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 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:

$$\iiint_V \frac{\partial(\rho\zeta)}{\partial t}\mathrm{d}V \quad + \quad \iint_S \left(\rho\zeta\,\vec{u}\cdot\vec{n}\right)\mathrm{d}S \quad = \quad \iint_S \left[\alpha_\zeta \nabla\zeta\right]\mathrm{d}S \quad + \quad \iiint_V S_\zeta\,\mathrm{d}V \tag{1.1}$$

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)):

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

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 three-dimensional 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.

Temperatures

$t_{h,I}$ - hot liquid inlet
$t_{h,O}$ - cooled fluid outlet
$t_{c,I}$ - cold fluid inlet
$t_{c,O}$ - heated fluid outlet
$t_{h,s}$ - fixed wall temperature, hot side
$t_{c,s}$ - fixed wall temperature, cold side
t_h - temperature curve in the cooled liquid
t_c - temperature curve in the heated liquid

Fluid flows

$Q_{m,c}$ - mass flow of heated fluid
$Q_{m,h}$ - mass flow of cooled liquid

S - heat exchange surface
q - heat flow
d - thickness of solid wall
$\alpha_{c,h}$ - heat transfer coefficient

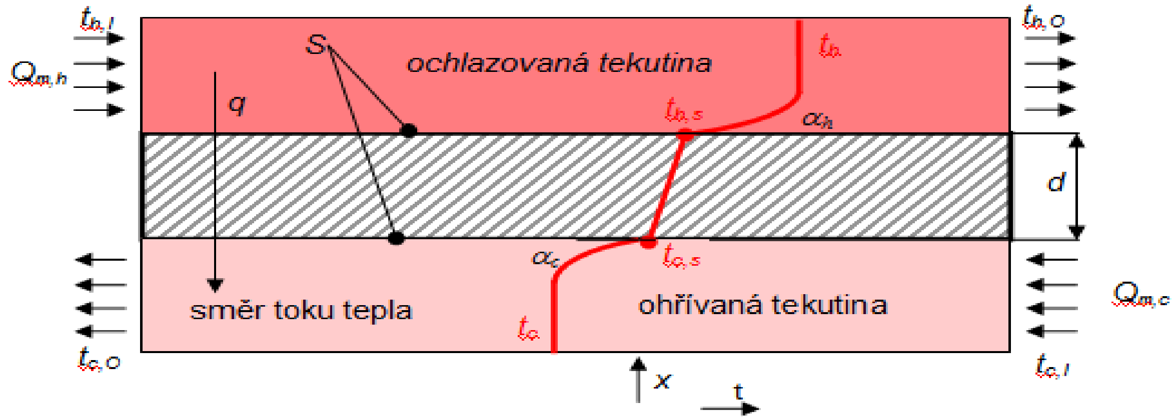


Fig. 2.1 - Scheme of fluid and heat flows through the exchanger (counterflow exchanger)

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

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

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 [m^2], 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

$$\begin{aligned} P &= \alpha_c (t_{c,s} - t_c) S \\ P &= \alpha_h (t_{h,s} - t_h) S \end{aligned} \tag{2.2}$$

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**

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

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

$$\Delta T_{lm} = \frac{(T_s - T_I) - (T_s - T_O)}{\ln\left(\frac{(T_s - T_I)}{(T_s - T_O)}\right)} \tag{2.5}$$

where T_I, T_O are the inlet and outlet temperatures of the flowing medium. The outlet temperature that is needed to determine ΔT_{lm} can be estimated from the relationship

$$\frac{T_s - T_O}{T_s - T_I} = \exp\left(-\frac{\pi d N \bar{\alpha}}{\rho v N_T S_T c_p}\right) \Rightarrow T_O = -\left(\exp\left(-\frac{\pi d N \bar{\alpha}}{\rho v N_T S_T c_p}\right).(T_s - T_I) - T_s\right)$$

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

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

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:

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

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

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

l 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.

$$\Delta p = N_L \zeta \left(\frac{\rho u_{\max}^2}{2}\right) \text{ resp. } \Delta p = N_L \zeta \left(\frac{8 Q_m^2}{\rho \pi^2 d^4}\right) \tag{2.8}$$

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

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

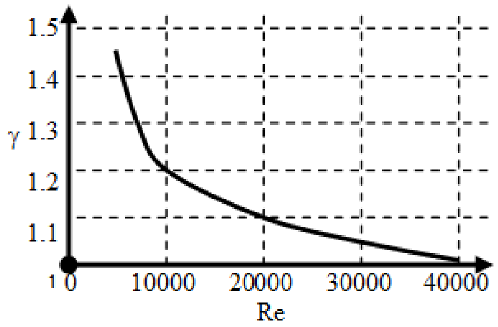
Where $A = 0.028\left(\frac{S_T}{2a}\right)^2$ $a = \frac{S_T - d}{2}$ $B = \left(\frac{S_T}{2a} - 1\right)^2$ (2.9)

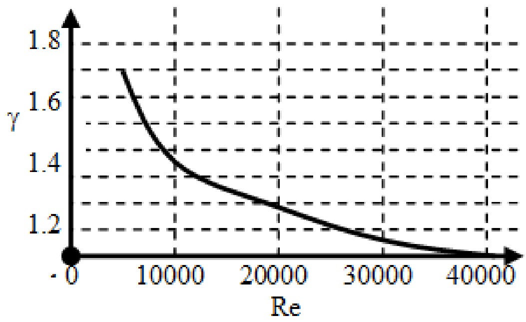
In cross-sectional arrangement of tubes, it is similarly defined:

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

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.





arrangement behind cross arrangement

Fig. 2.2 - Coefficient values γ depending on Re number

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.

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

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].

$$\mathrm{Re} = \frac{u d_h}{\nu} \tag{2.12}$$

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.

$$\mathrm{Pr} = \frac{\rho c_p \nu}{\lambda} = \frac{\nu}{a} \tag{2.13}$$

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

$$\mathrm{Fo} = \frac{\lambda \tau}{c_p \rho d_h^{\,2}} \tag{2.14}$$

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).

$$\mathrm{Nu} = \frac{\alpha d_h}{\lambda} \tag{2.15}$$

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.

$$\mathrm{Nu} = \frac{P\, d_h}{S\, \Delta T\, \lambda} \tag{2.16}$$

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

$$\mathrm{Nu} = f(\mathrm{Re}, \mathrm{Pr}, \mathrm{Fo}) \tag{2.17}$$

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 plate, T_S is constant	$\mathrm{Nu} = 0{,}664\,\mathrm{Re}_L^{1/2}\,\mathrm{Pr}^{1/3}$ $\quad 0{,}6 \le \mathrm{Pr}$ $\mathrm{Re}_L = \dfrac{uL}{\nu}$, $10^4 \le \mathrm{Re}_L \le 5.10^5$, L plate length
laminar flow around the plate, q is constant	$\mathrm{Nu} = 0{,}908\,\mathrm{Re}_L^{1/2}\,\mathrm{Pr}^{1/3}$ $\quad 0{,}6 \le \mathrm{Pr}$ $\mathrm{Re}_L = \dfrac{uL}{\nu}$, $10^4 \le \mathrm{Re}_L \le 5.10^5$, L plate length
turbulent flow around the plate, T_S is constant	$\mathrm{Nu}_x = 0.0405\,\mathrm{Re}_{L.}^{4/5}\,\mathrm{Pr}^{1/3}$ $\quad 0{,}6 \le \mathrm{Pr} \le 60$ $5.10^5 \le \mathrm{Re}_L \le 10^8$
laminar flow in the tube	Nu=4.36 pro q=const. for wall Nu=3.66 pro T=const. for wall
turbulent flow in the pipe	$\mathrm{Nu} = 0{,}023\,\mathrm{Re}^{0{,}8}\,\mathrm{Pr}^m$, $\quad$ m=0.3 for cooling $3.10^4 \le \mathrm{Re}_L \le 10^6$ $\quad$ m=0.4 for heating
laminar, transitional and turbulent transverse pipe wrap	$\mathrm{Nu} = C_1\,\mathrm{Re}^{C_2}\,\mathrm{Pr}^{0{,}38}$ 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 turbulent wrapping of the tube bundle, N_L is the number of tubes	$\mathrm{Nu}_D = C_1\,\mathrm{Re}_{D,\max}^m$ pro $N_L \rangle 10$, $2000 \le \mathrm{Re}_{D,\max}^m \le 40000$ $\quad \mathrm{Pr} = 0.7$, constants C_1 a m are given in the table S_L – horizontal pipe spacing, S_T – vertical pipe spacing

Tab. 2.2 Constants for determining the Nusselt number when wrapping a bundle of tubes

direct system	$S_T/D=$ 1.25		$S_T/D=$ 1.50		$S_T/D=$ 2.00		$S_T/D=$ 3.00	
S_L/D	$C1$	m	$C1$	m	$C1$	m	$C1$	m
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	$S_T/D=$ 1.25		$S_T/D=$ 1.50		$S_T/D=$ 2.00		$S_T/D=$ 3.00	
S_L/D	C_1	m	C_1	m	C_1	m	C_1	m
1.000			0.497	0.558				
1.125					0.478	0.565	0.518	0.560
1.250	0.518	0.556	0.505	0.554	0.519	0.556	0.522	0.562
1.500	0.451	0.568	0.460	0.562	0.452	0.568	0.488	0.568
2.000	0.404	0.572	0.416	0.568	0.482	0.556	0.449	0.570
3.000	0.310	0.592	0.356	0.580	0.448	0.562	0.482	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.

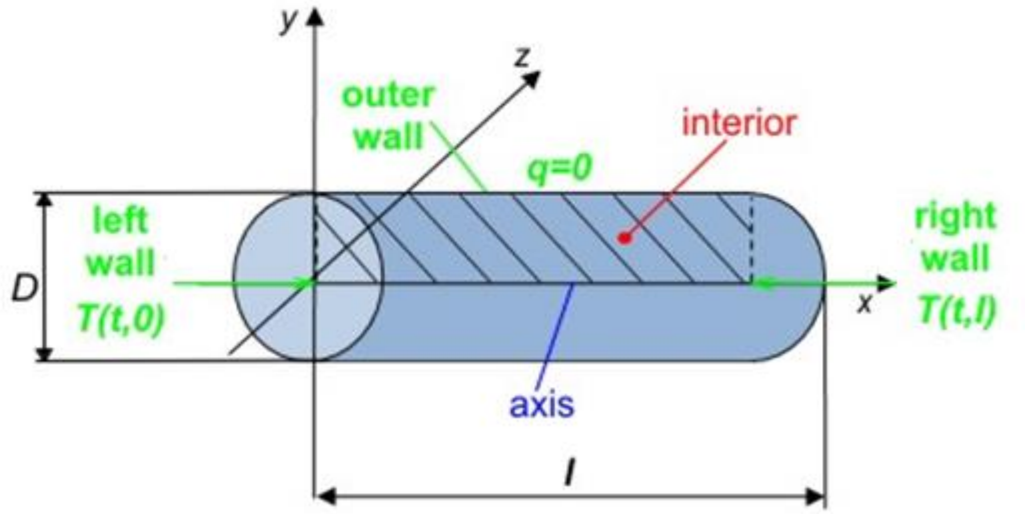


Fig. 3.1 - Bar of defined length

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 *l* **[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_p **[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
$T_0 = 50°C$	$T_l = -10°C$	$q = 0\ W/m^2$

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.

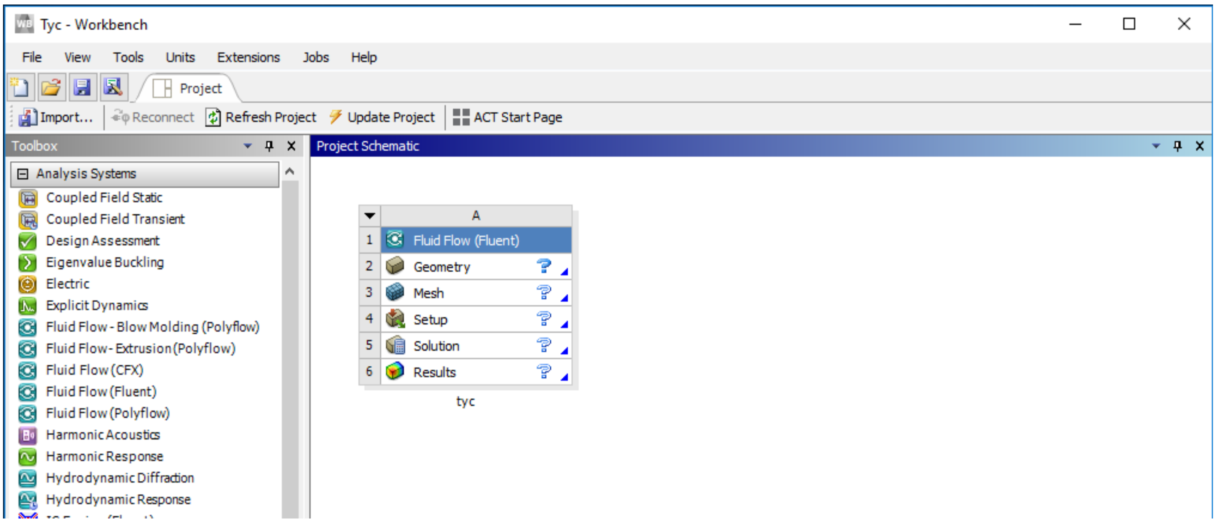


*Fig. 3.2 - Working environment of **ANSYS Workbench 2019 R3** with Fluid flow block.*

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..

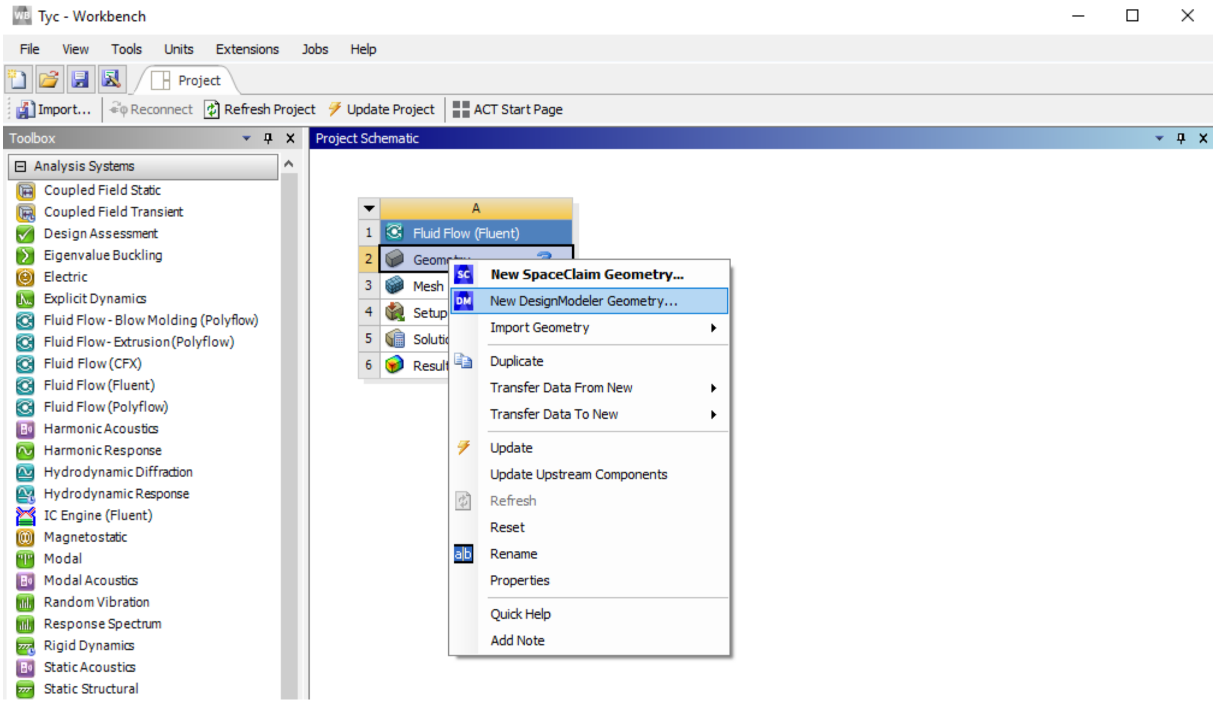


Fig. 3.3 - Starting DesignModeller

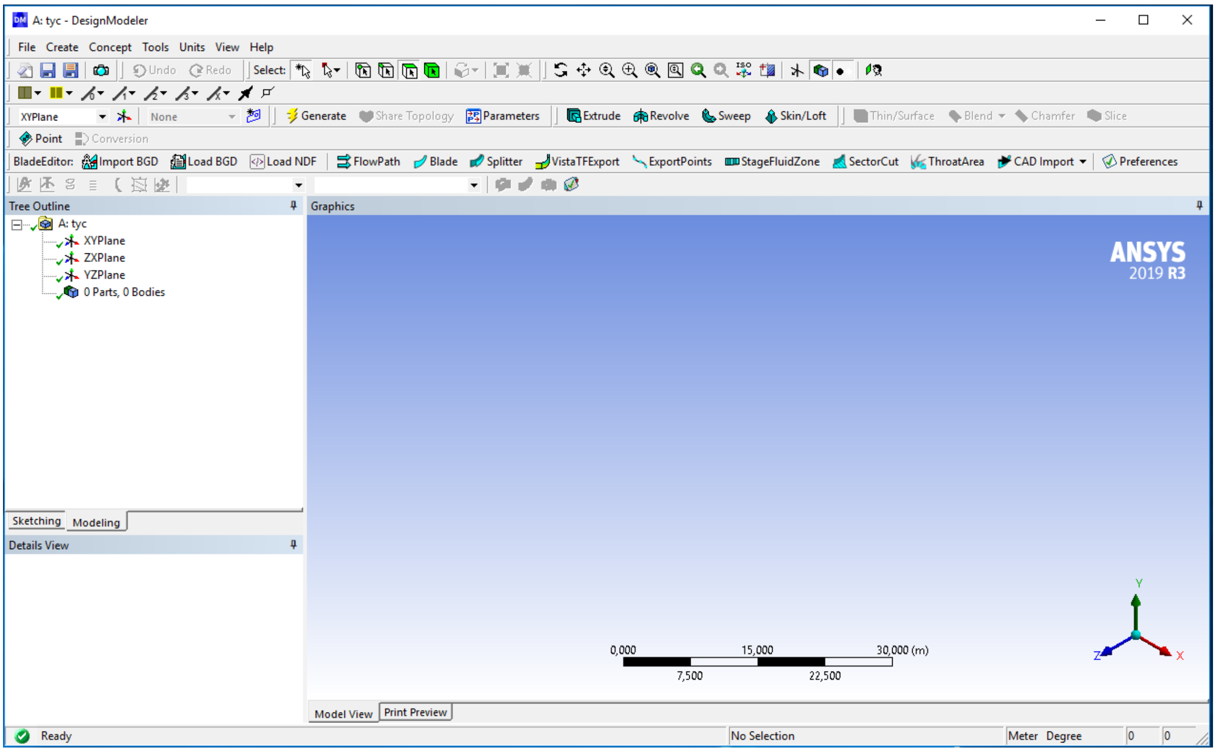


Fig. 3.4 - Program DesignModeler

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.

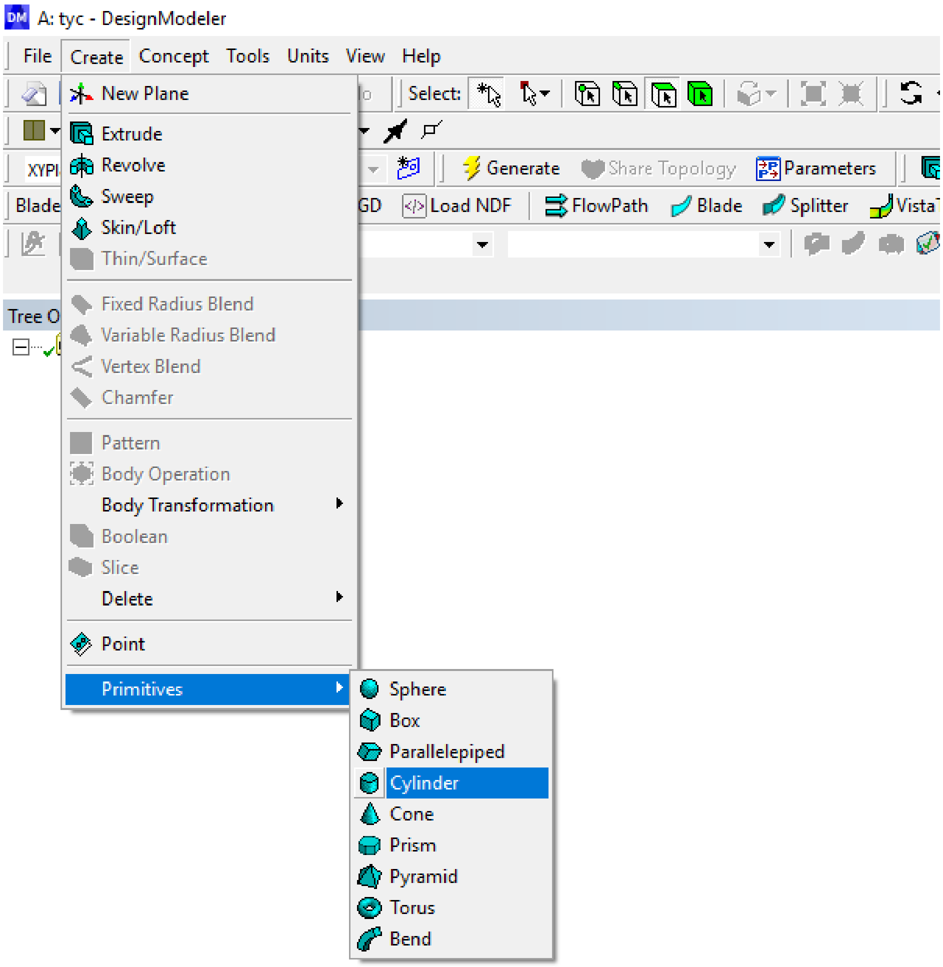
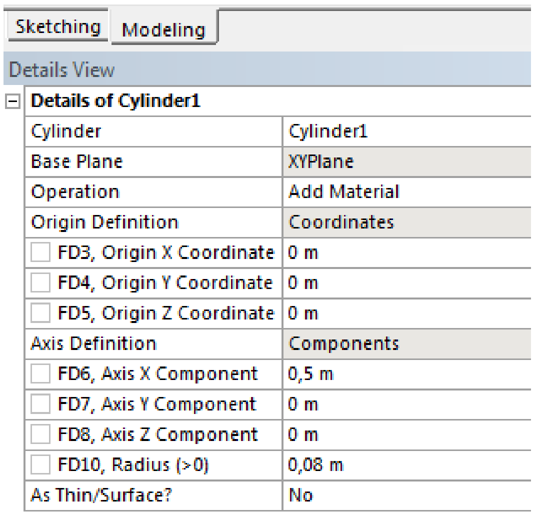


Fig. 3.5 - Creating cylinder geometry


Sketching | Modeling

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
FD5, Origin Z Coordinate	0 m
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

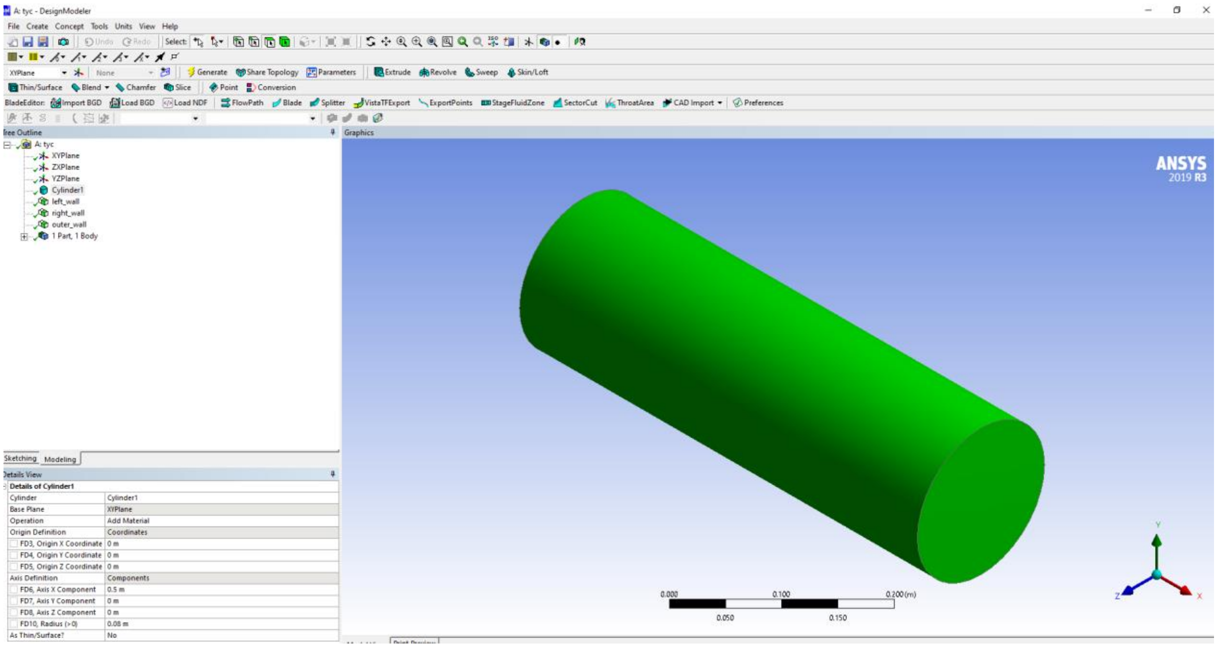


Fig. 3.7 - Resulting geometry

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).

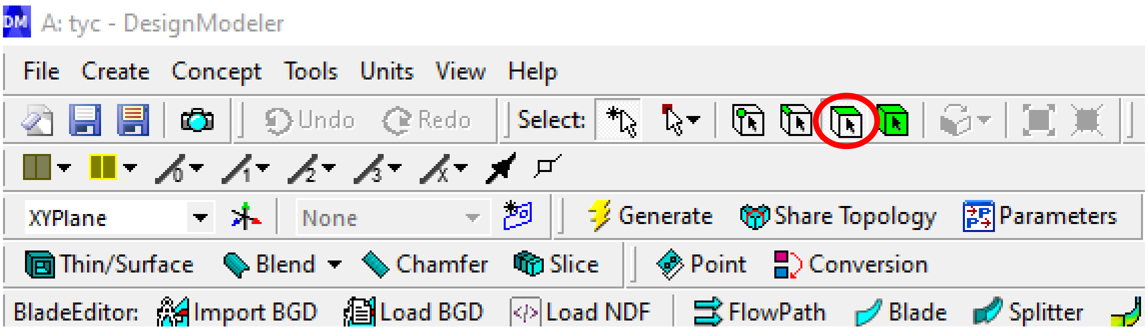


*Fig. 3.8 - Selection of type of face selection mode (“**Face**”)*

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.

