\Pr = \frac{\rho \cdot c_p \cdot v}{\lambda} = \frac{998.2 \cdot 4182 \cdot 1.01e - 6}{0.6} = 6.99$$
(7.2)

 $Nu = 0.023 \cdot Re^{0.8} \cdot Pr^{0.3}$ $Nu = 0.023 \cdot 12000^{0.8} \cdot 6.99^{0.3} = 75.5$ (7.3)

Then the heat transfer coefficient is

$$\alpha = \frac{Nu}{d_h} \cdot \lambda = \frac{75.5}{0.04} \cdot 0.6 = 1132.6 \ W.\ m^{-2}.\ K^{-1}$$
(7.4)

7.2 Geometry and mesh creation.

Geometry is given by two entities, ie a cylinder and a box, using a Boolean subtraction to create a water area and an air area. The methodology of networking is identical with the methodology described in chap. 6, ie the inflation and sweep method for the pipe and the inflation method for the air volume. The mesh has the following shape.



Fig. 7.2 - Surface mesh and detail with inflation.

7.3 Calculating the Gravity Problem.

Adjustments to deal with ambient heat transfer will be carried out in Fluent as follows:

The heat transmitted by the air flow to the surroundings is significantly influenced by gravity. It is entered eg in menu "**Physics / Operating Conditions / Gravity**" and density is specified in "**Physics / Operating Conditions / Operating Density**", whose value is 0. Then the stratification of pressure in the result can be observed.

e	Domain	Phys	sics	User-Defined	S	olution	Results		
	Solver	[Operat	ing Conditions					>
9 os E	🗸 Operating Co	nditions	Pressure			Gravity			
ieral	Reference Va	lues	Operating 101325	Pressure (pascal)	,	Gravity	Acceleratio	on	
ne View			Reference	ce Pressure Locati	DN	X (m/s2) 0			-
			X (m) 0		•	Y (m/s2) -9.81			-
er Text			Y (m) 0		*	Z (m/s2) 0		8	-
tup			Z (m) 0		•	Rouccinoca Da	ramotore	-	
Genera © Models	l ; als					Operating Tem 288.16	iperature (k)	-	
🗄 Cell Zo	ne Conditions					Variable-Dens	ity Parame	eters	
Bound Bound Source The second seco	ary Conditions et ernal tlet					Specified	Operating D sity (kg/m3)	ensity	,
⊕ ⊒ Wa	11					0		•	
Dynam	iic Mesh nce Values nce Frames		10000	ОК	Car	ncel Help			

Fig. 7.3 Operating Conditions

Physical properties of air will depend on temperature or pressure, so density is given by state equation and other physical properties by so-called kinetic theory.

Materials		
Materials		
Materials		
Fluid		
water-liquid		Q
air		
Solid	Create/Edit Materials	
aluminum	Name	Material Type
	air	fluid
	Chemical Formula	Fluent Fluid Materials
		air
		Mixture
		none
	Properties	
	Density (kg/m3)	ideal-gas
	Cp (Specific Heat) (j/kg-k)	kinetic-theory
	Thermal Conductivity (w/m-k)	Kinetic-theory
	Viscosity (ka/m-s)	kinetic-theory
	(ig),	
		· · ·
Create/Edit	Chang	ge/Create Delete Close Help

Fig. 7.4 Properties of Air

Too "loose" boundary condition of atmospheric pressure causes significant backflow and then divergence. Therefore, it is advantageous to use a velocity condition with a very small value, e.g. 0.001 m/s.

Boundary Conditions							
Zone Filter Text							
inlet_water interior-air interior-water outlet_air outlet_water	Velocity Inlet Zone Name outlet_air						
wall_bottom	Momentum Thermal Radiation Species DPM Multiphas						
wall_water-shadow	Velocity Specification Method Components						
	Reference Frame Absolute						
	Supersonic/Initial Gauge Pressure (pascal)						
_	Coordinate System Cartesian (X, Y, Z)						
	X-Velocity (m/s) 0.01						
	Y-Velocity (m/s) 0						
	Z-Velocity (m/s)						
	Turbulence						
Phase Type	Specification Method Intensity and Viscosity Ratio						
mixture velocity-inlet	Turbulent Intensity (%) 1						
Edit Copy	Turbulent Viscosity Ratio 0.5						
Display Mesh	OK Cancel Help						

Fig. 7.5 Velocity inlet for air

When gravitational acceleration is entered, hydrostatic pressure is generated automatically in the air and tube area. Therefore, the pressure condition at the water outlet from the pipe will be replaced by hydrostatic pressure.

Boundary Conditions	
Zone Filter Text	Pressure Outlet
inlat water	Zone Name
interior-air	outlet_water
interior-water outlet_air	Momentum Thermal Radiation Species DPM Multip
outlet_water wall bottom	Backflow Reference Frame Absolute
wall_water	Gauge Pressure -g*Position.y*Density
wall_water-shadow	Pressure Profile Multiplier
	Backflow Direction Specification Method Normal to Boundary
	Backflow Pressure Specification Total Pressure
	Prevent Reverse Flow
	Radial Equilibrium Pressure Distribution
	Average Pressure Specification
	Target Mass Flow Rate
	Turbulence
Phase Type	Specification Method Intensity and Viscosity Ratio
mixture 🔻 pressure-ou	Backflow Turbulent Intensity (%)
Edit Copy	Backflow Turbulent Viscosity Ratio 0.02
Parameters Display Mesh	OK Cancel Help

Fig. 7.6 Pressure outlet for water

Then the task converges well and the results will be real.

7.4 Výsledky

Initialization is realized mainly by real temperature values (300 K). PATCH at 363 K is used for the water area.



Fig. 7.8 - Contours of the hydrostatic pressure at the air / pipe boundary and pressure drop in the pipe (PLOT XY)).



Fig. 7.9 - Temperature distribution in axial section of the area with air heating above the pipe.