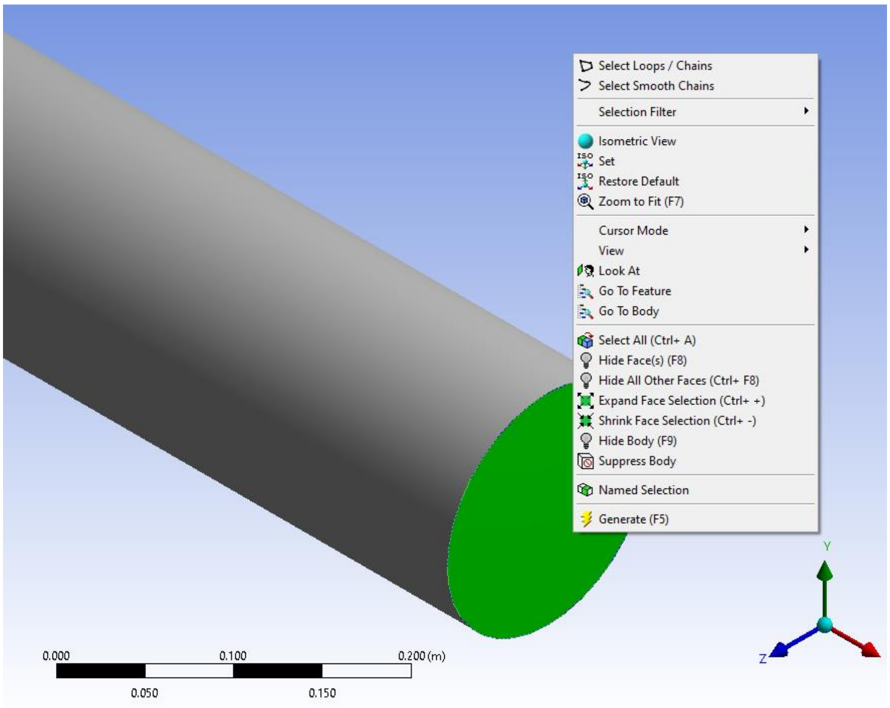


*Fig. 3.9 - Area selection for naming the boundary condition ("**Named Selection**")*

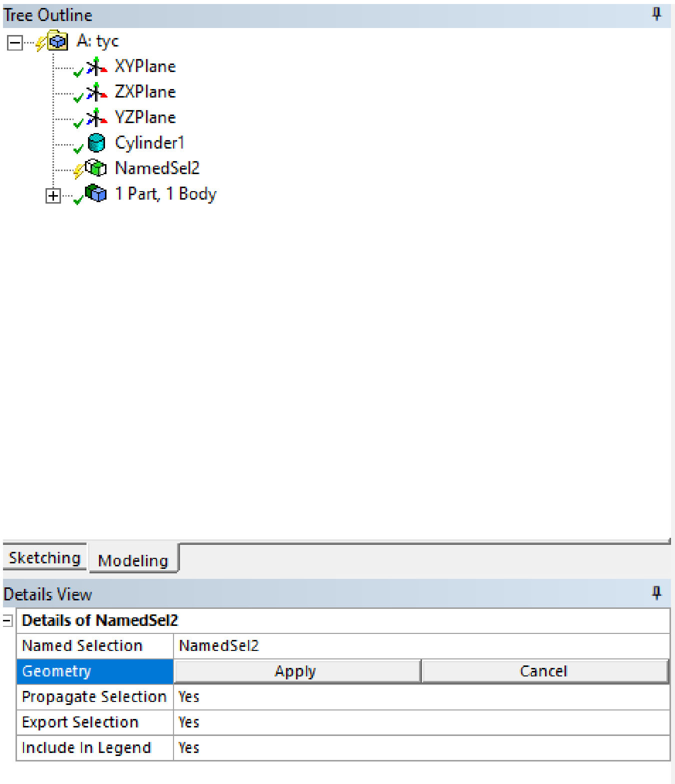


Fig. 3.10 - Naming the boundary condition

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

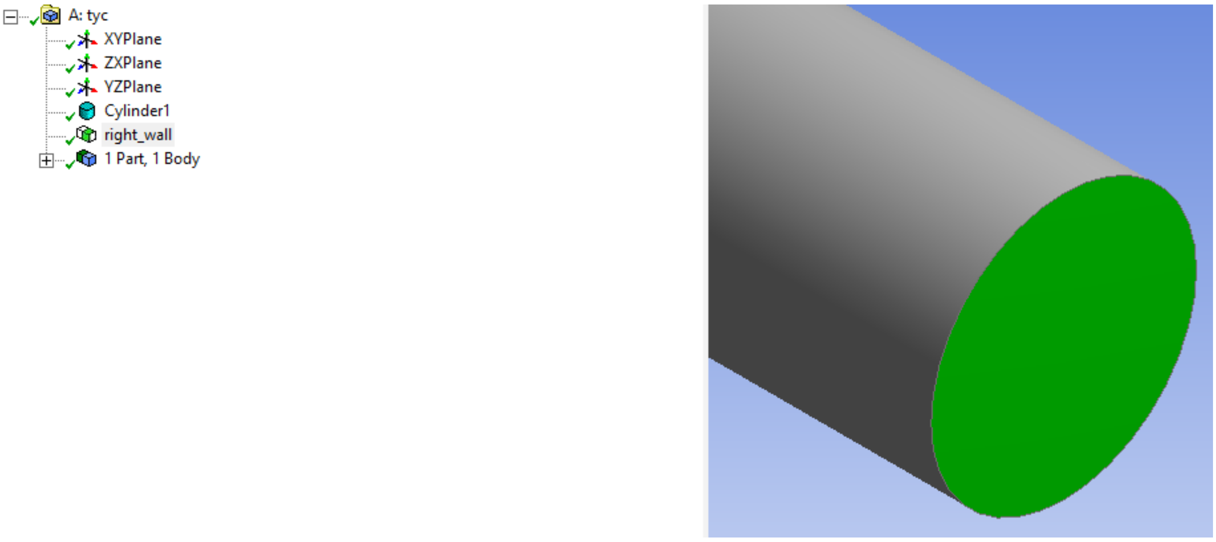


*Fig. 3.11 - Representation of the boundary condition "**right_wall**"*

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.

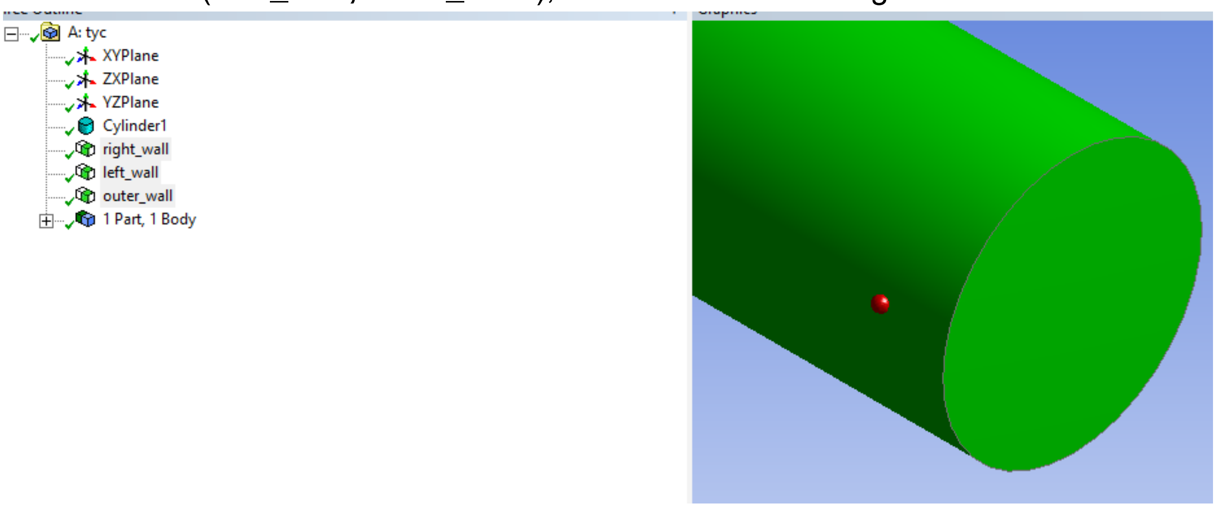


Fig. 3.12 - Naming boundary conditions

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).

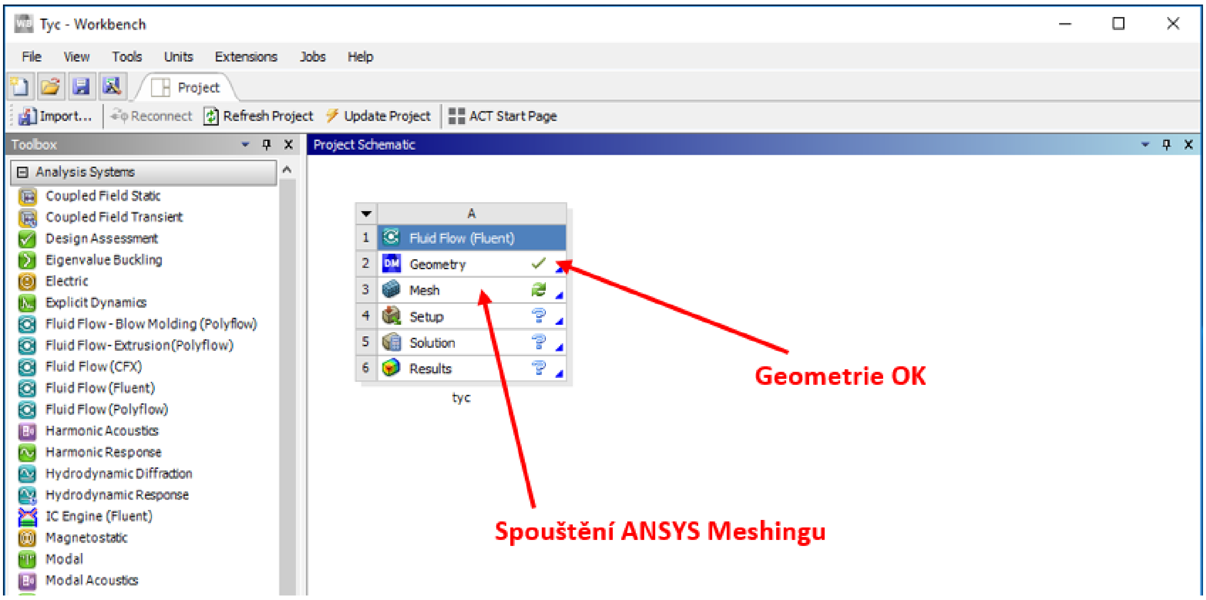


Fig. 3.13 - Workbench environment after creating geometry without errors

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.

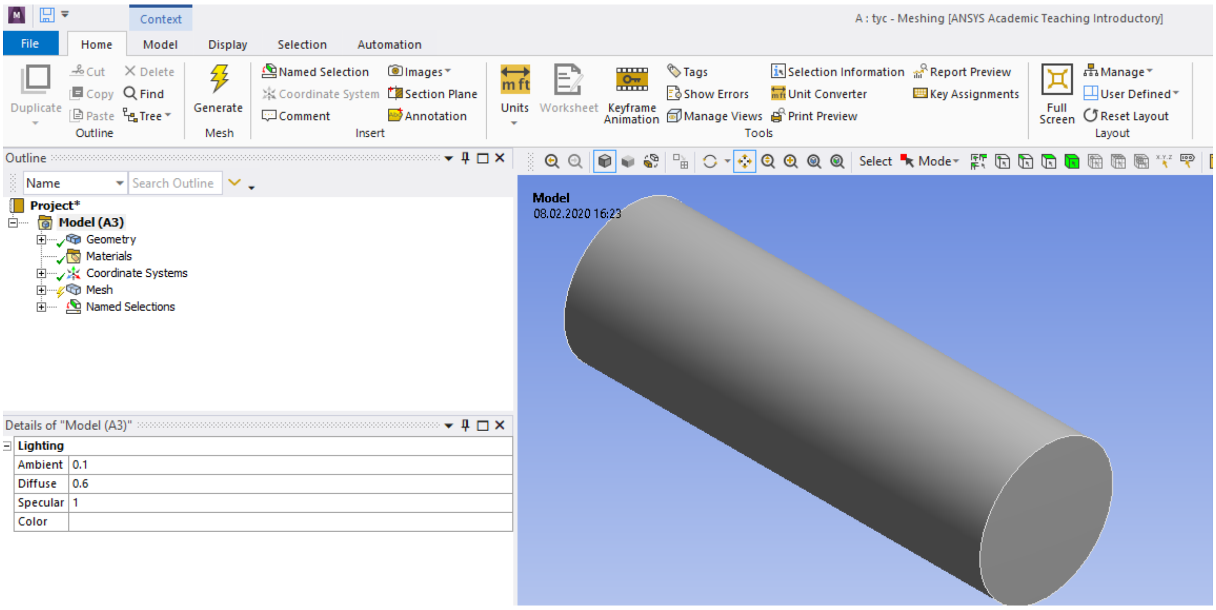


*Fig. 3.14 - Environment of the **ANSYS Meshing** program*

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

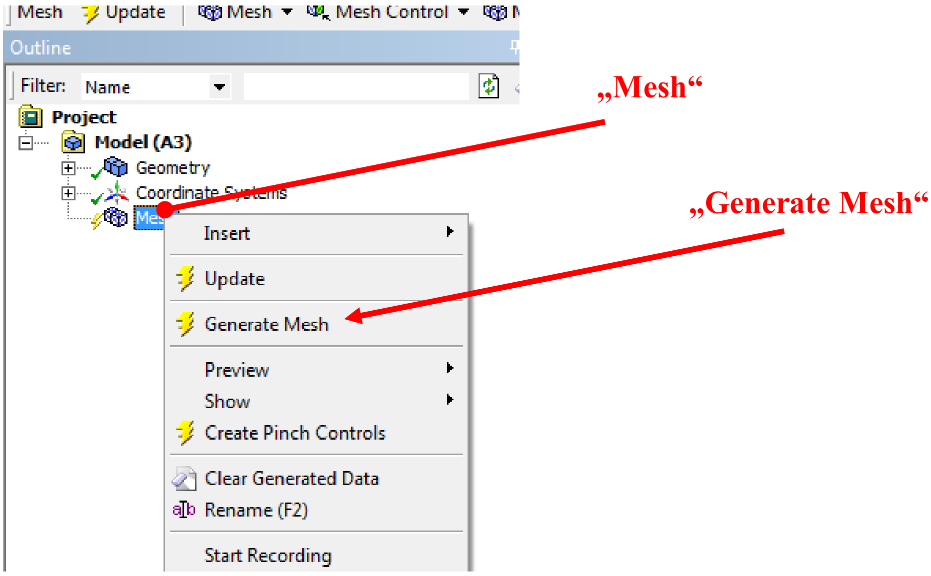


Fig. 3.15 - Creating a simple automatic network

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**. Ie. 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**”.

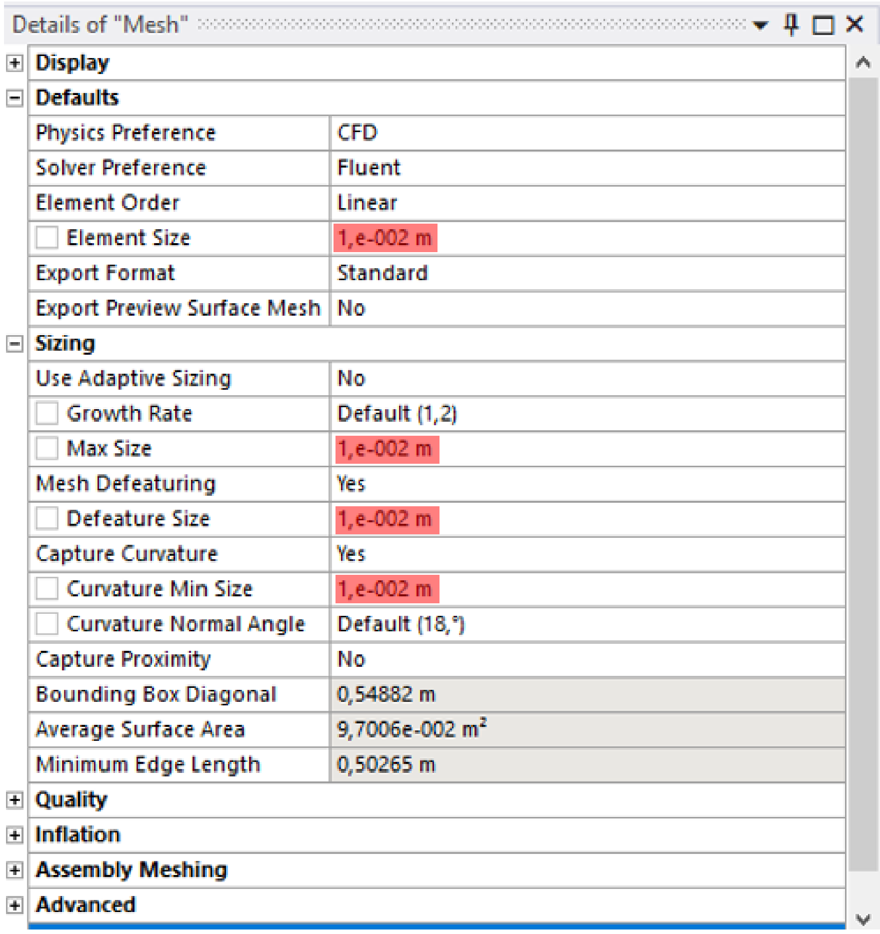
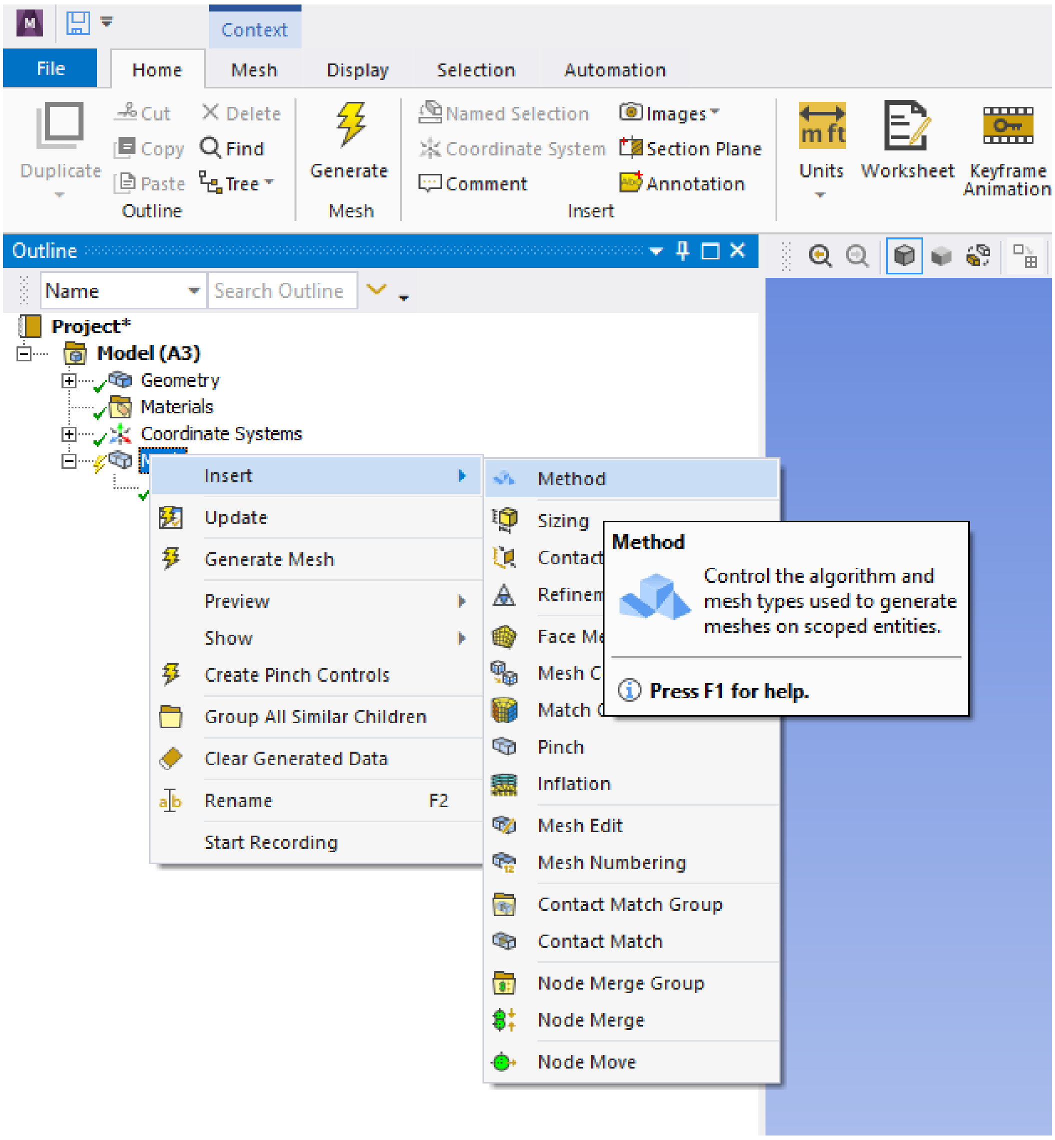


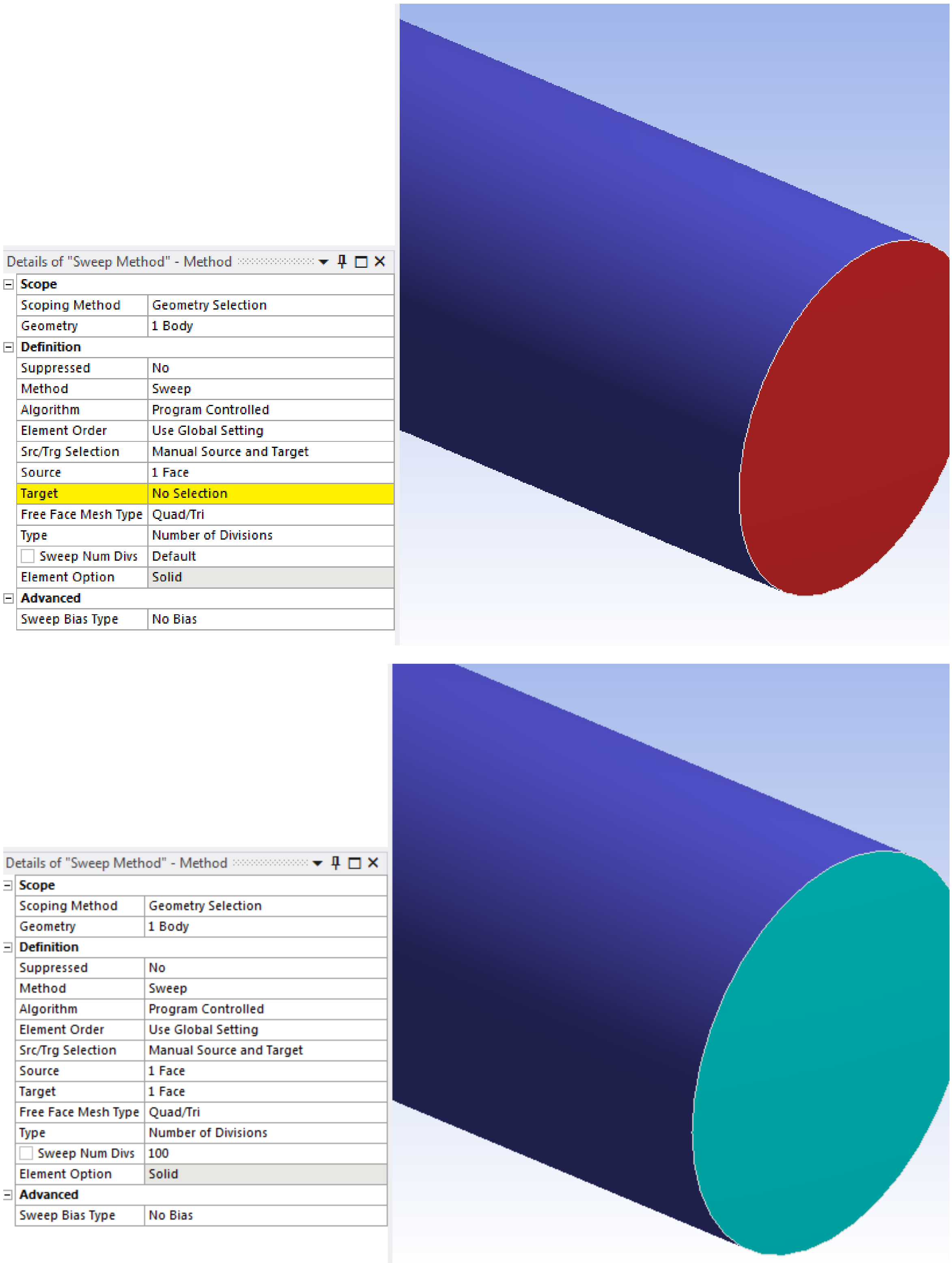
Fig. 3.16 - Details of Mesh

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.



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.



Details of "Sweep Method" - Method

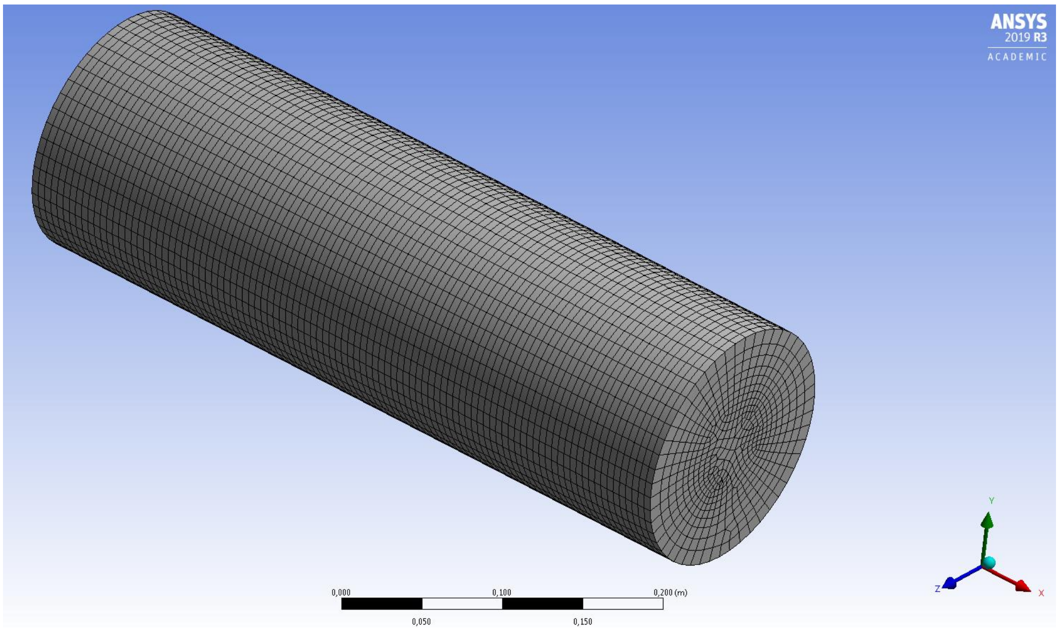
Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Method	Sweep
Algorithm	Program Controlled
Element Order	Use Global Setting
Src/Trg Selection	Manual Source and Target
Source	1 Face
Target	No Selection
Free Face Mesh Type	Quad/Tri
Type	Number of Divisions
Sweep Num Divs	Default
Element Option	Solid
Advanced	
Sweep Bias Type	No Bias

Details of "Sweep Method" - Method

Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Method	Sweep
Algorithm	Program Controlled
Element Order	Use Global Setting
Src/Trg Selection	Manual Source and Target
Source	1 Face
Target	1 Face
Free Face Mesh Type	Quad/Tri
Type	Number of Divisions
Sweep Num Divs	100
Element Option	Solid
Advanced	
Sweep Bias Type	No Bias

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).

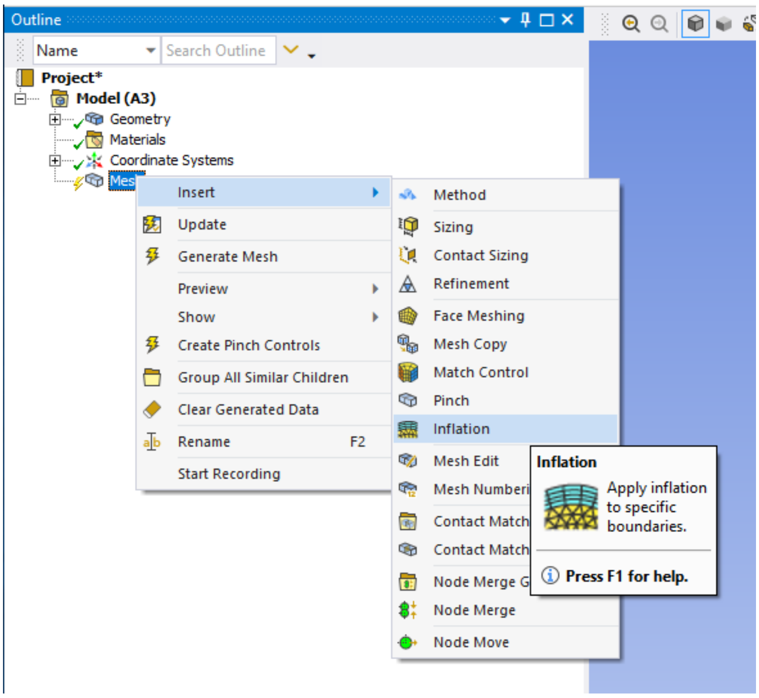


*Fig. 3.20 - Selecting the "**Inflation**"*

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.

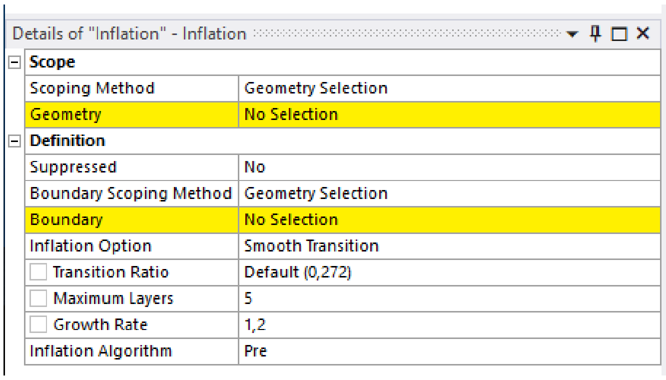


*Fig. 3.21 - **Inflation** characteristics*

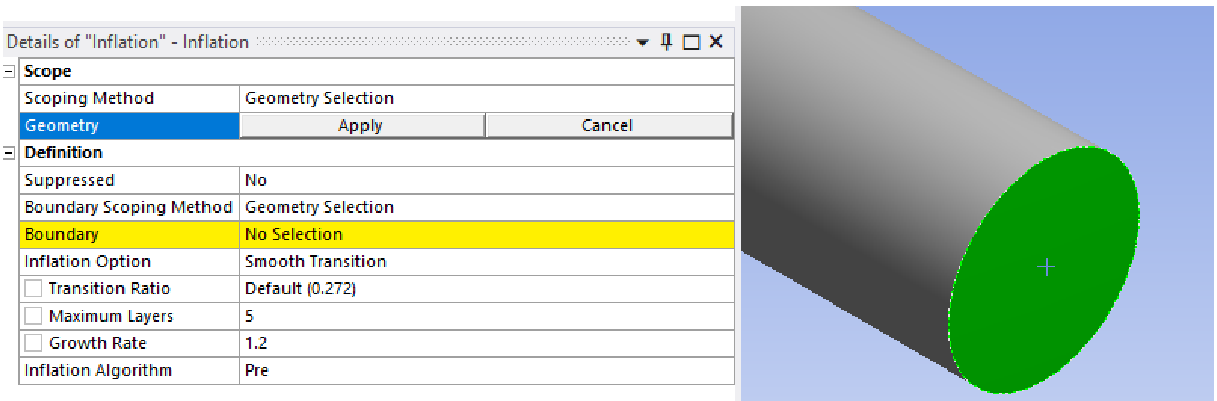


*Fig. 3.22 - Selection of the area in which **Inflation** will be created*

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.

Details of "Inflation" - Inflation

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)
Maximum Layers	5
Growth Rate	1.2
Inflation Algorithm	Pre

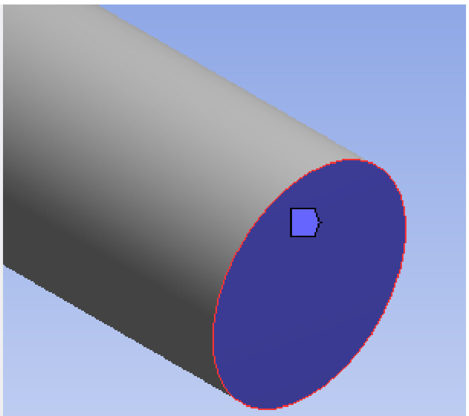
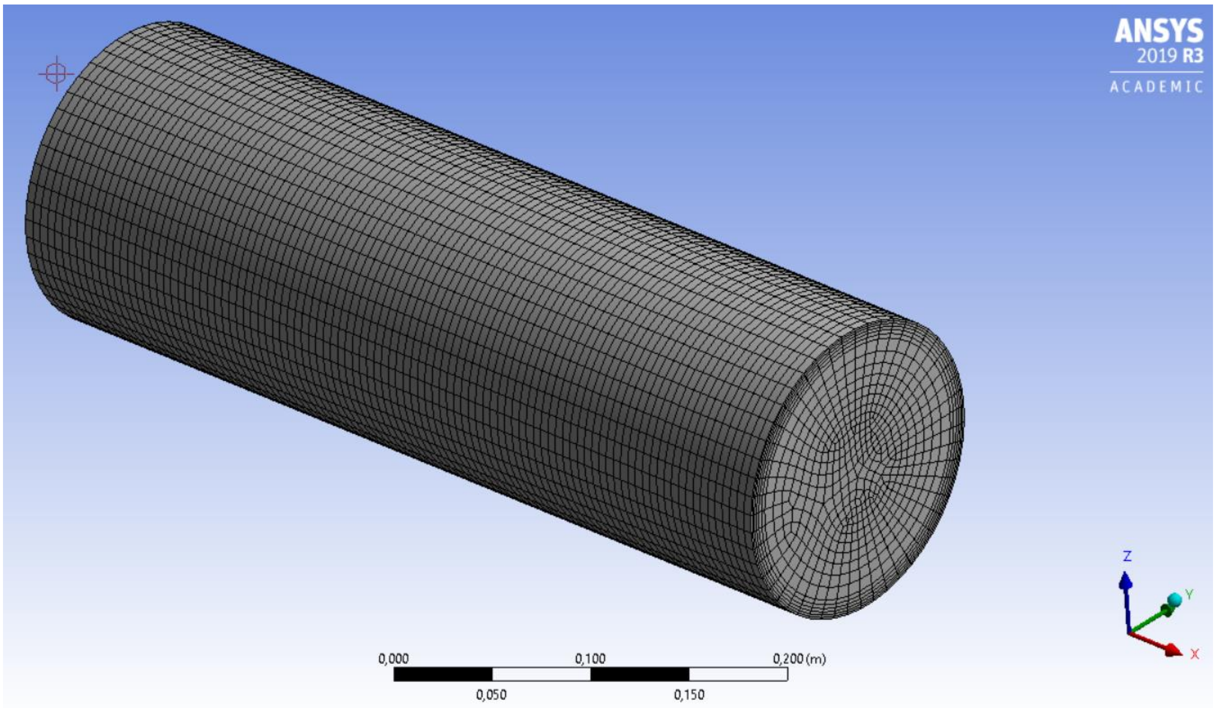


*Fig. 3.23 - Edge selection to define **Inflation***

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).

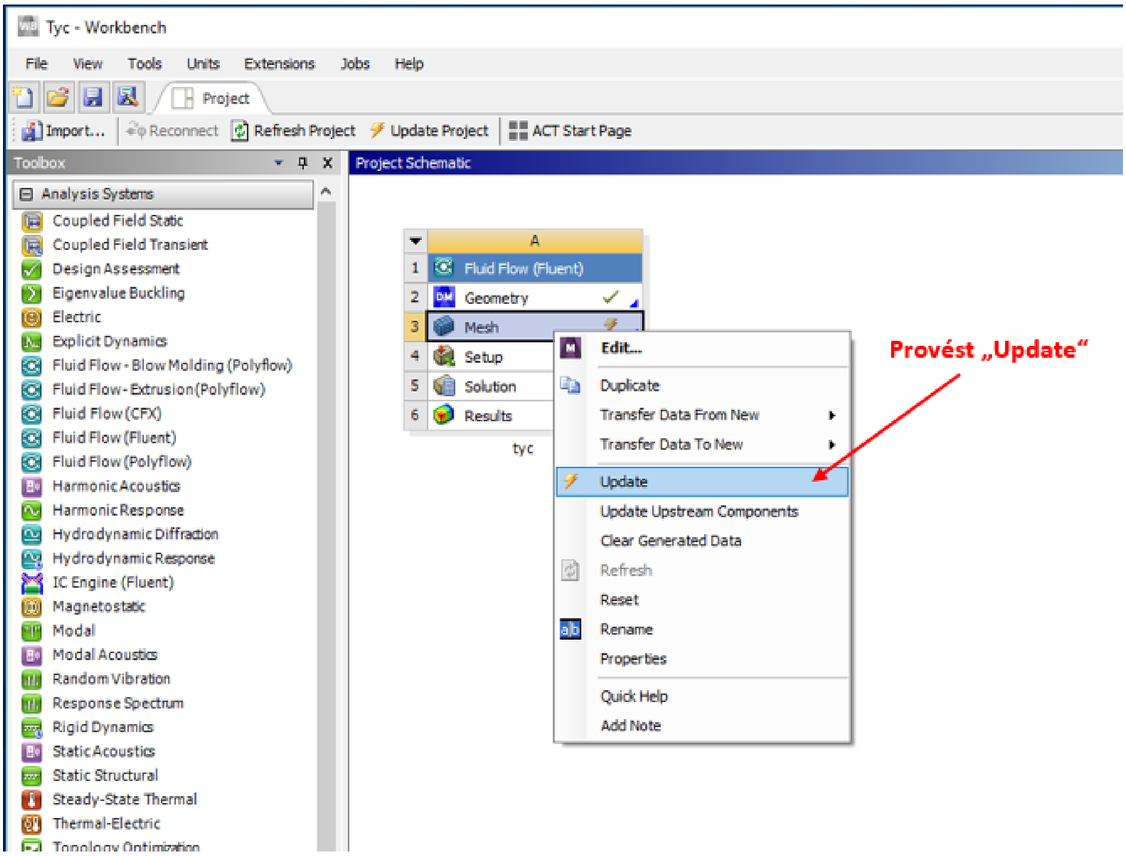


Fig. 3.25 - Illustration of the Update mesh check mark

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

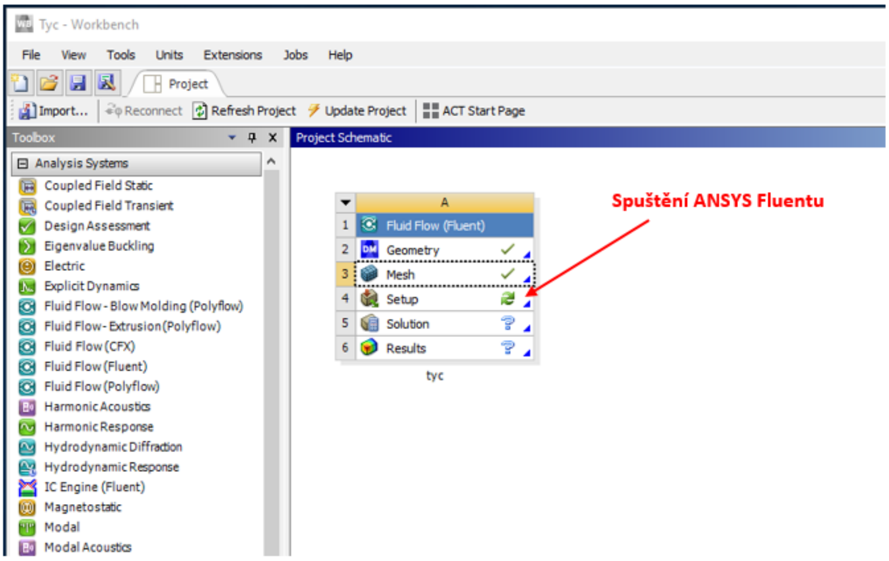


Fig. 3.26 - Resulting panel project

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.

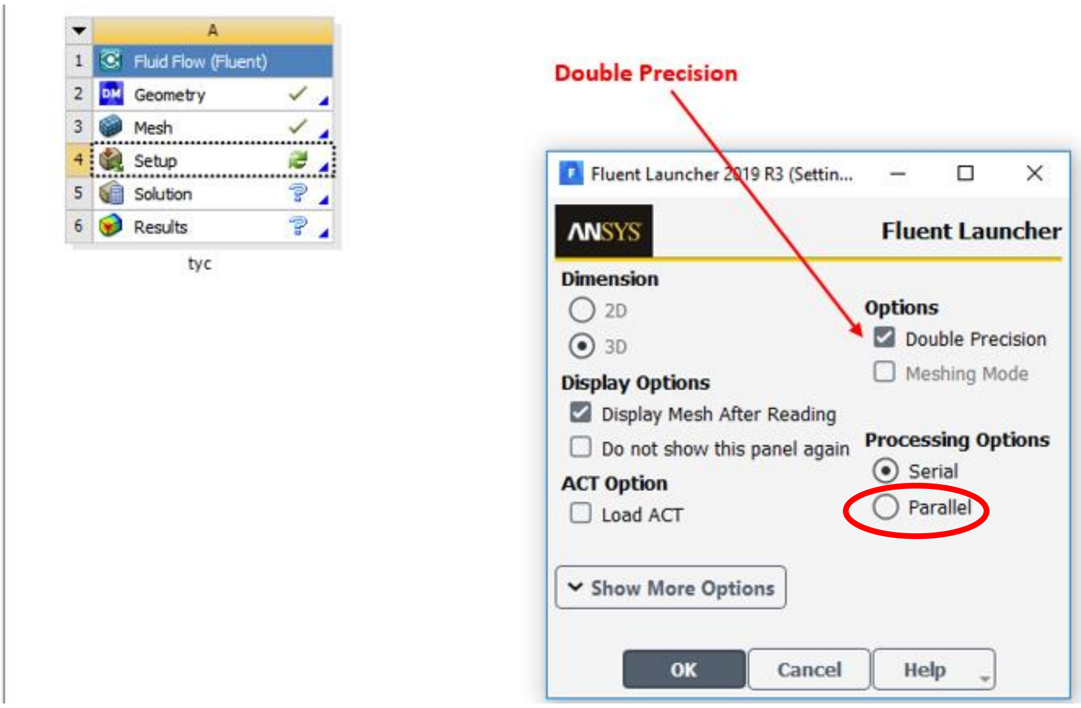


*Fig. 3.27 - Starting **ANSYS Fluent 2019 R3***

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

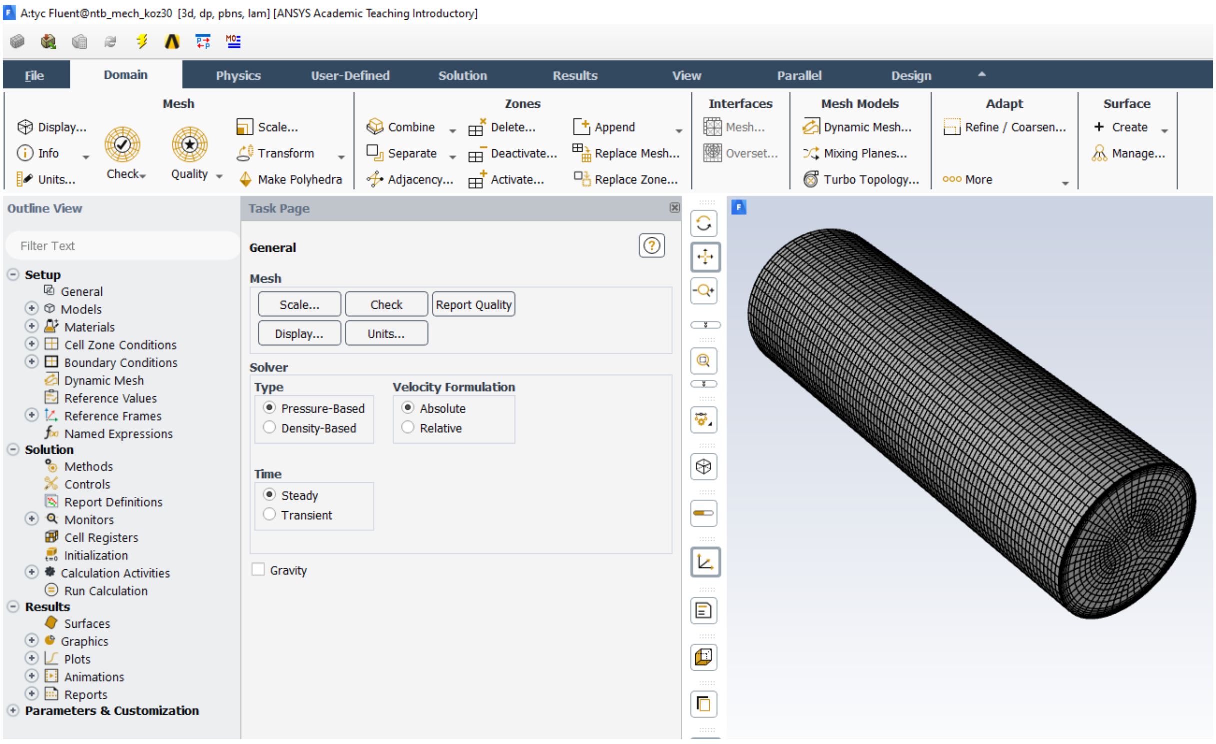


*Fig. 3.28 - Basic environment of the program **ANSYS Fluent***

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.

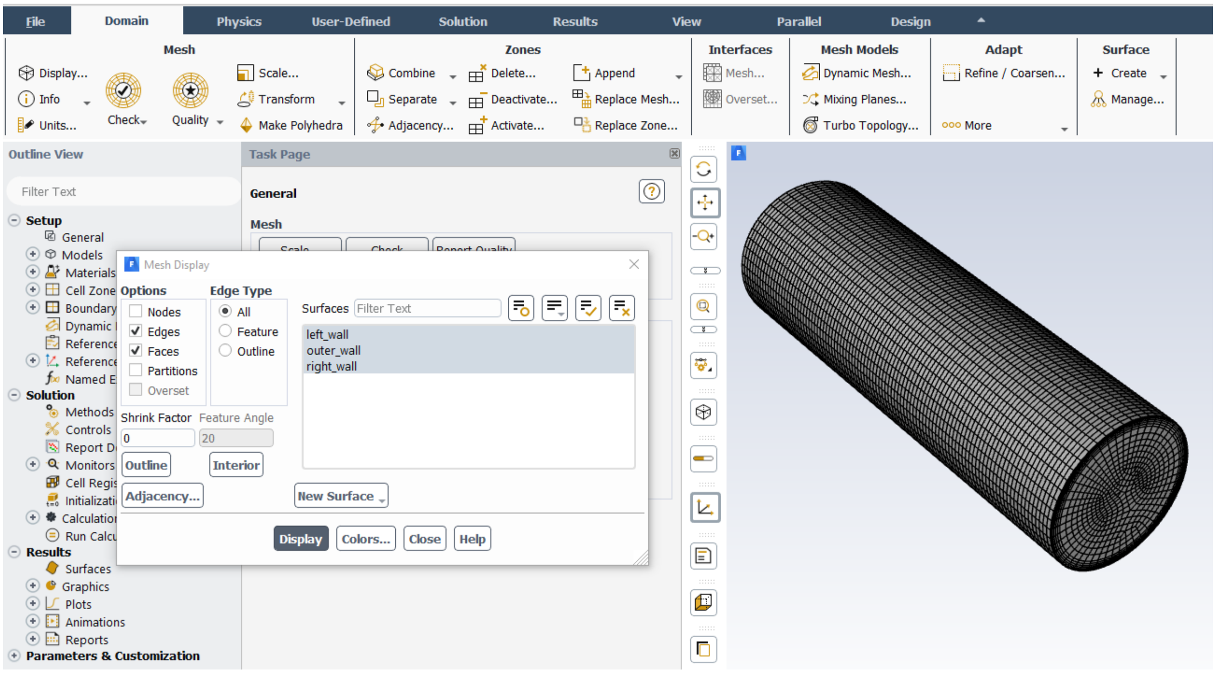


Fig. 3.29 - Mesh and boundary conditions check

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).

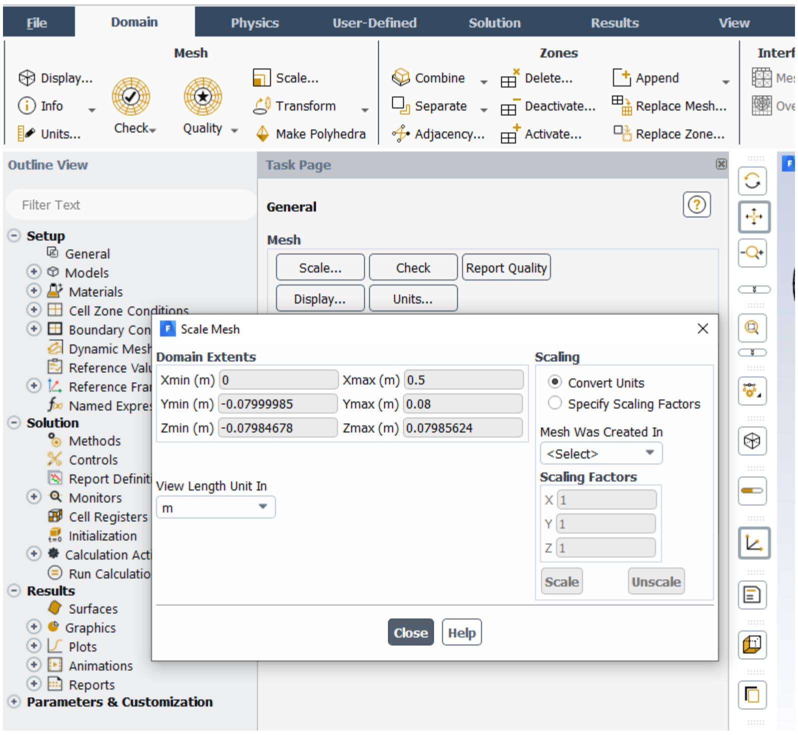


Fig. 3.30 - Checking the unit of dimensions

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

 1 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).

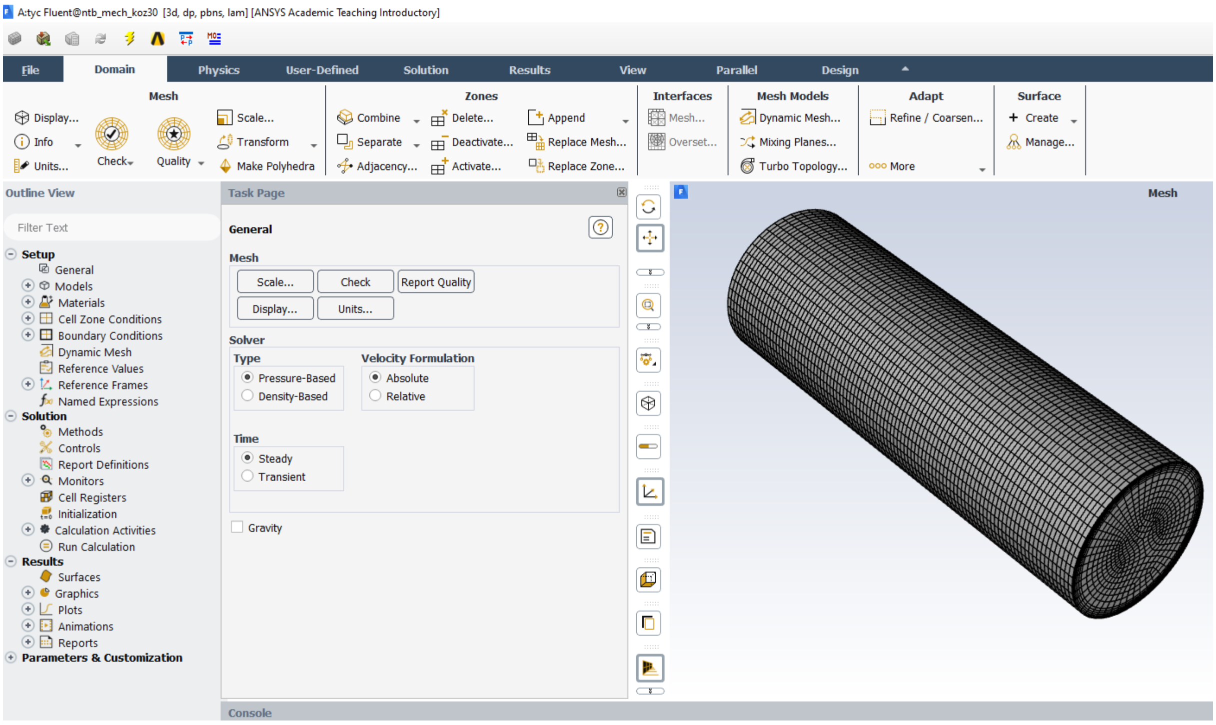


Fig. 3.33 - Menu of commands defining a mathematical model

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.

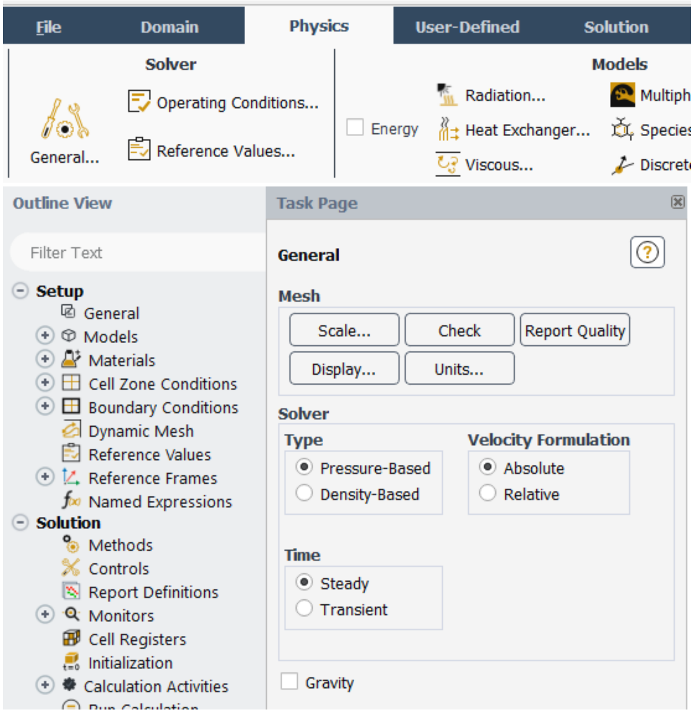


*Fig. 3.34 - Commands from the "**Solver**"*

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)).

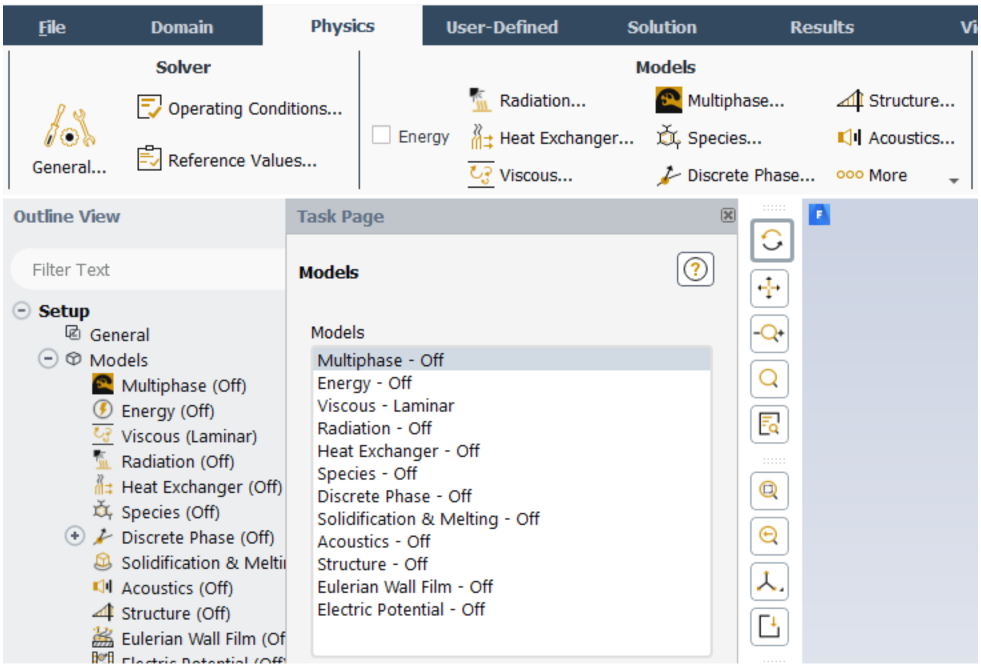


*Fig. 3.35 - Characteristics of the "**Models**"*

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).

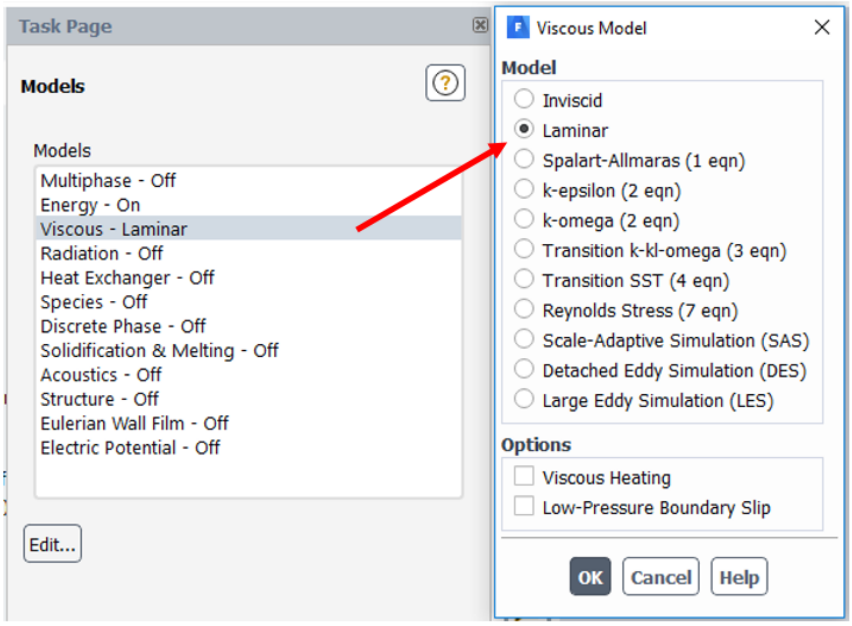


Fig. 3.36 - Setting the mathematical model of the solved problem

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.

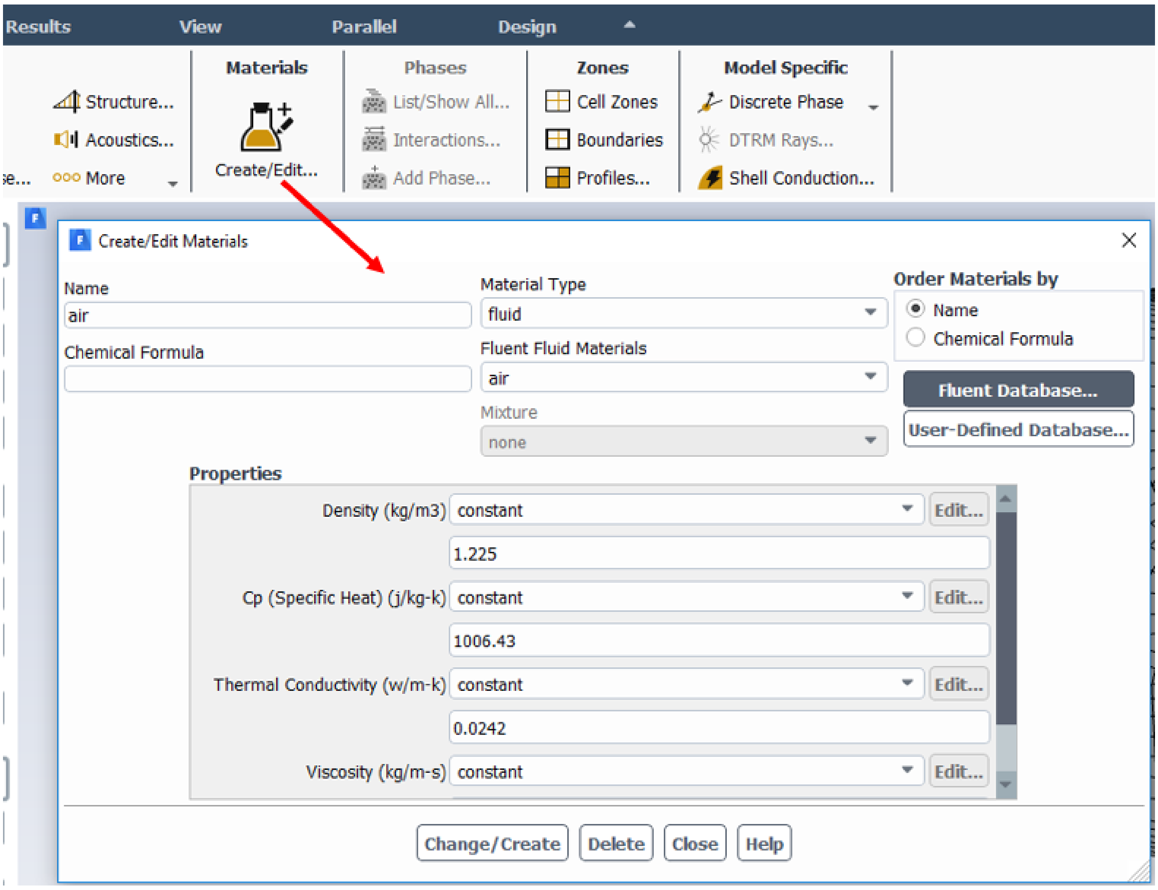


*Fig. 3.37 Characteristics of the “**Materials**“*

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.

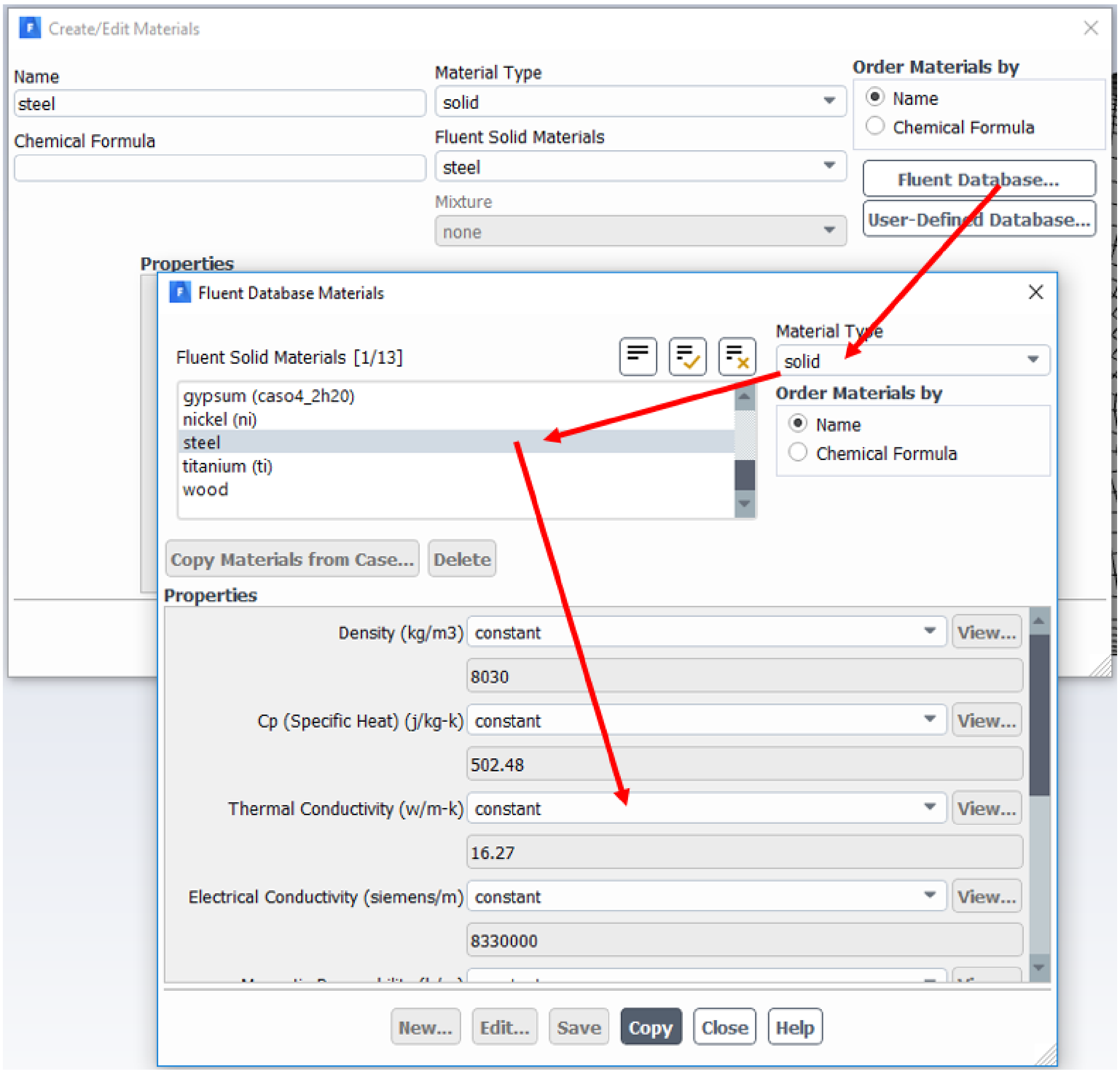


Fig. 3.38 - Steel selection from ANSYS Fluent database

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

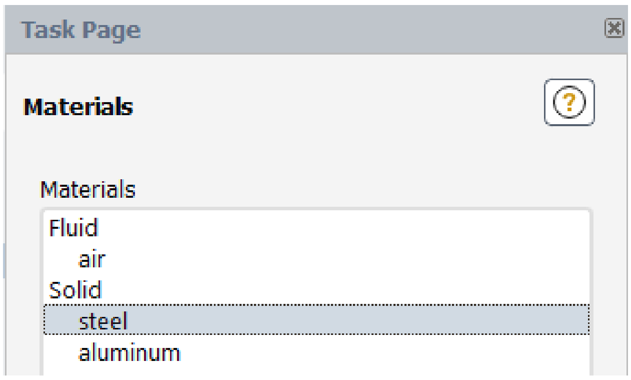


*Fig. 3.39 **"Steel"** material in the "**Materials**"*

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).

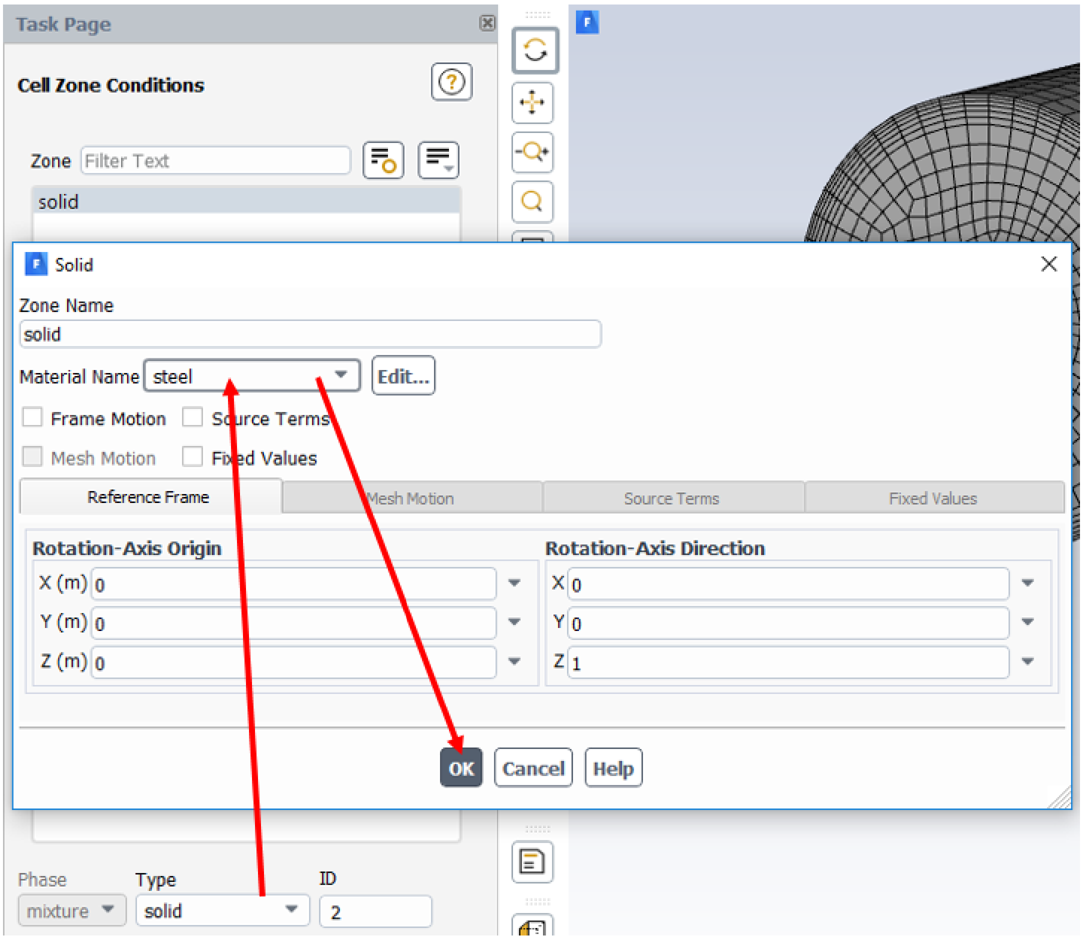


Fig. 3.40 - Characteristics of the Cell Zone Conditions command

Defining boundary conditions

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

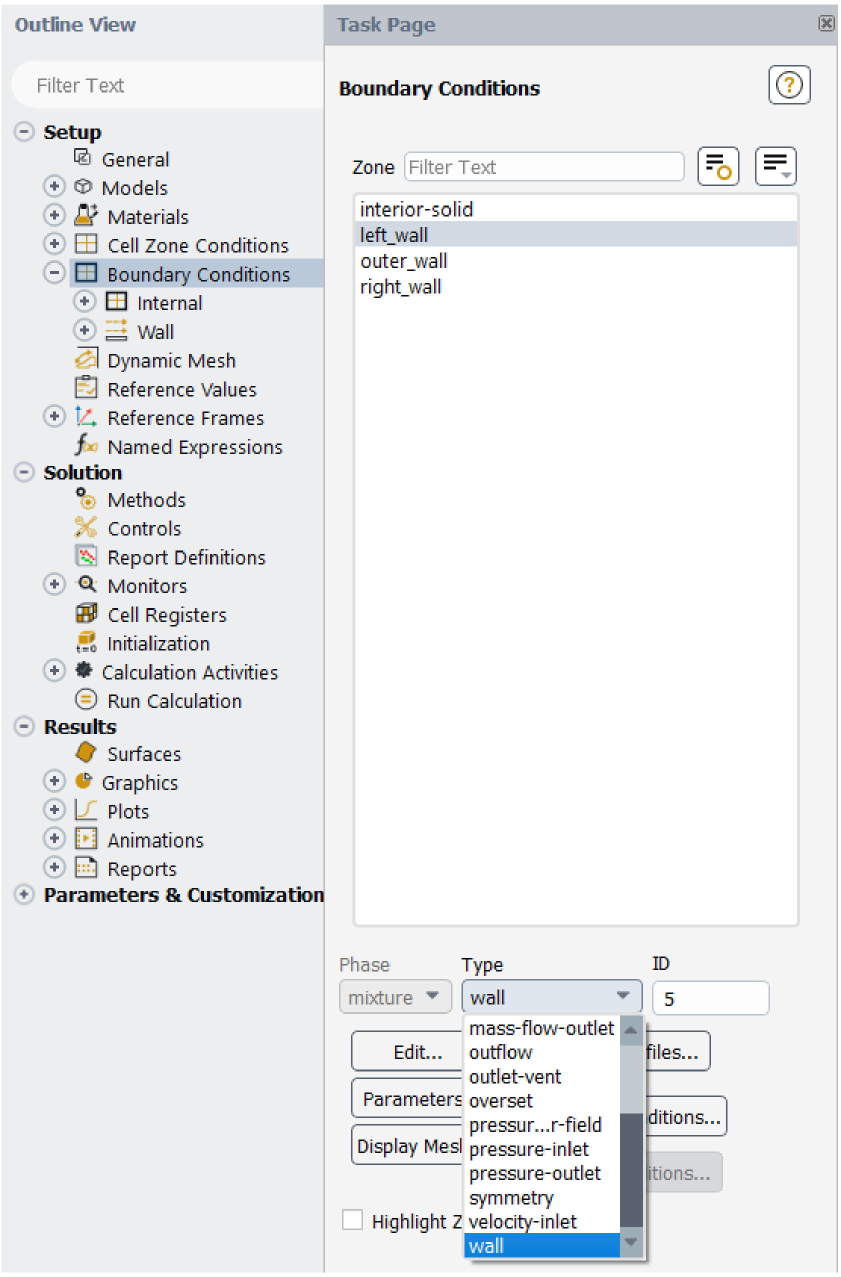


*Fig. 3.41– **Boundary Conditions** and types of boundary conditions*

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“** ($T_0 = 50\ °C = 323.15 K$), see Figure 3.42.

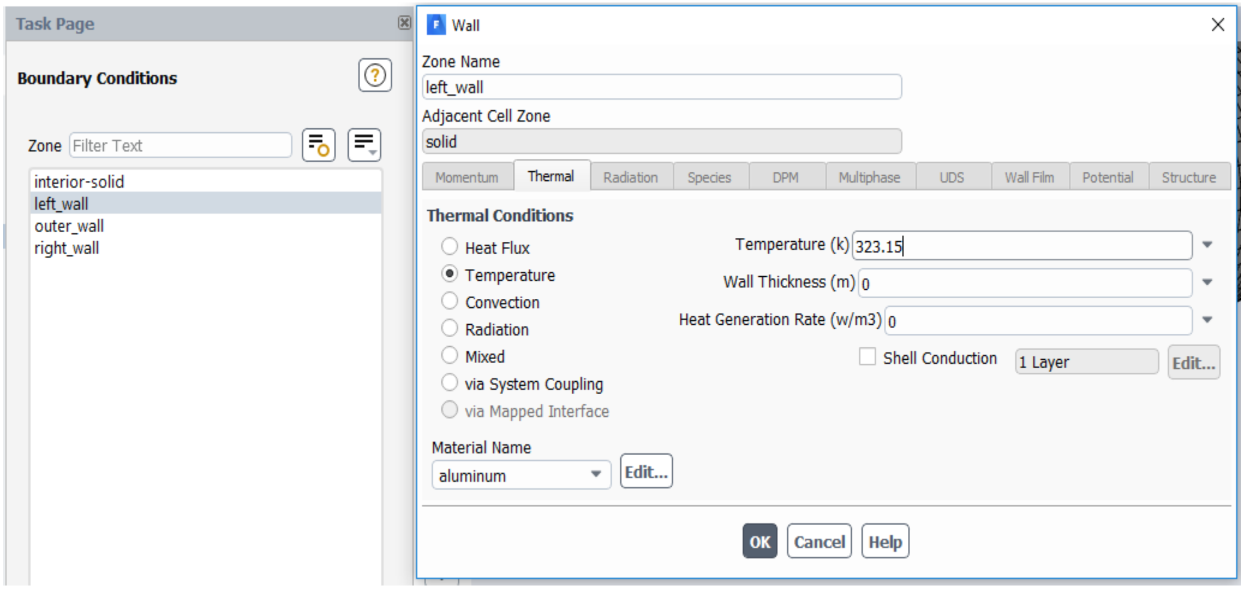


*Fig. 3.42 - Defining boundary condition „**left_wall**“*

- **right_wall** – type **„wall“ - „Edit“** $T_l = -10°C = 263.15K$, see Figure 3.43.

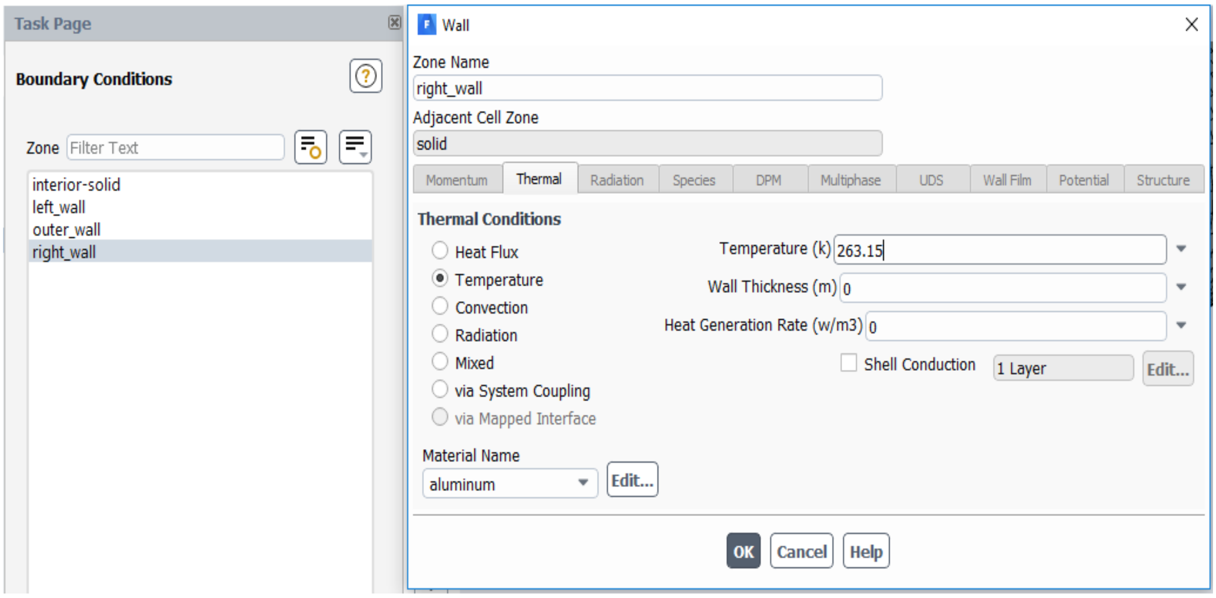
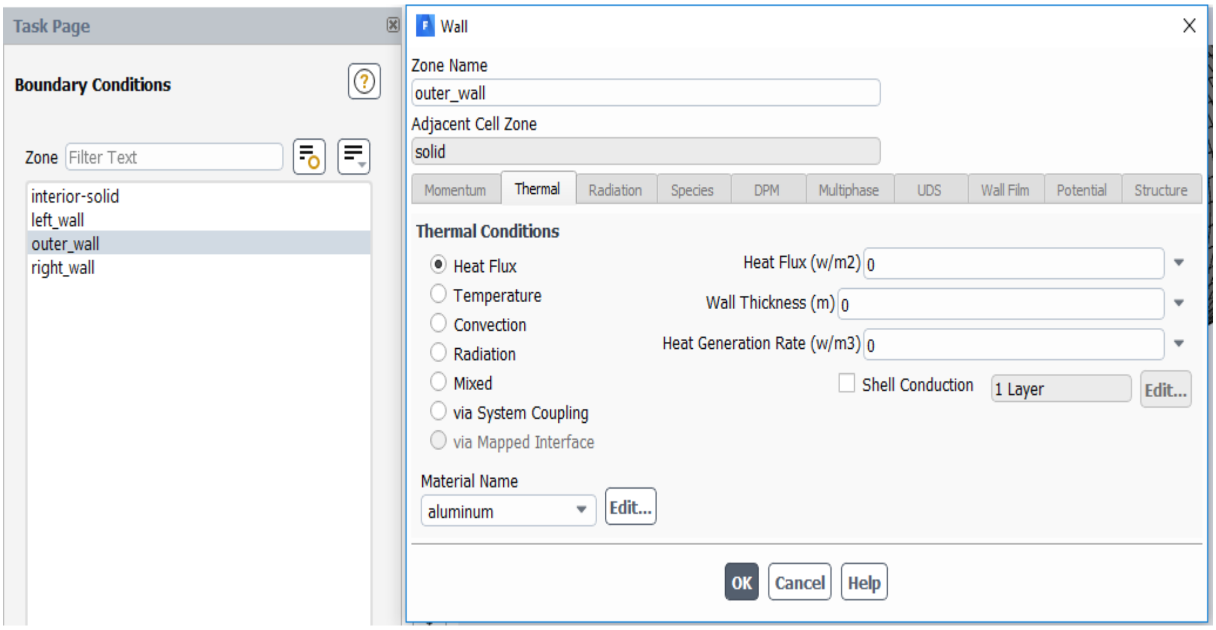


*Fig. 3.43 - Defining the boundary condition **"right_wall"***

- **outer_wall** – type **„wall“ - „Edit“** $q = 0\ W/m^2$, see Figure 3.44.



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).

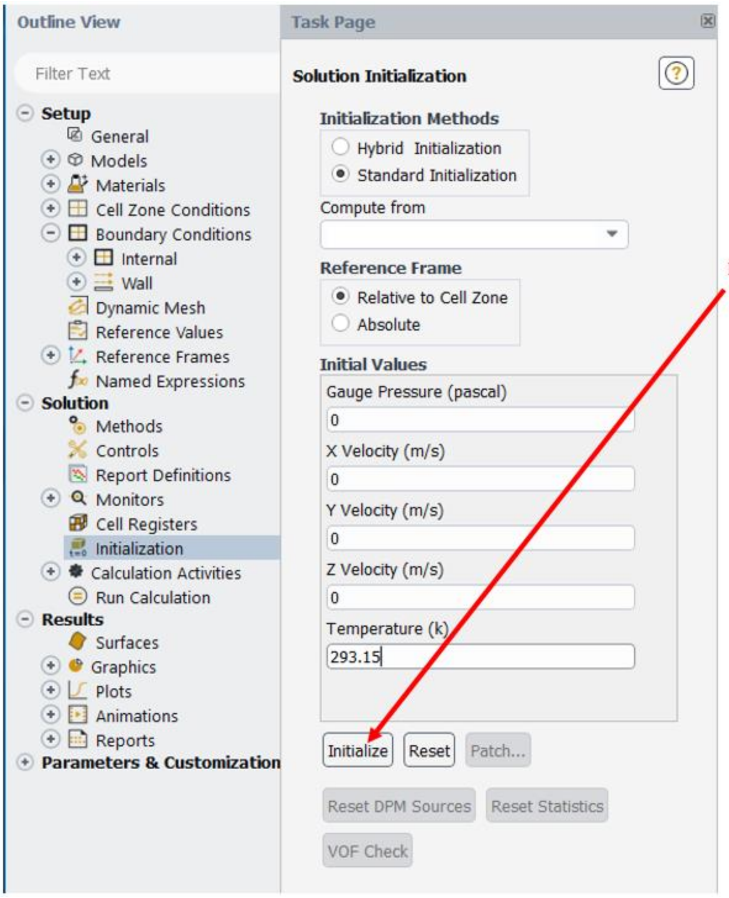


Fig. 3.45 - Initialization of the computation area

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.

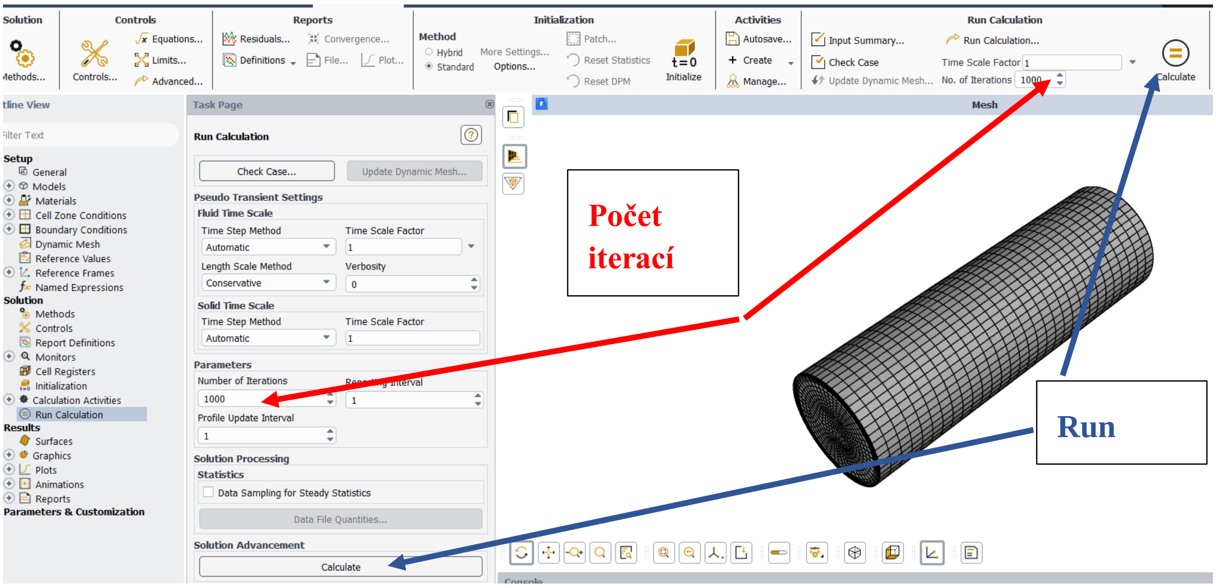


*Figure 3.46 - Run „**Canculation command**"*

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

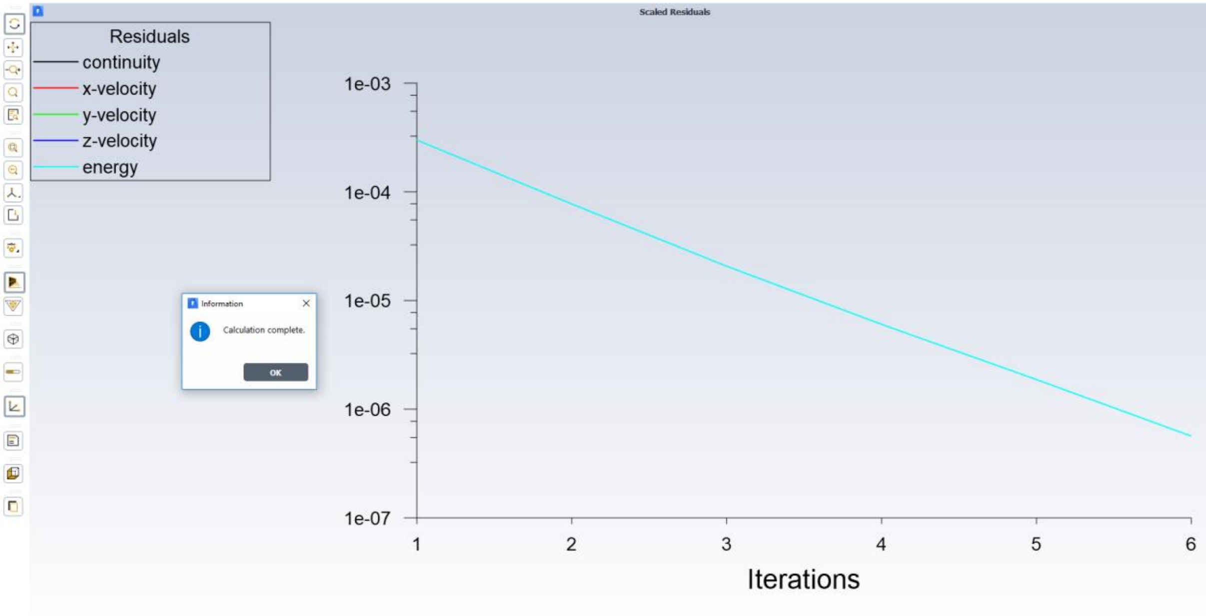
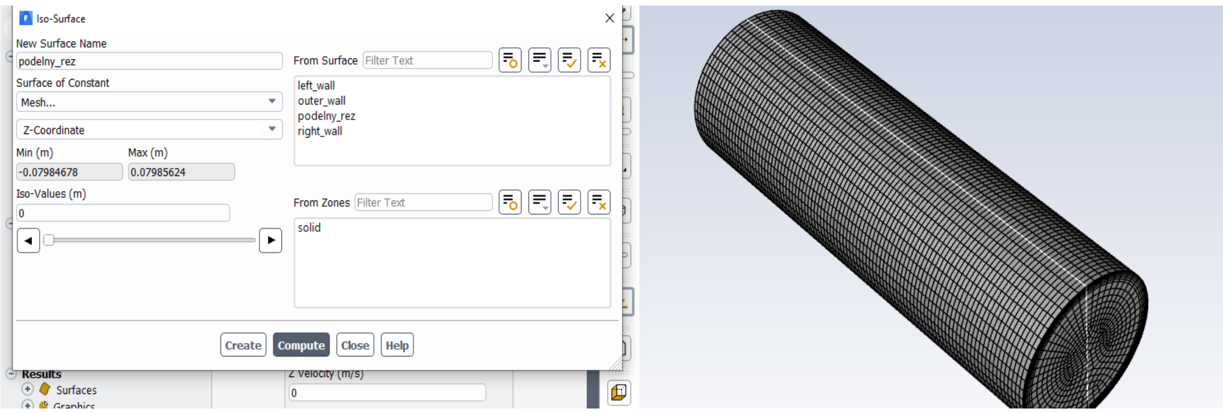


Fig. 3.47 - Progression of convergence

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.

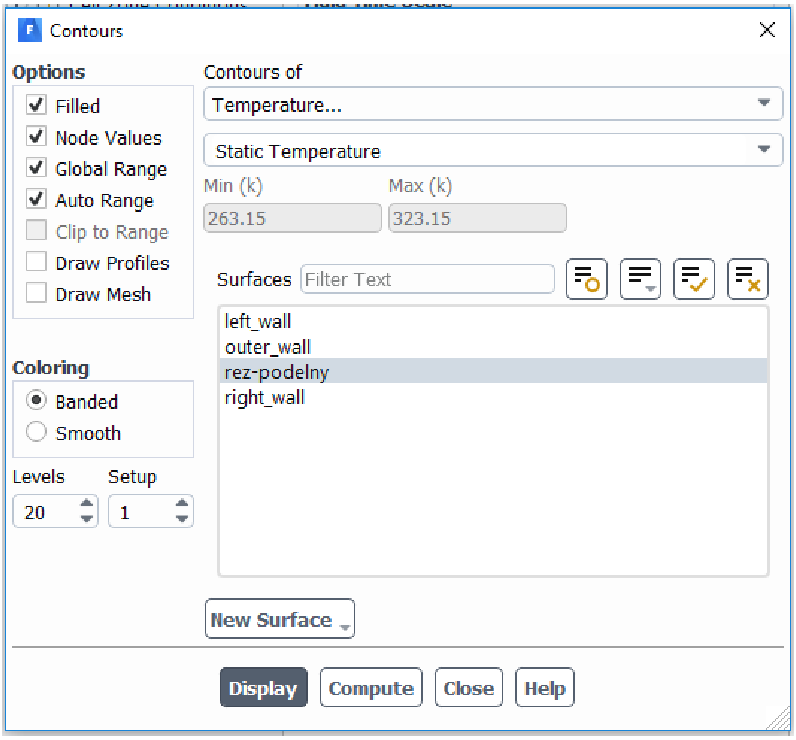


*Fig. 3.49 - Setting the drawing of temperature isolines by command „**Contours**“*

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)).

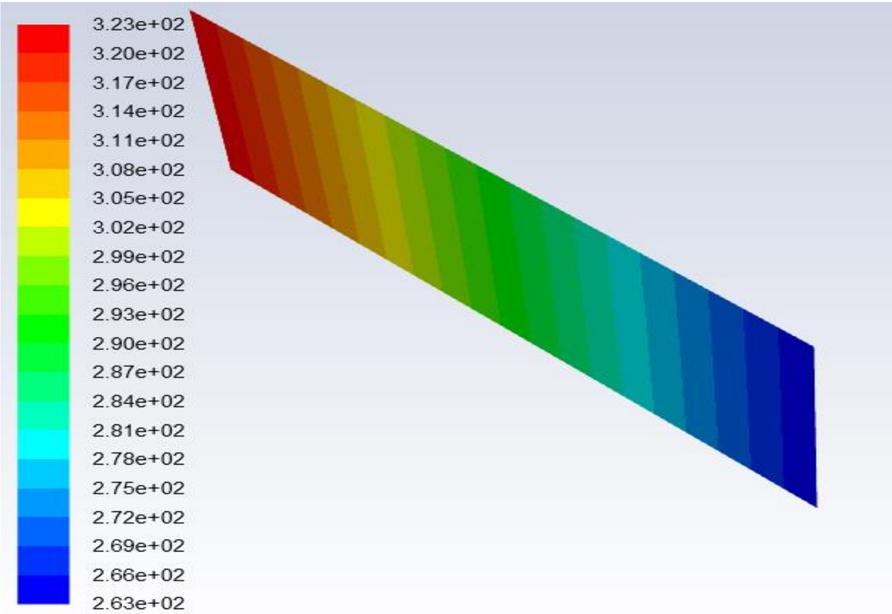


Fig. 3.50 - Temperature distribution in the whole area [K]

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.

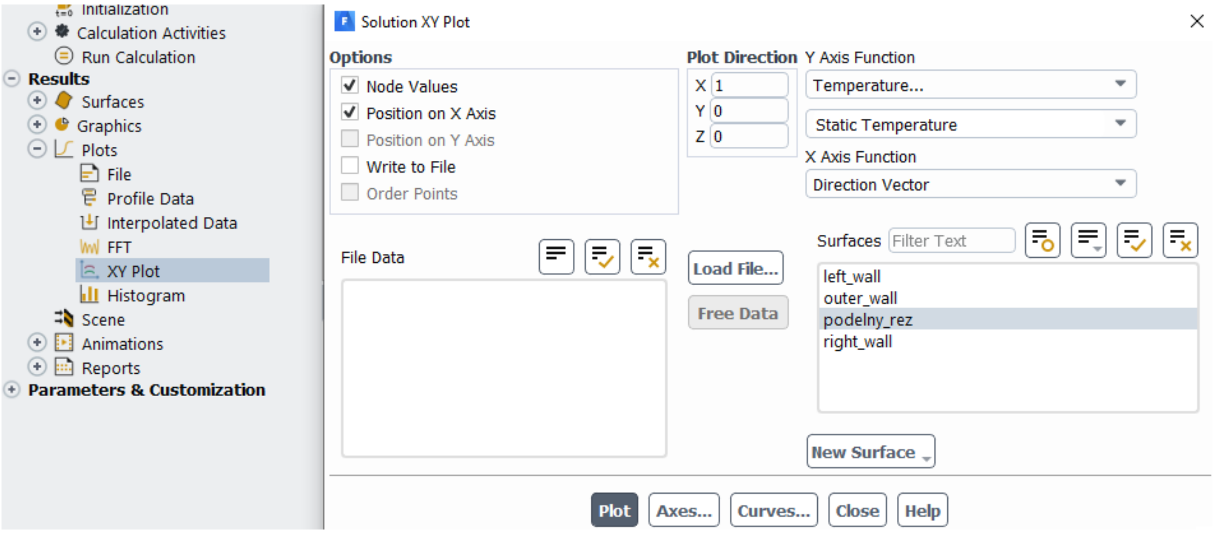


*Fig. 3.51 - Characteristics of „**XY Plot command**"*

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

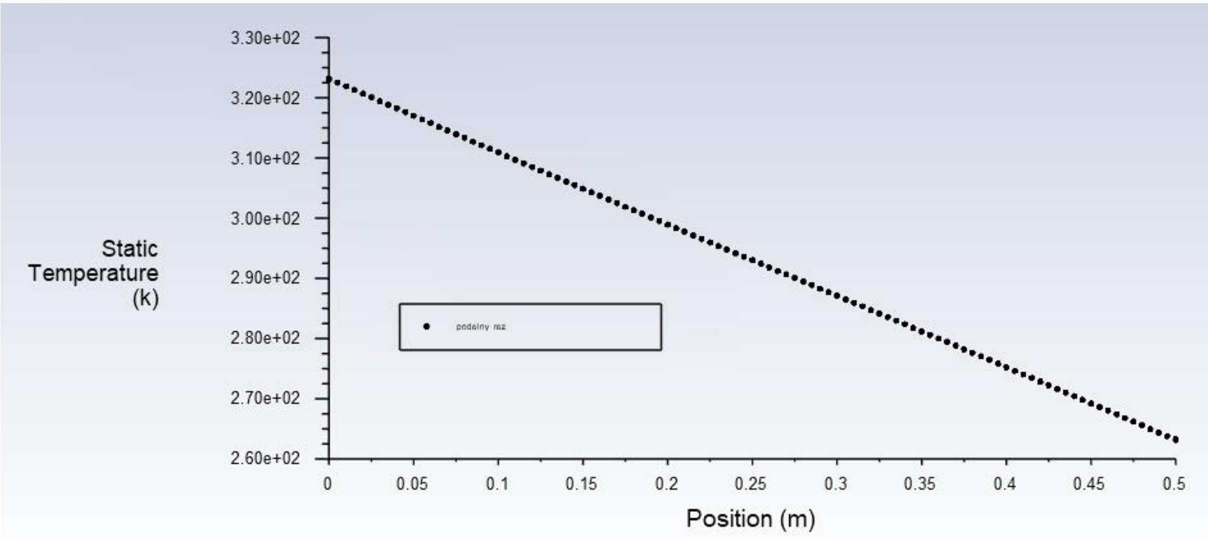


Fig. 3.52 - Temperature distribution along the length of the areas

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.

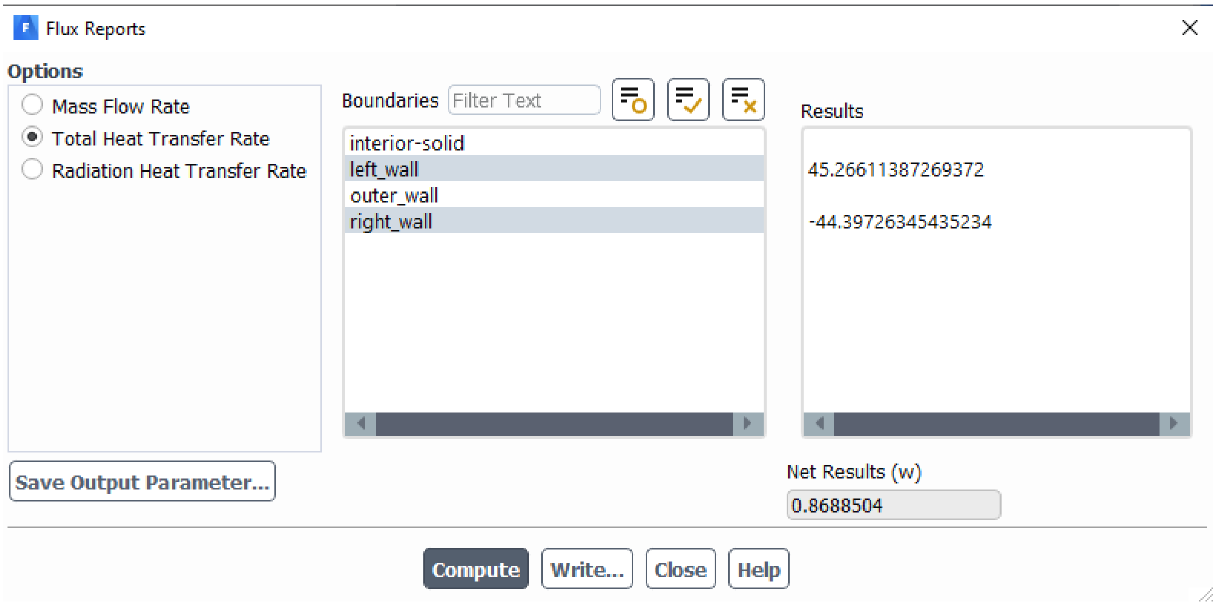


*Fig. 3.53 - “**Fluxes**“*

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“

.

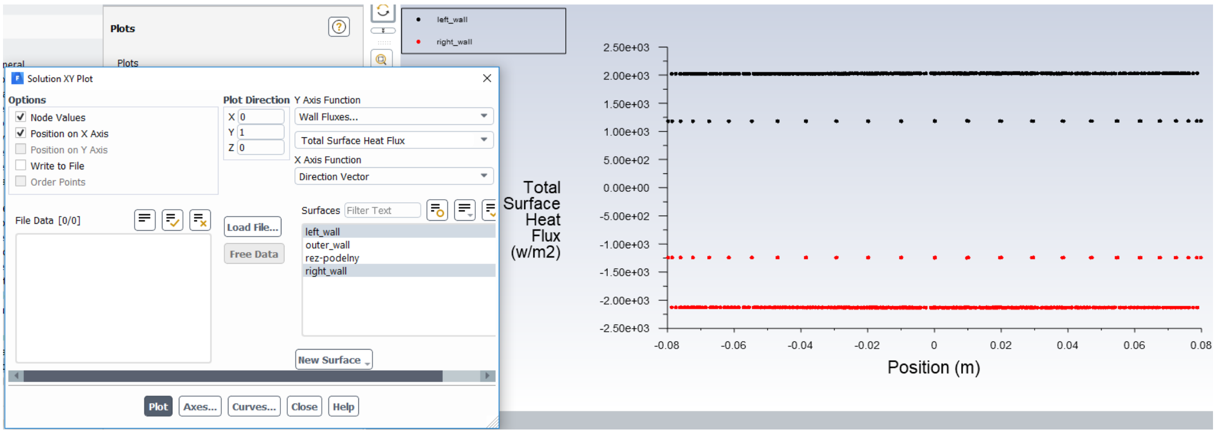


*Fig. 3.54 - Distribution of heat flow through walls „**left wall** and **right wall**“*

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 ρ **[kg·m^{-3}]**	700	8030	2719	8978
specific heat capacity c_p **[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.

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

	OKRAJOVÉ PODMÍNKY				
Varianta	**left wall**	**right wall**	**right wall**	**right wall**	
	$T_0\,[°C]$	$T_l\,[°C]$	$q_l[W.m^{-2}]$	$\alpha[W.m^{-2}.K^{-1}]$	$T_\infty[°C]$
A	50	-10			
B	-20	100			
C	50		162700		
D	50		0		
E	50			1000	-10

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“**

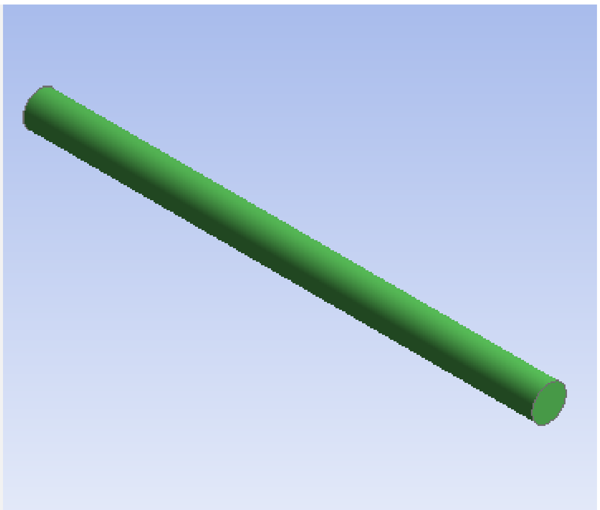
- Prepare a solution area consisting of three rods of different diameters, when defining the geometry, always use the movement of the coordinate system to the end of the rod (“**Create / New Plane**”). The boundary conditions are the same. When sweeping, use the sweep method on the first and third rods, the second rod relative to a different diameter is transmitted only by inflation

Geometry creation:

Sketching | Modeling

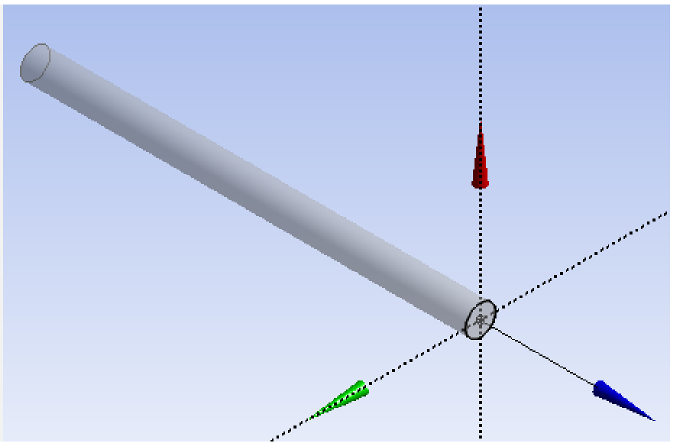
Details View

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

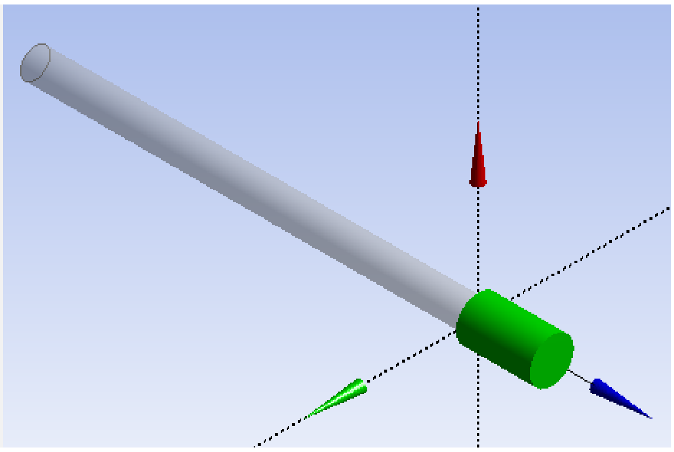


Details View

Details of EndPipe22	
Plane	EndPipe22
Sketches	0
Type	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

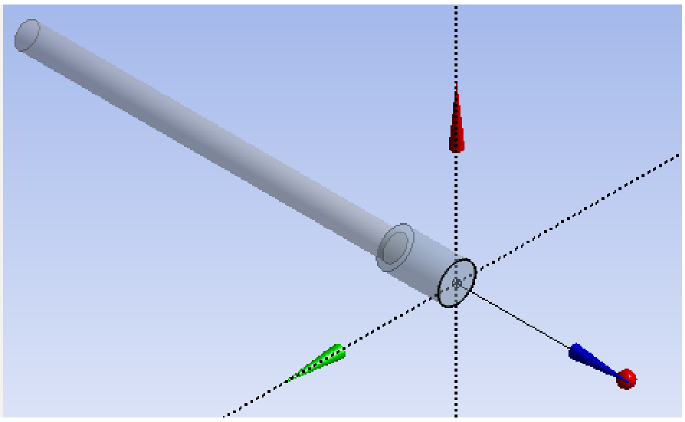


Details View	
Details of Pipe32	
Cylinder	Pipe32
Base Plane	EndPipe22
Operation	Add Frozen
Origin Definition	Coordinates
FD3, Origin X Coordinate	0 m
FD4, Origin Y Coordinate	0 m
FD5, Origin Z Coordinate	0 m
Axis Definition	Components
FD6, Axis X Component	0 m
FD7, Axis Y Component	0 m
FD8, Axis Z Component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



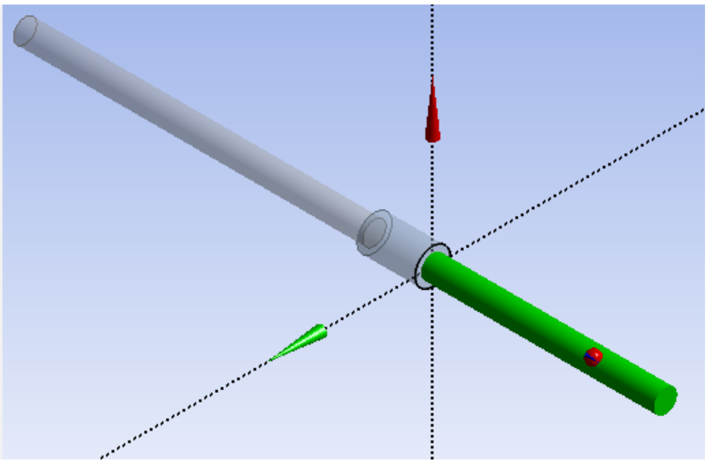
Sketching Modeling

Details View	
Details of EndPipe32	
Plane	EndPipe32
Sketches	0
Type	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

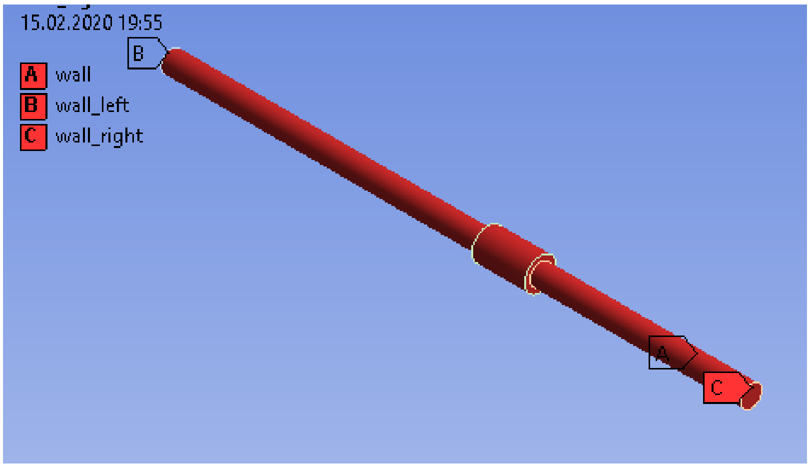


Sketching Modeling

Details View	
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

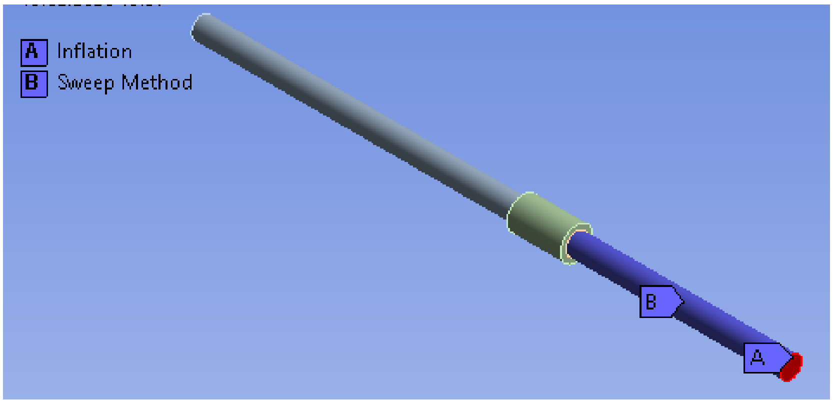


Boundary conditions:

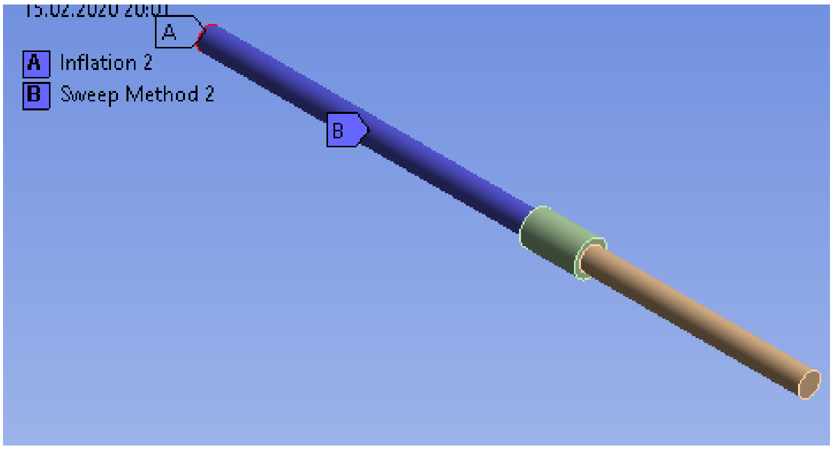


Meshing:

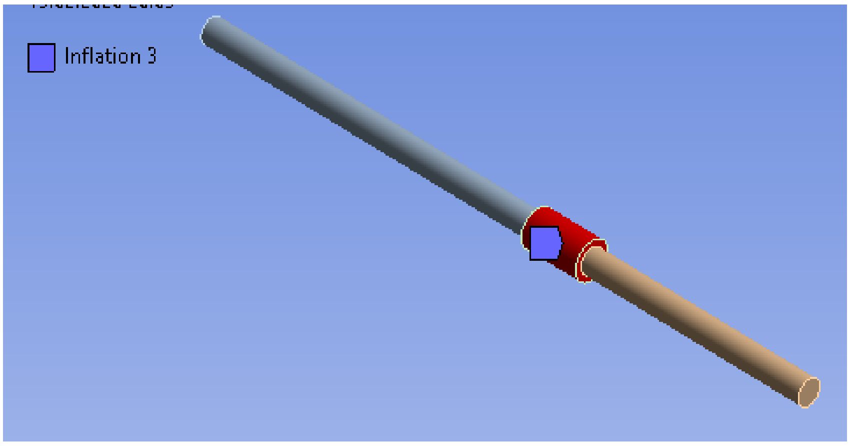
Inflation to the **right_wall**, the border is a circle, sweep is applied only to the third tube (the next tube changes the diameter, so it is not possible to continue with the sweep)



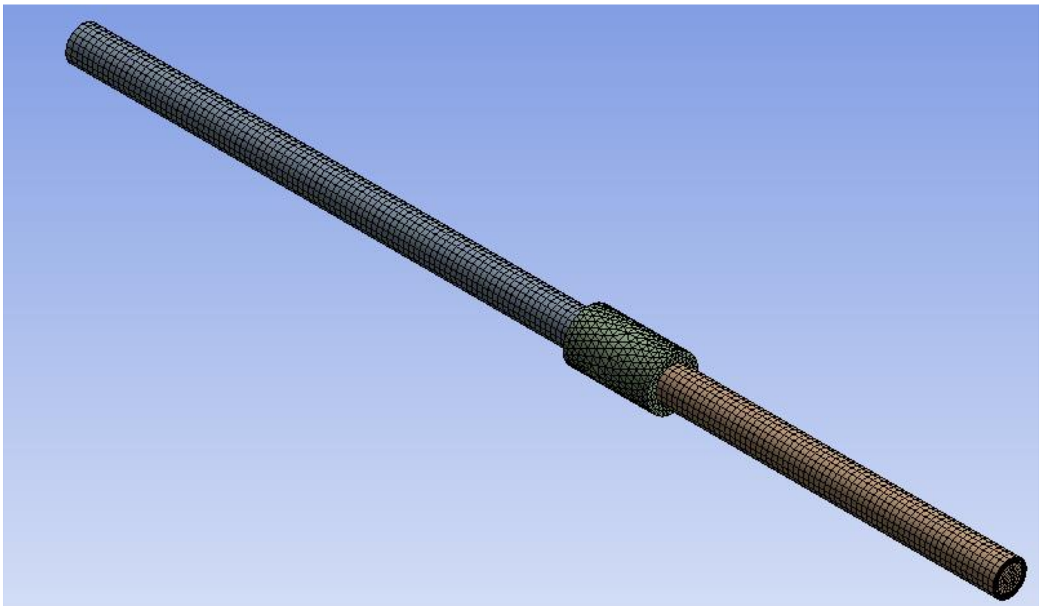
Inflation on the left_wall, the border is a circle, the sweep is applied only to the first pipe

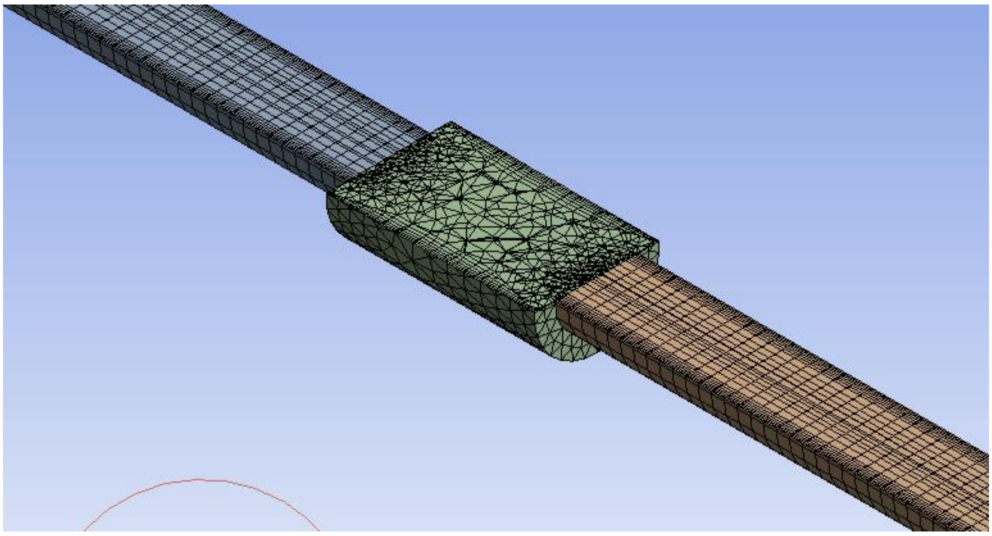


Inflation on the volume of the second tube, the boundary is the area of the tube



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.

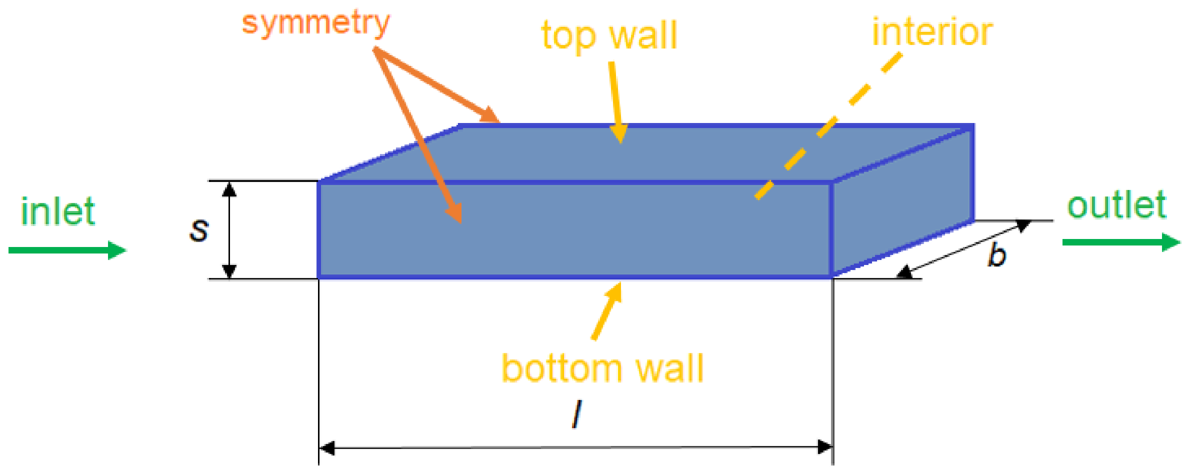


Fig. 4.1 - Area diagram

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.

Tab. 4.1 – Geometry area

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

Tab. 4.2 - Physical properties of water

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.

Tab. 4.2 – Boundary conditions

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).

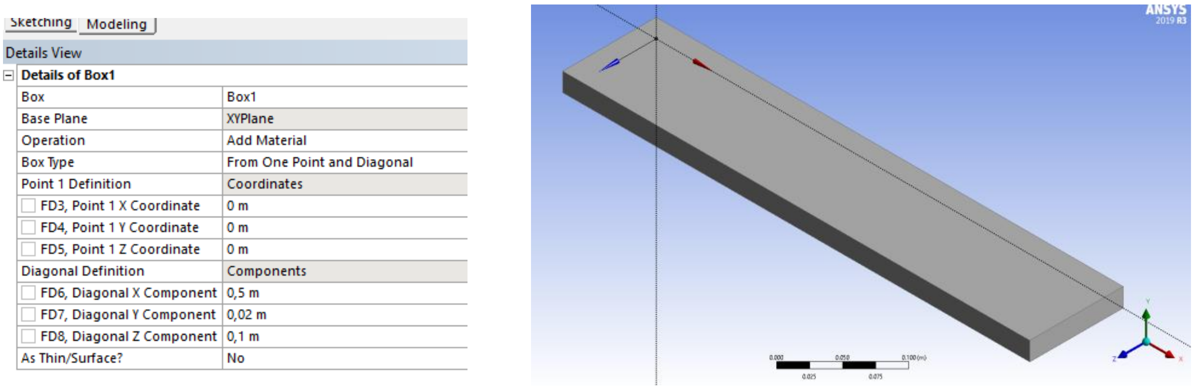
The laminarity criterion is Reynolds number:

$$\mathrm{Re} = \frac{u.d}{\nu} = \frac{0.05\,.\,0.02}{1.10^{-6}} = 1000$$

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.

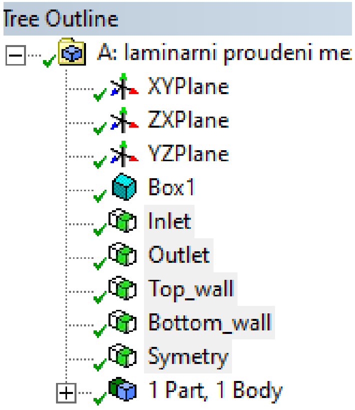


Fig. 4.3 - Marking of boundary conditions

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.

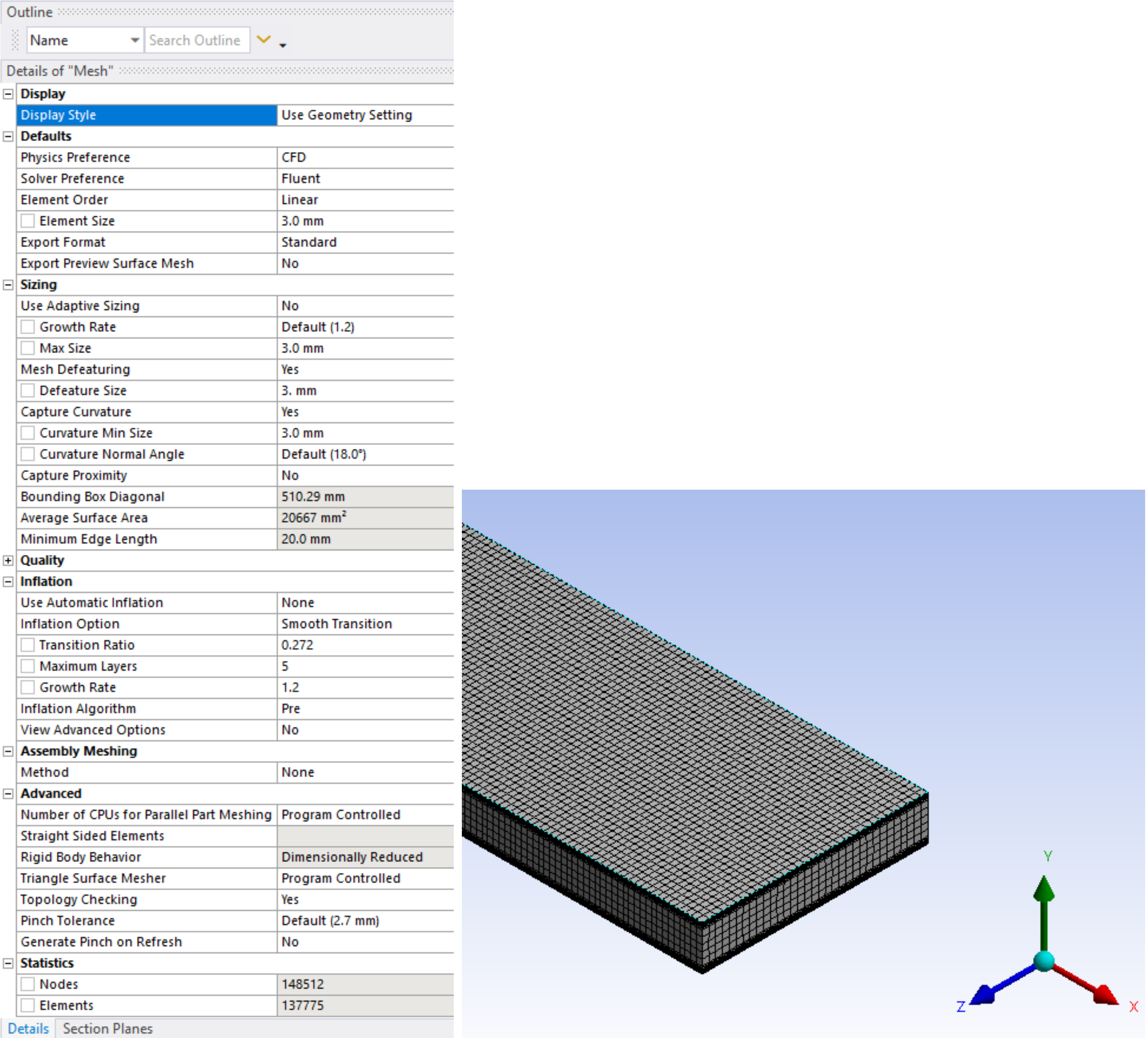
Details of "Inflation" - Inflation

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	0.2
Maximum Layers	10
Growth Rate	1.2
Inflation Algorithm	Pre

Details of "Sweep Method" - Method

Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Method	Sweep
Algorithm	Program Controlled
Element Order	Use Global Setting
Src/Trg Selection	Manual Source
Source	1 Face
Target	Program Controlled
Free Face Mesh Type	All Quad
Type	Number of Divisions
Sweep Num Divs	200
Element Option	Solid
Advanced	
Sweep Bias Type	No Bias

*Fig. 4.4 - **Inflation** parameters and **Sweep** method*



Display	
Display Style	Use Geometry Setting
Defaults	
Physics Preference	CFD
Solver Preference	Fluent
Element Order	Linear
Element Size	3.0 mm
Export Format	Standard
Export Preview Surface Mesh	No
Sizing	
Use Adaptive Sizing	No
Growth Rate	Default (1.2)
Max Size	3.0 mm
Mesh Defeaturing	Yes
Defeature Size	3. mm
Capture Curvature	Yes
Curvature Min Size	3.0 mm
Curvature Normal Angle	Default (18.0°)
Capture Proximity	No
Bounding Box Diagonal	510.29 mm
Average Surface Area	20667 mm²
Minimum Edge Length	20.0 mm
Quality	
Inflation	
Use Automatic Inflation	None
Inflation Option	Smooth Transition
Transition Ratio	0.272
Maximum Layers	5
Growth Rate	1.2
Inflation Algorithm	Pre
View Advanced Options	No
Assembly Meshing	
Method	None
Advanced	
Number of CPUs for Parallel Part Meshing	Program Controlled
Straight Sided Elements	
Rigid Body Behavior	Dimensionally Reduced
Triangle Surface Mesher	Program Controlled
Topology Checking	Yes
Pinch Tolerance	Default (2.7 mm)
Generate Pinch on Refresh	No
Statistics	
Nodes	148512
Elements	137775

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**".

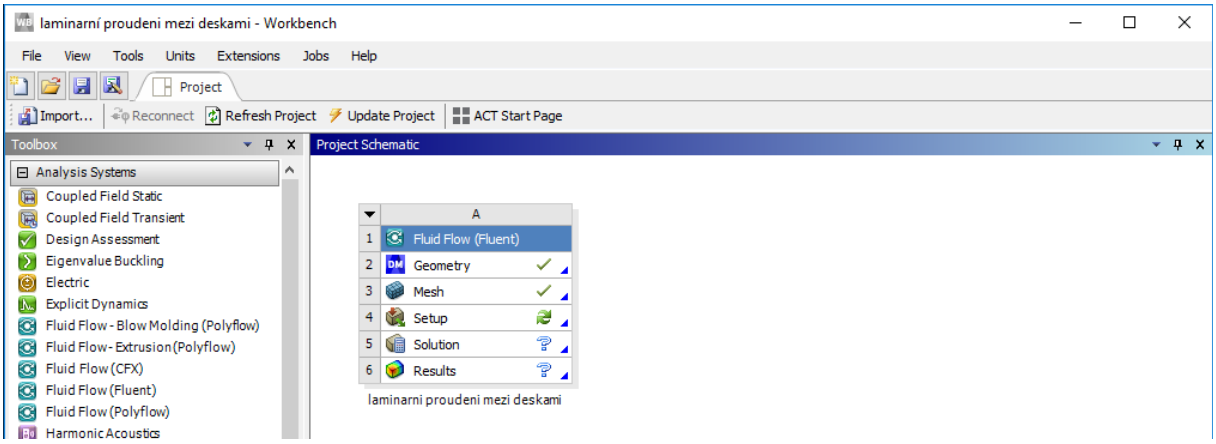


*Fig. 4.6 - **ANSYS Workbench** after update*

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.

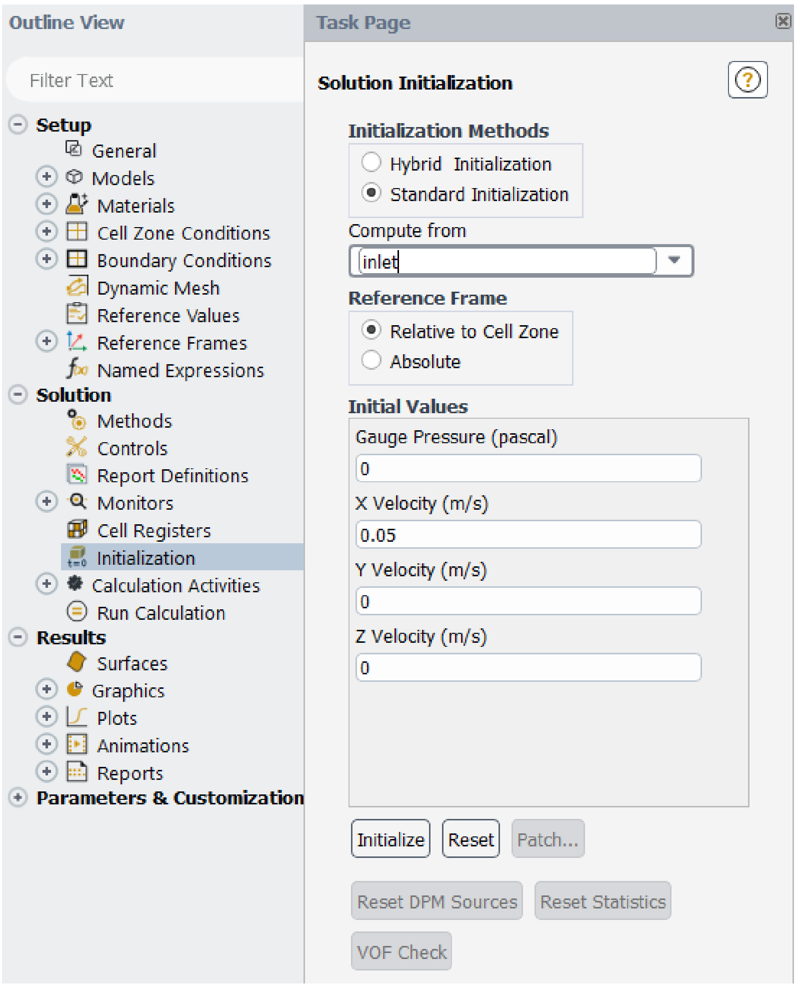


Fig. 4.7 - Initialization based on input boundary condition

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.

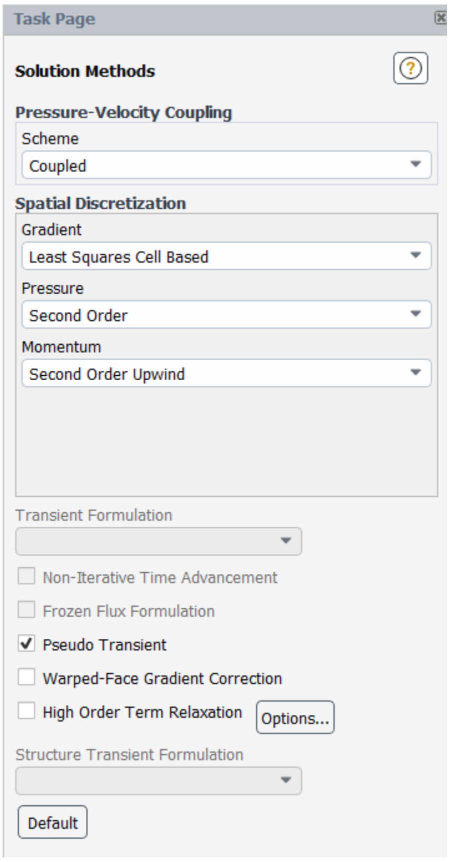


Fig. 4.8 - Setting of stability schemes

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.

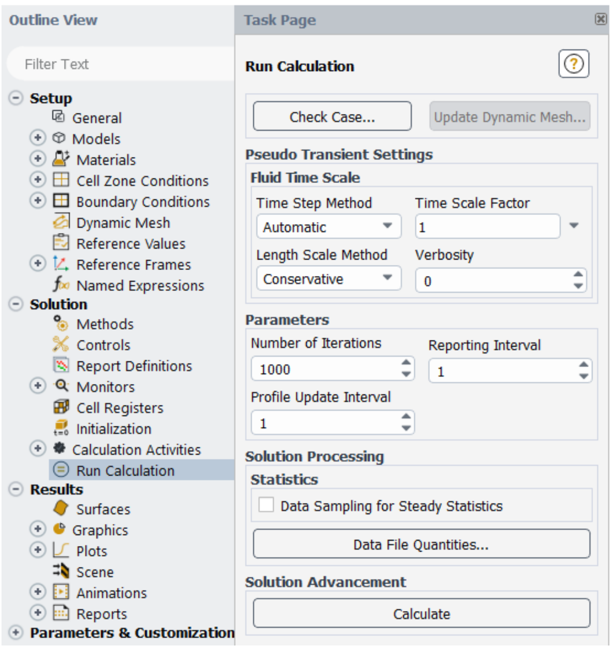


Fig. 4.9 - Setting the number of iterations and starting the calculation

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.

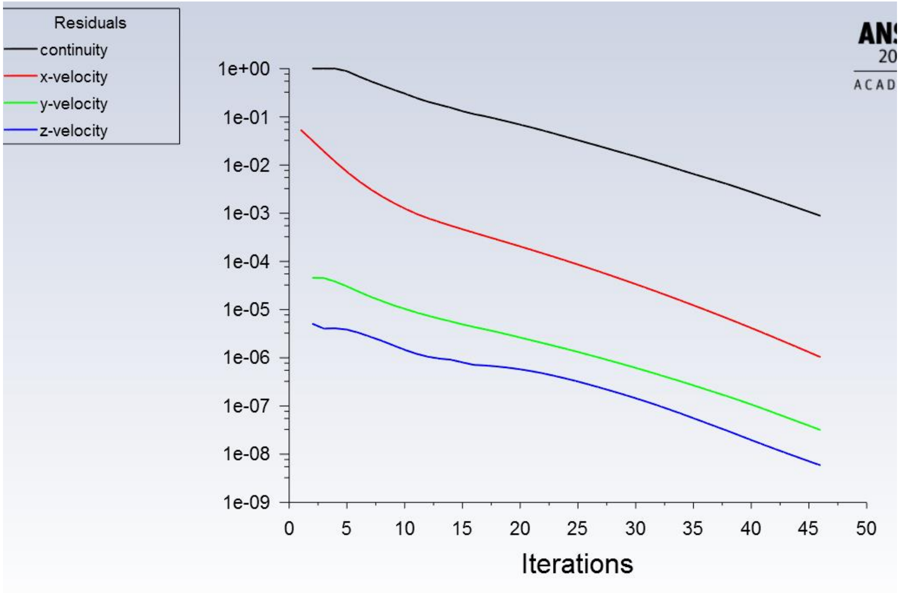


Fig. 4.10 - The course of residuals

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).

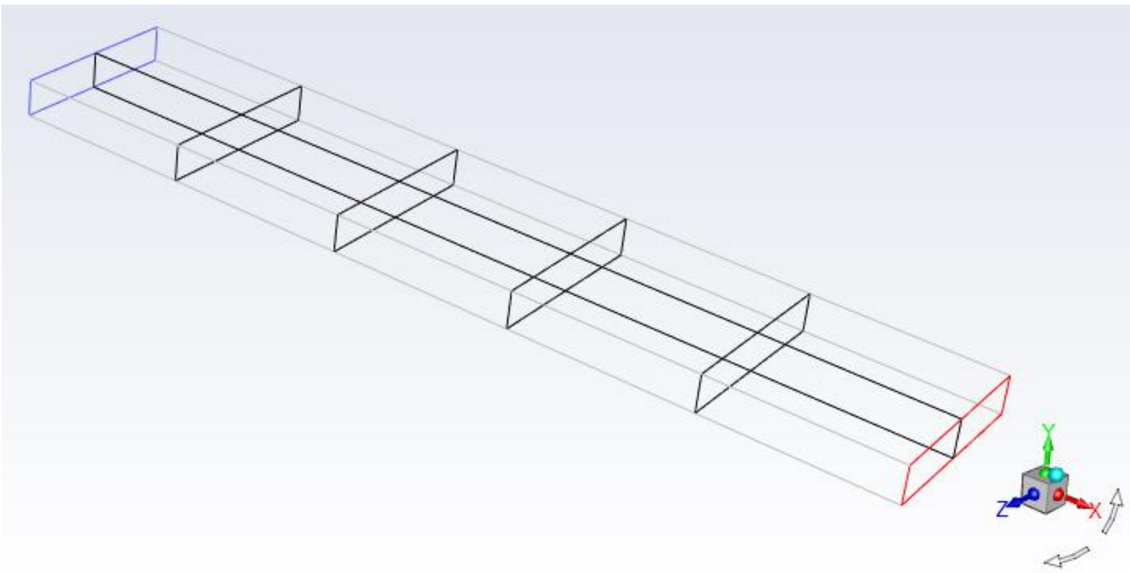


Fig. 4.11 - Created planes for evaluation

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**.

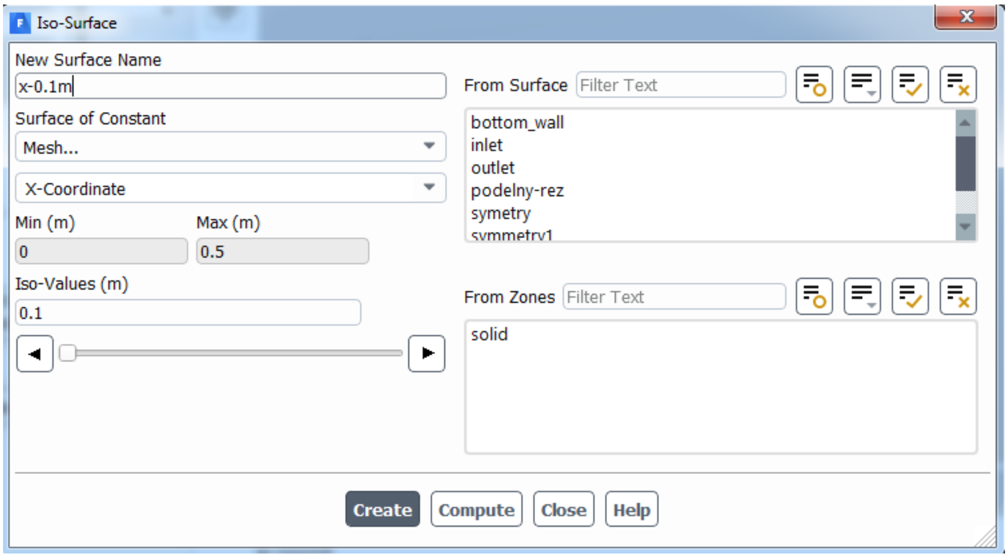


Fig. 4.12 - Creating an auxiliary plane at a distance x = 0.1 m

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.

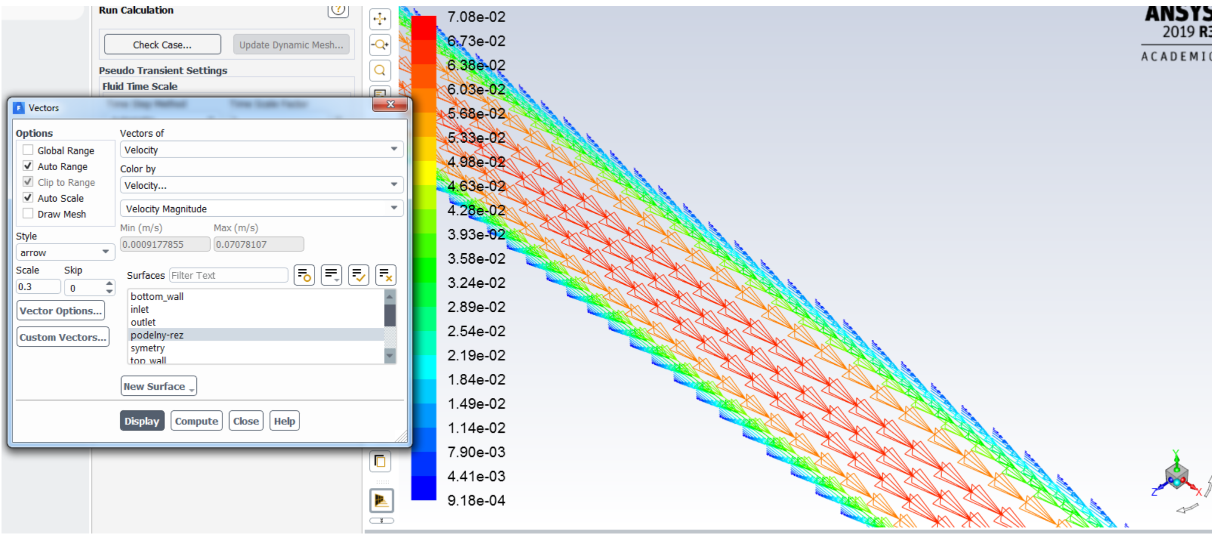


Fig. 4.13 - Speed vectors for Scale = 0.3 and Skip =0 ($u[m.s^{-1}]$)

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.

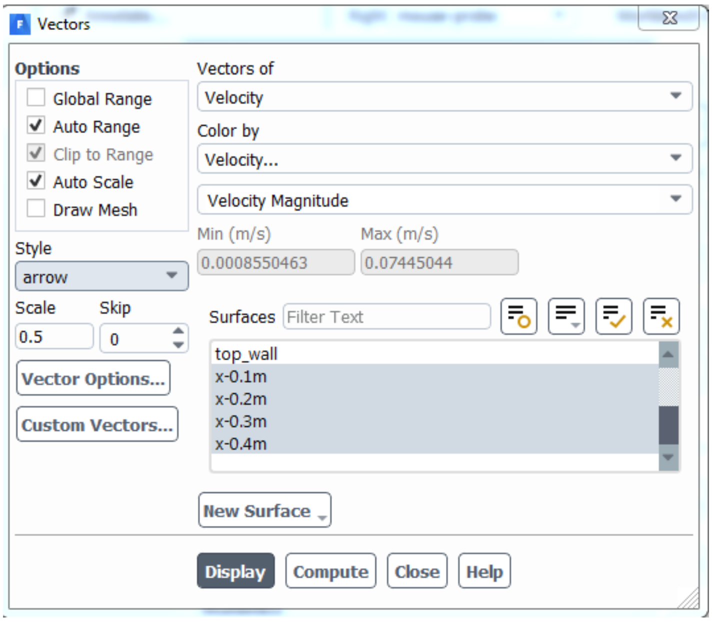


Fig. 4.14 - Definition of velocity vectors in cross-sections

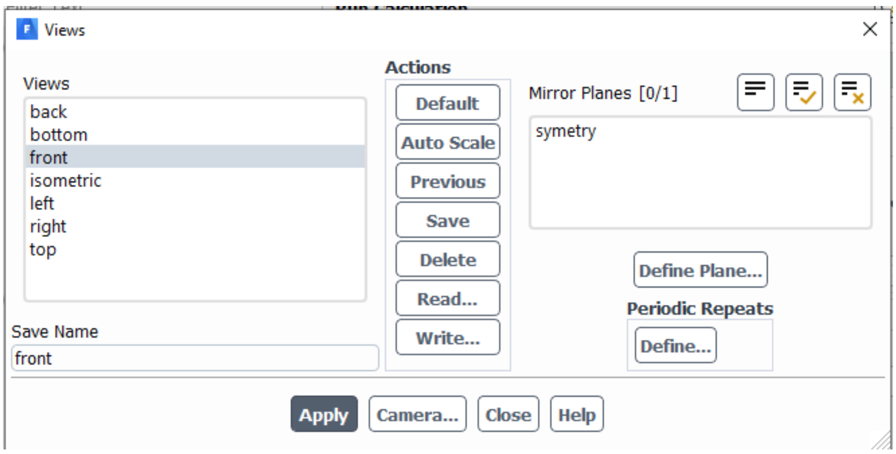


Fig. 4.15 - Menu for defining the front view.

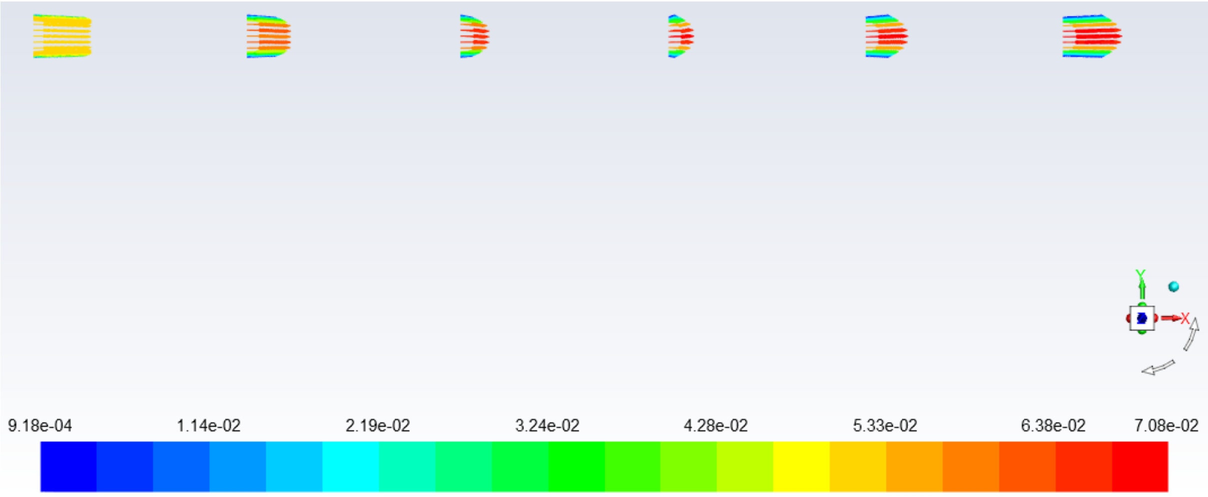


Fig. 4.16 - Speed vectors in individual sections ($u[m.s^{-1}]$)

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.

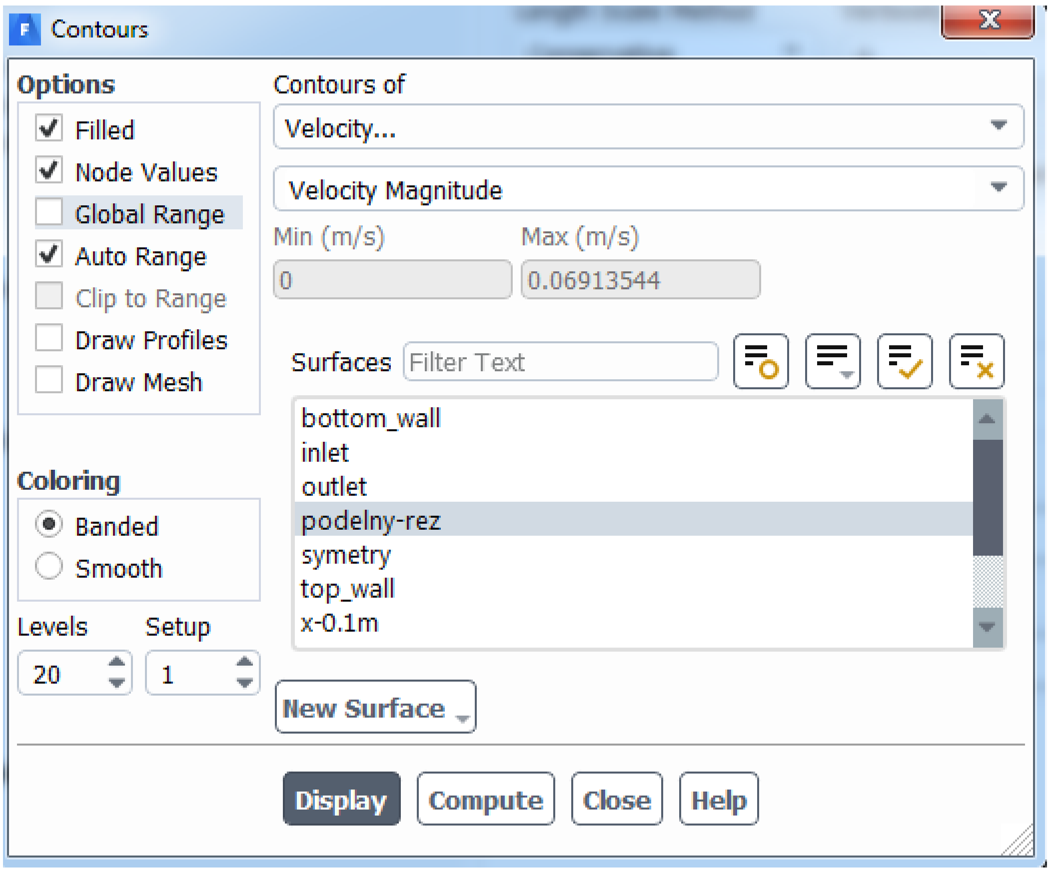


Fig. 4.17 - Menu for Creating Filled Isoplases of Velocity

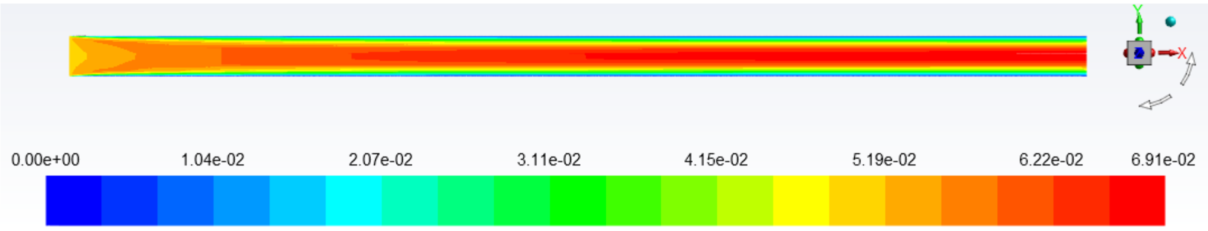


Fig. 4.18 - Contour velocity magnitude in the computational area ($u[m.s^{-1}]$)

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

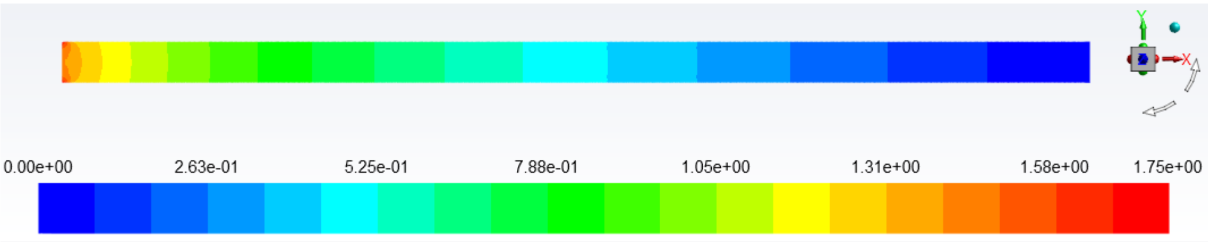


Fig. 4.19 - Filled isocurves of static pressure in the computational area p_{stat} *[Pa]*

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.

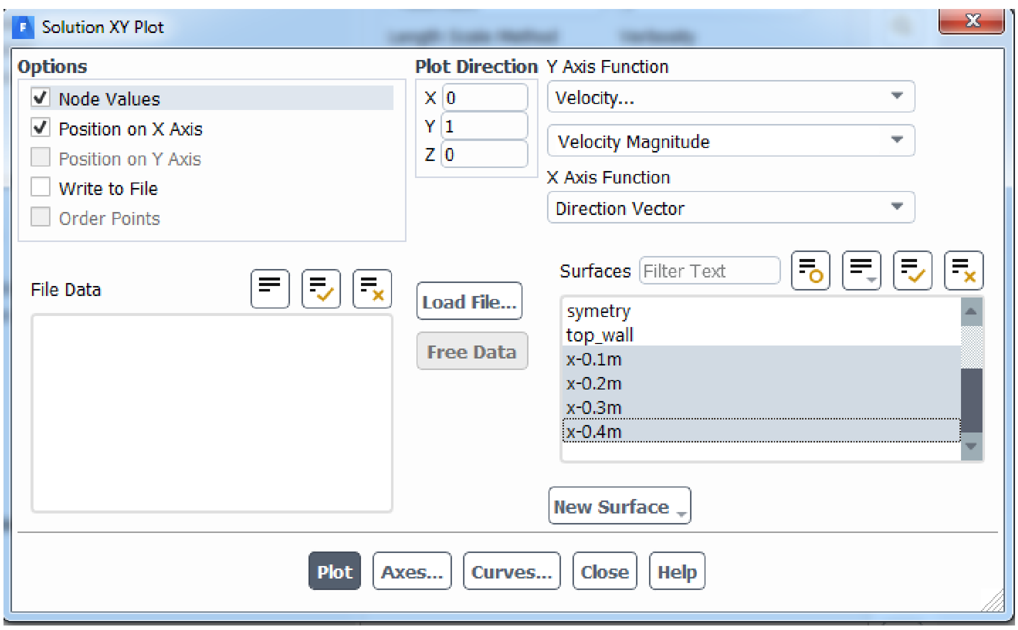


Fig. 4.20 - Menu for creating velocity profiles

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.

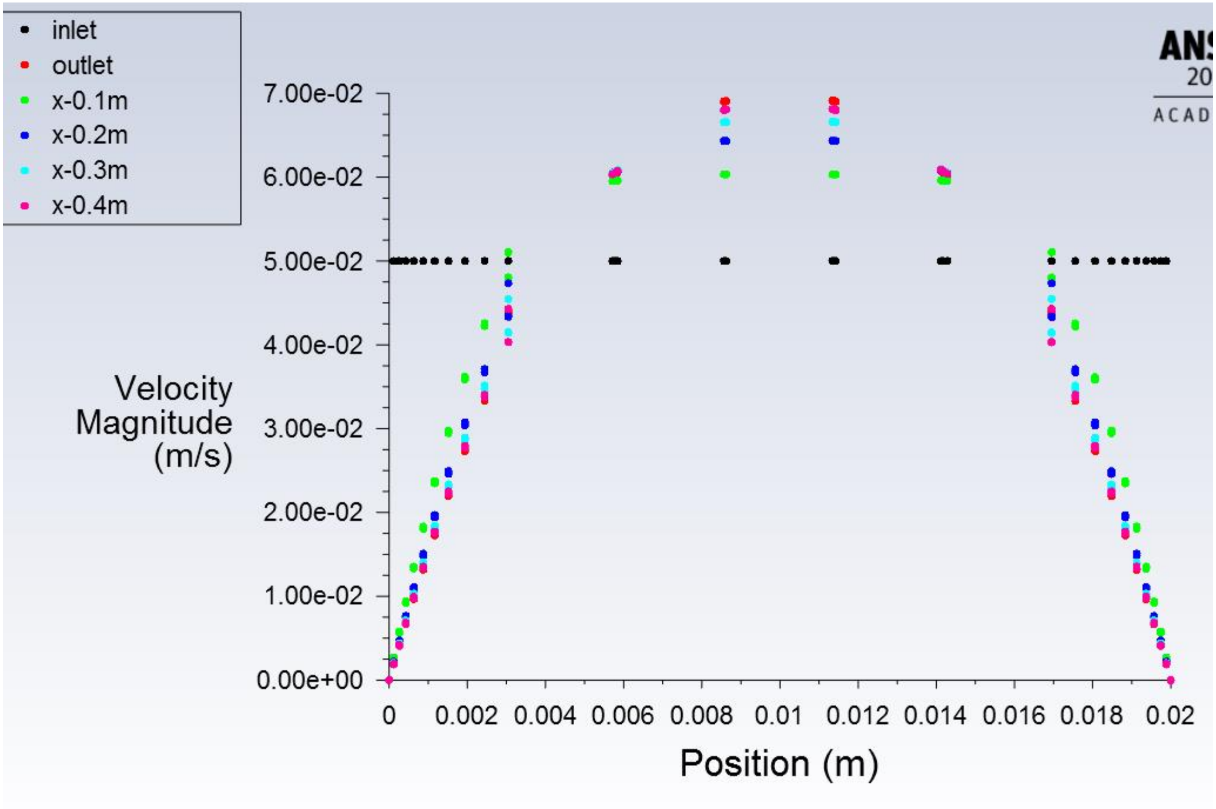


Fig. 4.21 - Formation of speed profile

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.

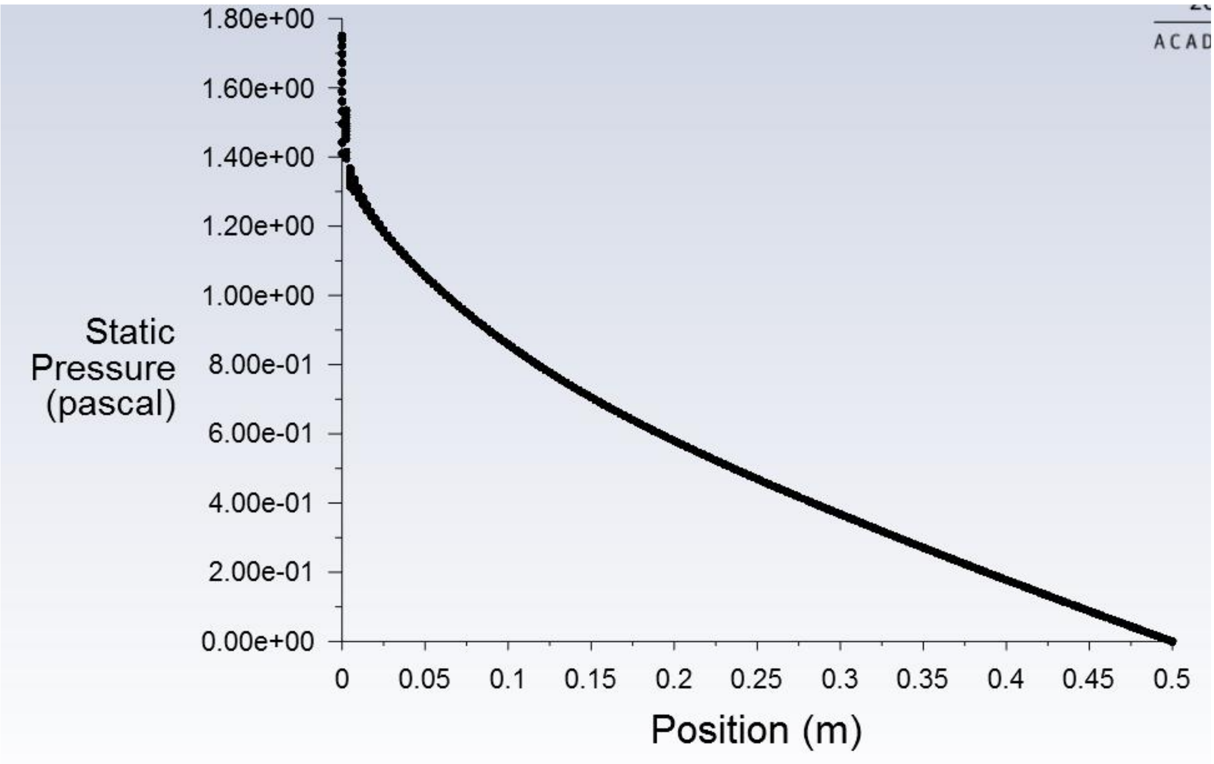


Fig. 4.22 - Static pressure profile along the length evaluated in the pipe axis (p_{stat} [Pa])

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.

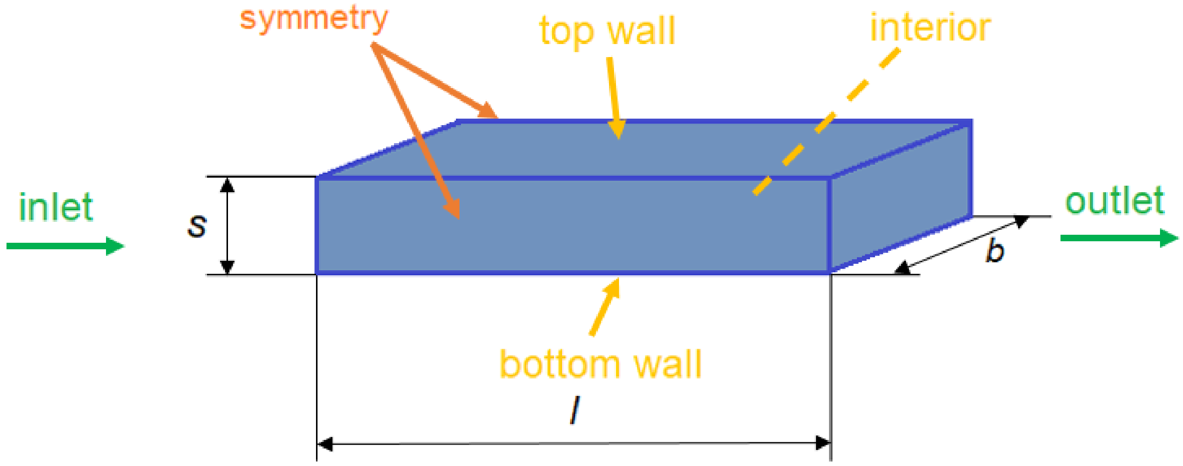


Fig. 5.1 - Area diagram

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.

Tab. 5.1 – Geometry area

area length l **[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
dynamic viscosity η **[kg.(m.s)$^{-1}$]**	0.001003

Boundary conditions

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.

Tab. 5.3 Boundary conditions

Inlet	Medium speed u_s [m.s^{-1}]	**1**
	Turbulent intensity [%]	**1**
	Hydraulic diameter [m]	**0.02**
Outlet	Static pressure p [Pa]	**0**
	Turbulent backflow intensity [%]	**1**
	Hydraulic diameter [m]	**0.02**

Mathematical model

The choice of mathematical model depends on Reynolds number.

The laminarity criterion is Reynolds number:

$$\mathrm{Re} = \frac{u.d}{\nu} = \frac{1.\,0.02}{1.10^{-6}} = 20000$$

The flow is therefore turbulent, but with a low Reynolds number, so **RNG k-ε** 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.

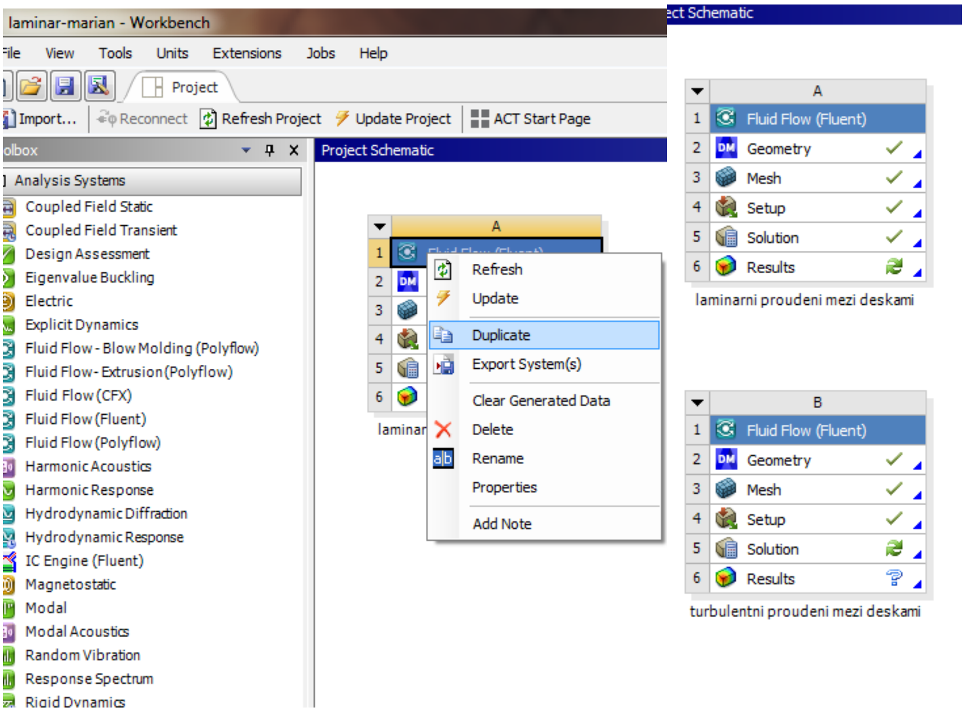


Fig. 5.2 - Copying of panel by command „Duplicate“

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.

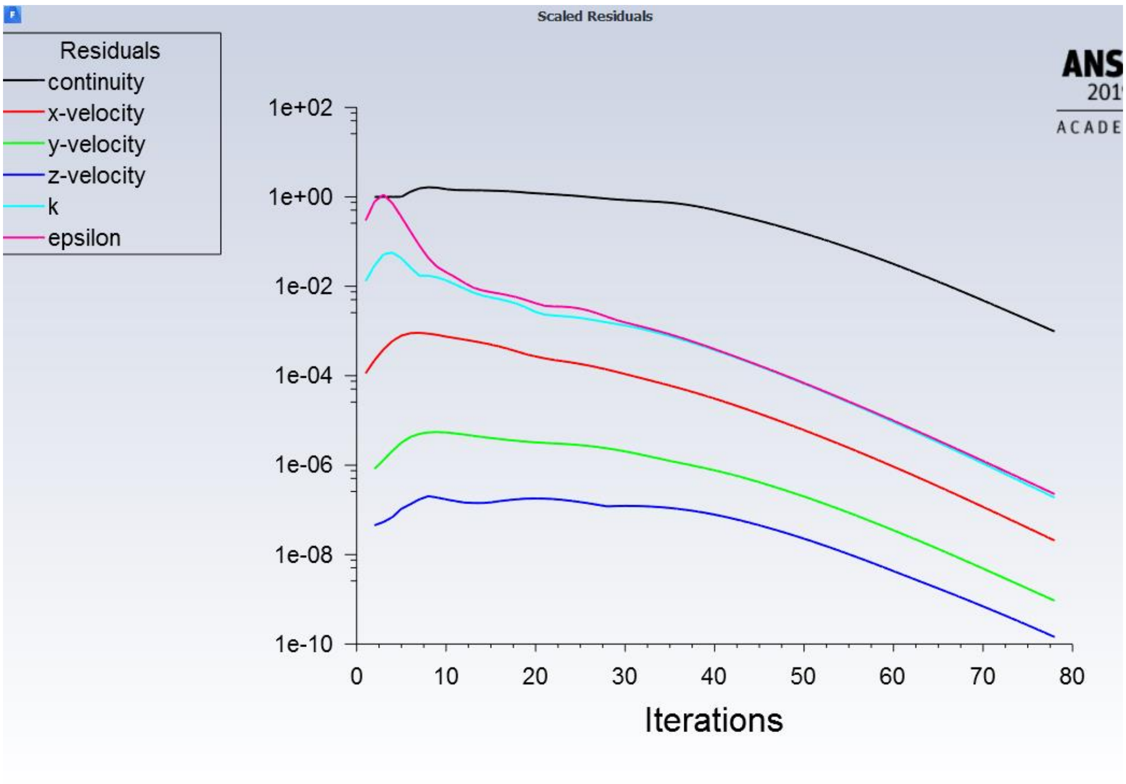


Fig. 5.3 - The course of residuals

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).

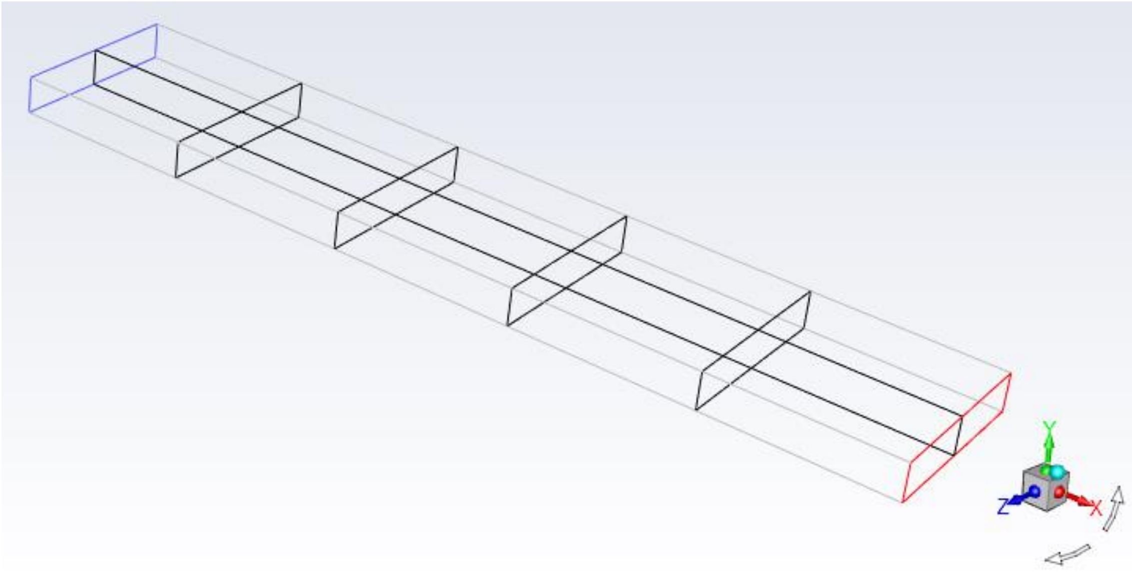


Fig. 5.4 - Created planes for evaluation

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**..

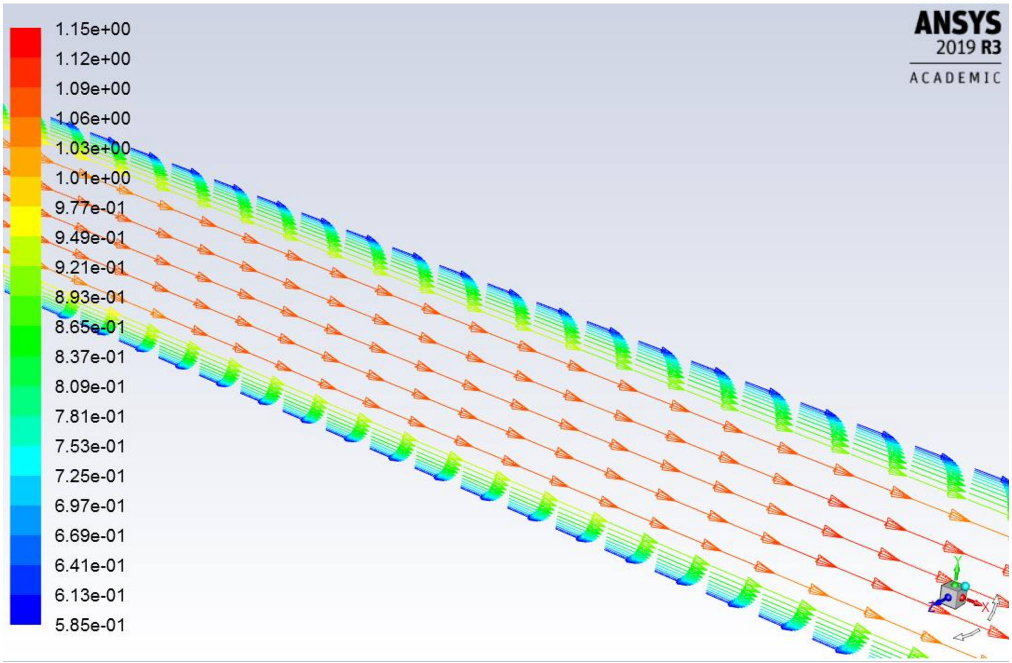


Fig. 5.5 - Speed vectors for Scale = 1 and Skip = 1 ($u[m.s^{-1}]$)

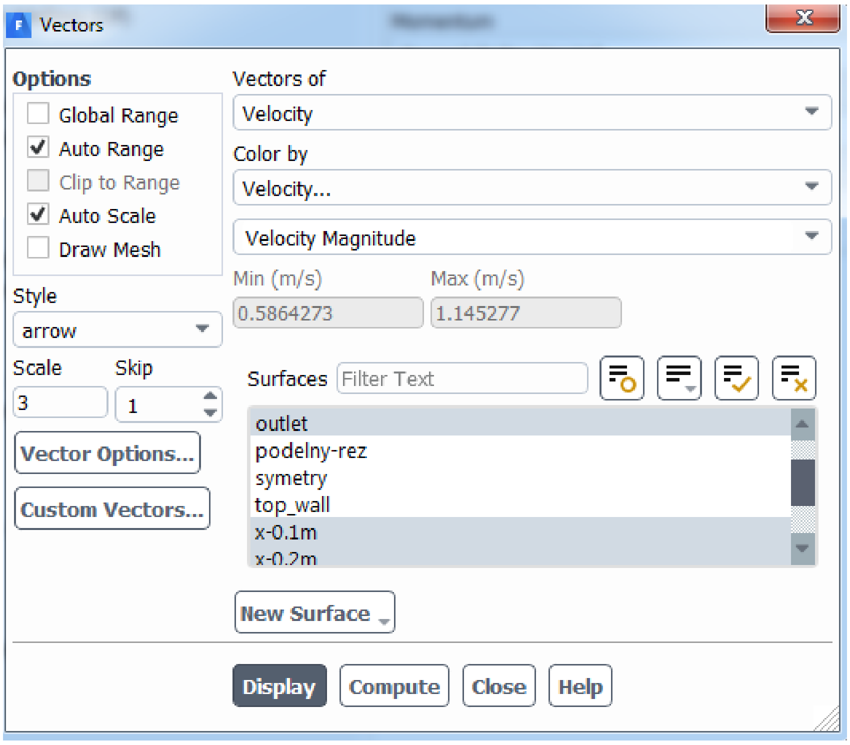
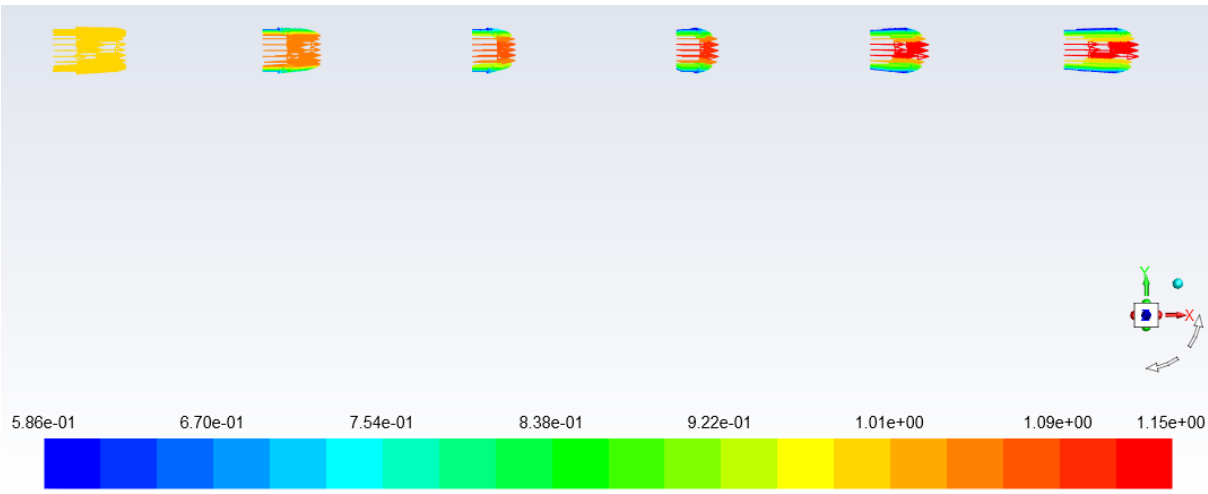


Fig. 5.6 - Menu for evaluation of velocity vectors in individual cross-sections



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**“.

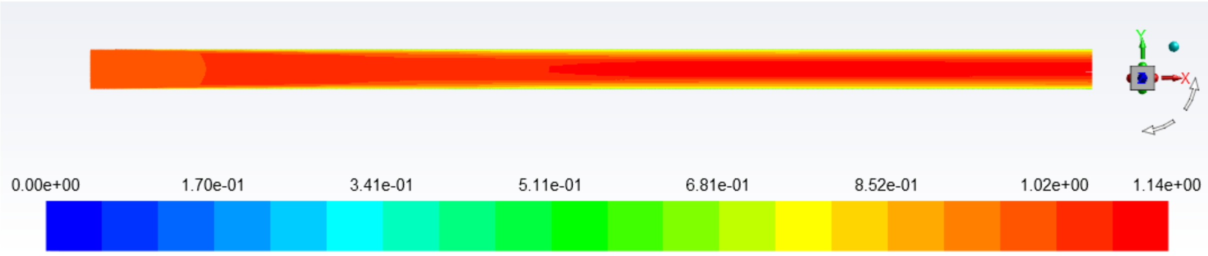


Fig. 5.8 - Contour velocities in the computational area ($u[m.s^{-1}]$)

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

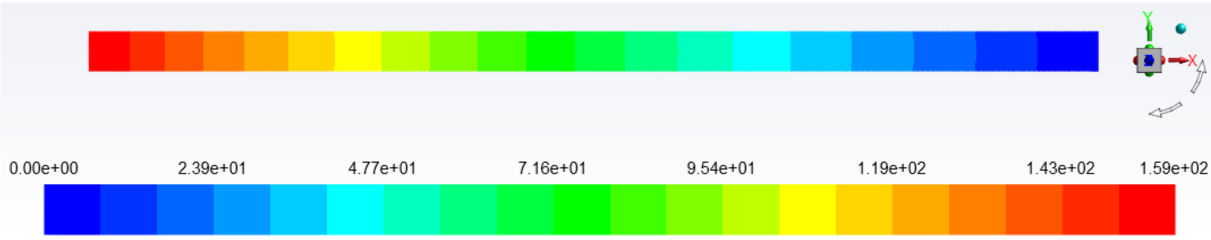


Fig. 5.9 - Filled static pressure lines in the computational area p_{stat} [Pa]

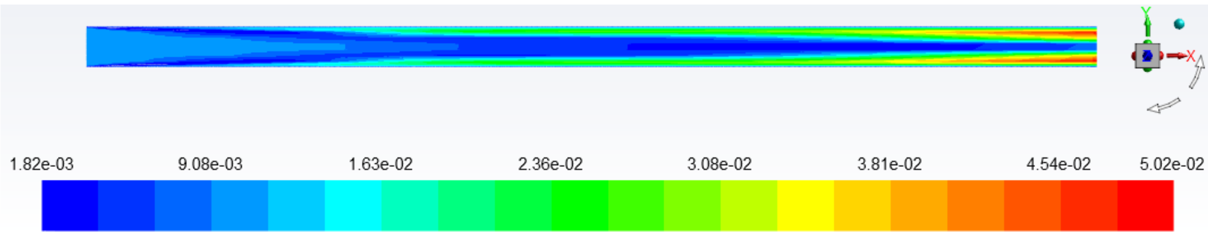


Fig. 5.10 - Filled isolines of effective viscosity

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.

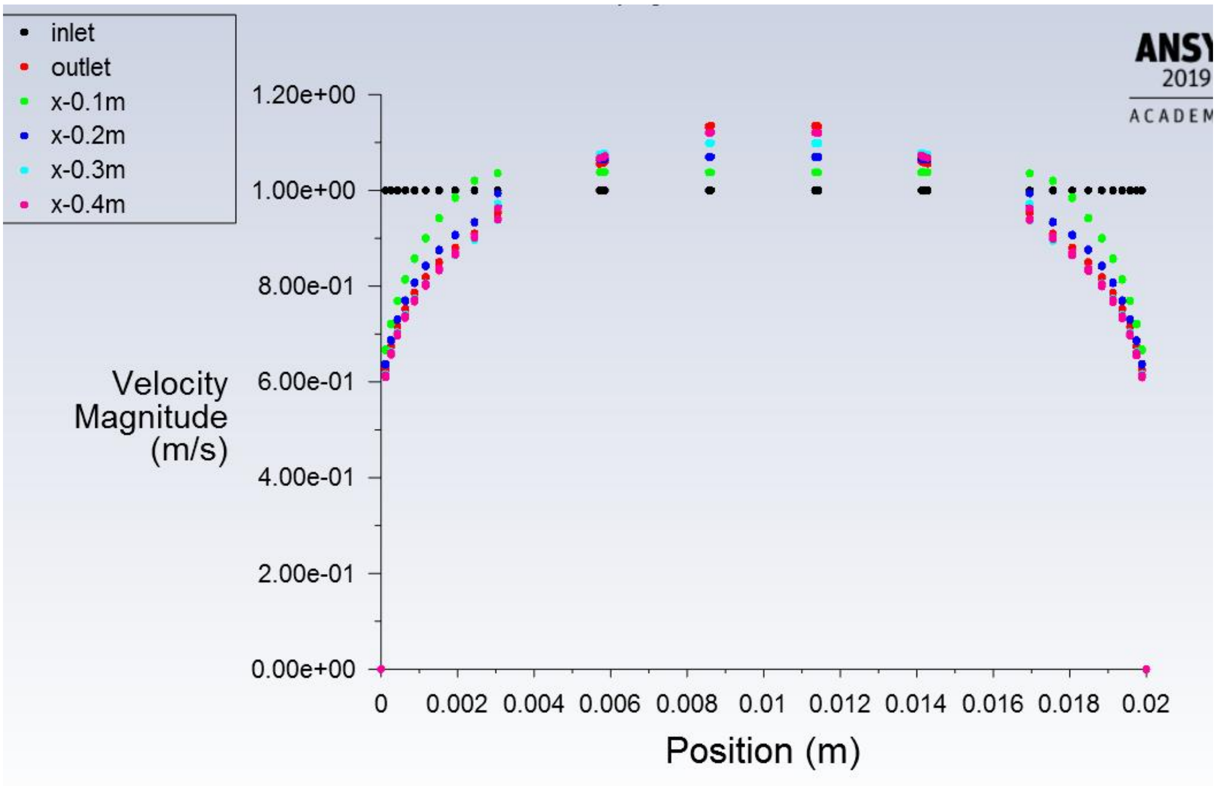


Fig. 5.11 - Formation of the speed profile

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.

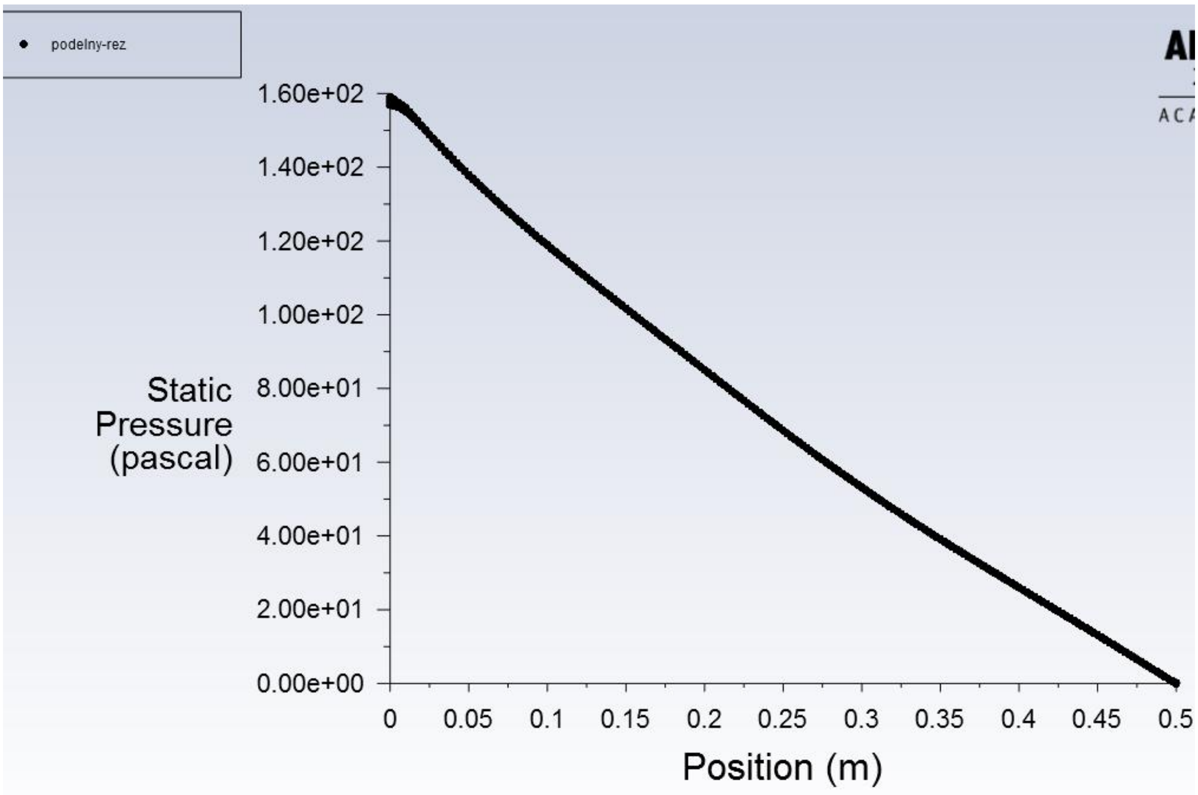


Fig. 5.12 The course of static pressure along the length evaluated in the pipe axis ([Pa])

6 SAMPLE EXAMPLE SOLUTION - CO-CURRENT EXCHANGER

Create a mathematical model of the co-current exchanger and perform a three-dimensional (3D) numerical simulation. The flowing fluids in the exchanger are water-air 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.

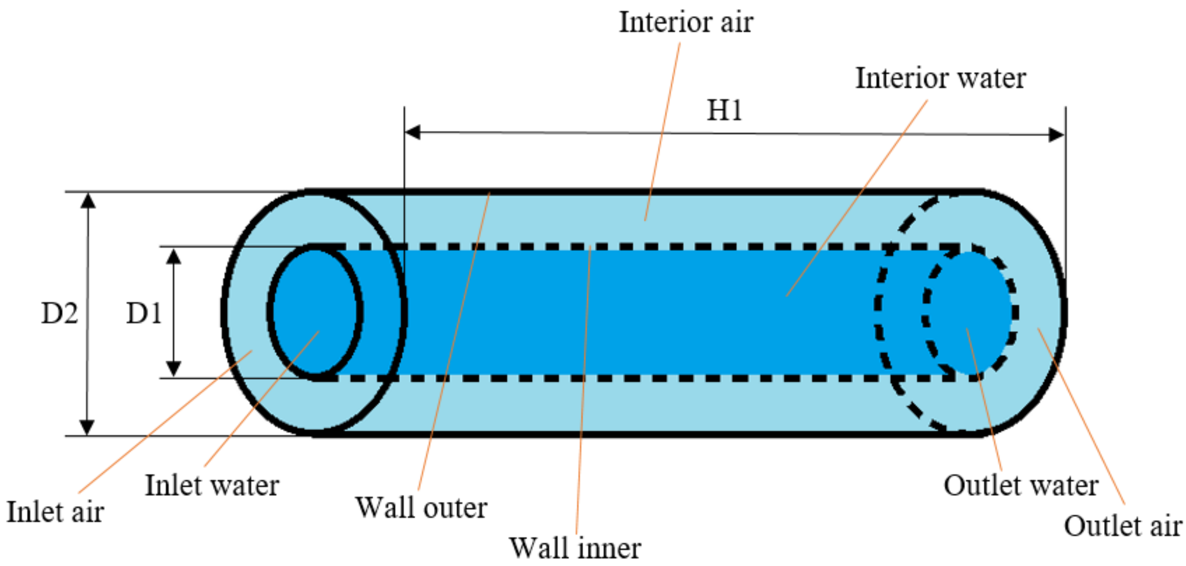


Fig. 6.1 - Co-current exchanger in 3D design.

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 ρ	8030	998.2	1.225	[kg.m^{-3}]
specific heat capacity c_p	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 p			0	0			[Pa]
intensity of turbulence I	1	1	1	1			[%]
hydraulic diameter d_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:

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

Re for air flow:

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

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 $\tilde{\alpha} = \frac{\mathrm{Nu}.\lambda}{d}$ [2].

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

$$\mathrm{Pr} = \frac{\rho \cdot c_p \cdot \nu}{\lambda} = \frac{998.2 \cdot 4182 \cdot 1.01e-6}{0.6} = 6.99 \tag{6.3}$$

$$Nu = 0{,}023 \cdot Re^{0{,}8} \cdot Pr^{0{,}3}$$

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

Then the heat transfer coefficient is

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

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

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

$$Nu = 0.023 \cdot Re^{0{,}8} \cdot Pr^{0{,}4}$$

$$Nu = 0.023 \cdot 4323^{0{,}8} \cdot 0.707^{0{,}4} = 16.79 \qquad (6.7)$$

Then the heat transfer coefficient is

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

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.

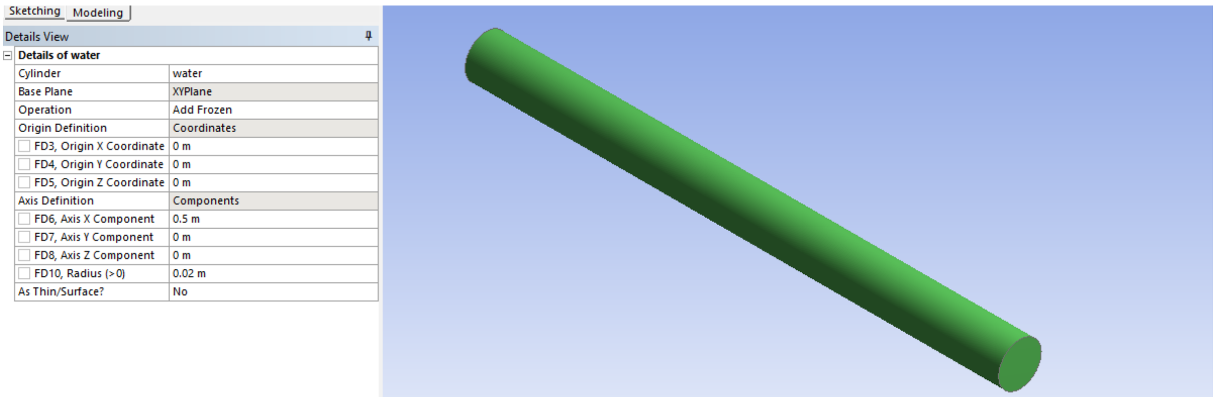


*Fig. 6.2 - Creating an interior water area ("**Cylinder“**)*

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

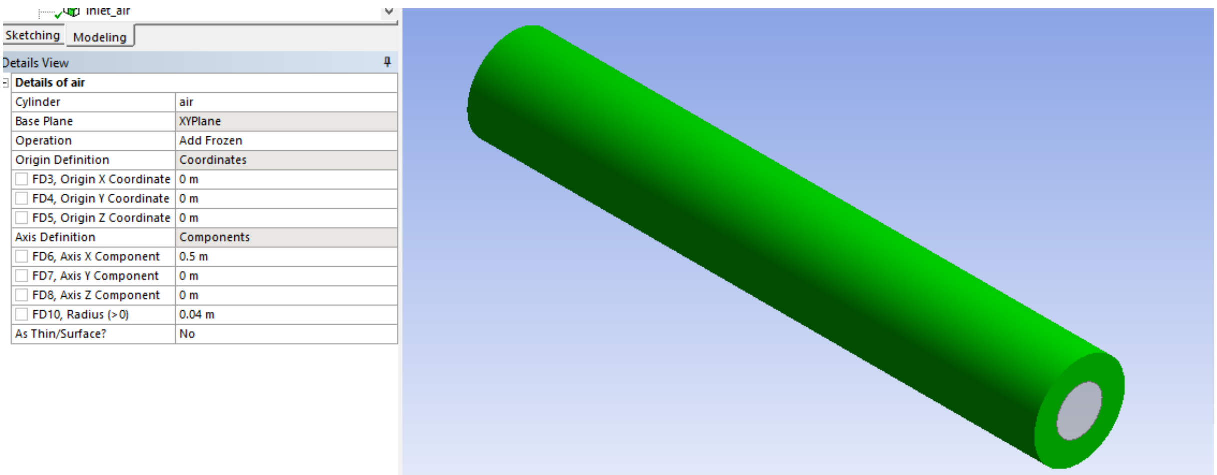


*Fig. 6.3 - Creating an **interior air** area ("**Cylinder**")*

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.

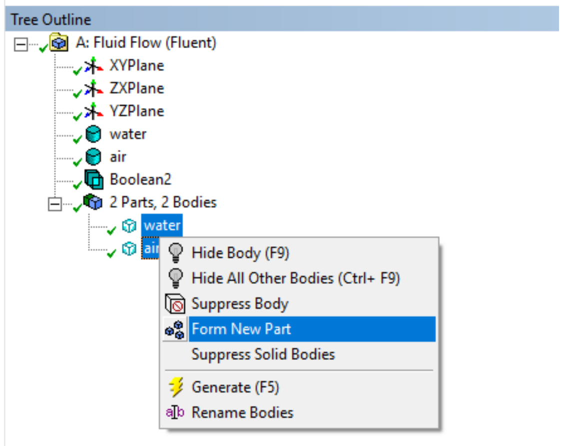


*Fig. 6.4 - Merging volumes into one unit ("**Form New Part**")*

The final form of the "**Form New Part**" command is shown in Figure 6.5.

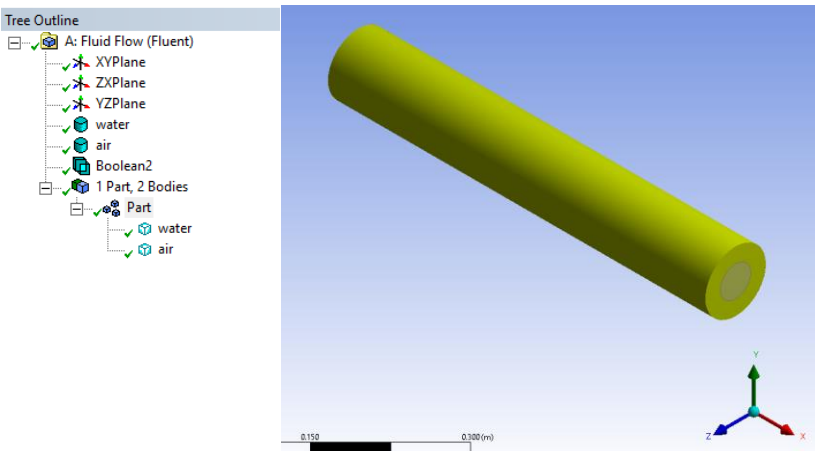


*Fig. 6.5 - The final form of the “**Form New Part**“*

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**).

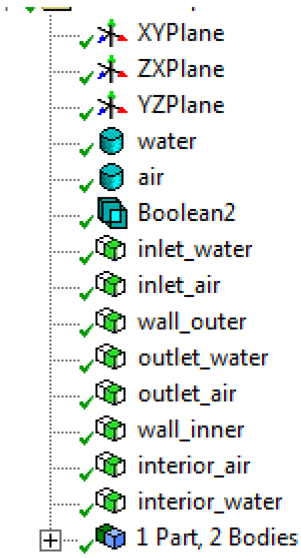


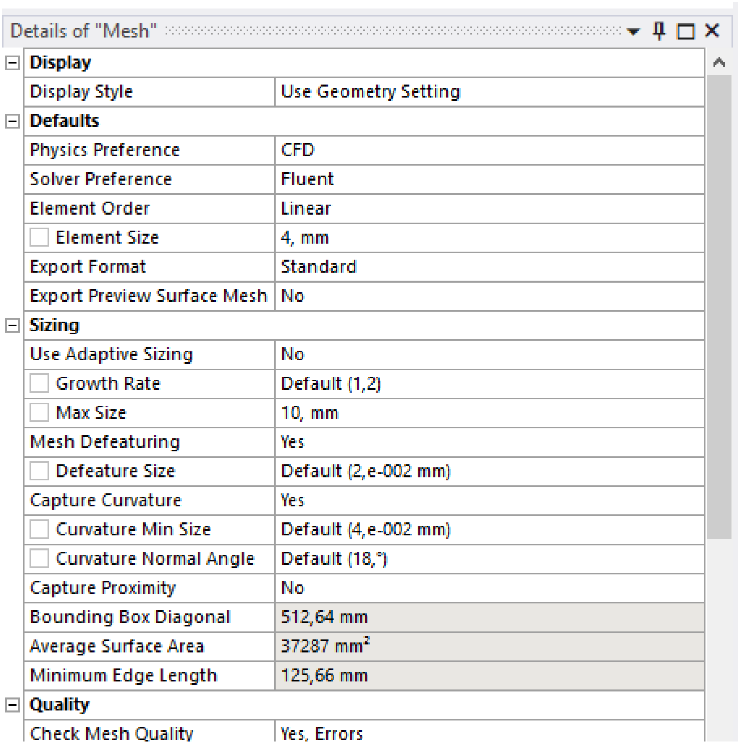
Fig. 6.6 - Marking of boundary conditions

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"

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 mm²
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.

Details of "Inflation 2" - Inflation	
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)
Maximum Layers	6
Growth Rate	1,2
Inflation Algorithm	Pre

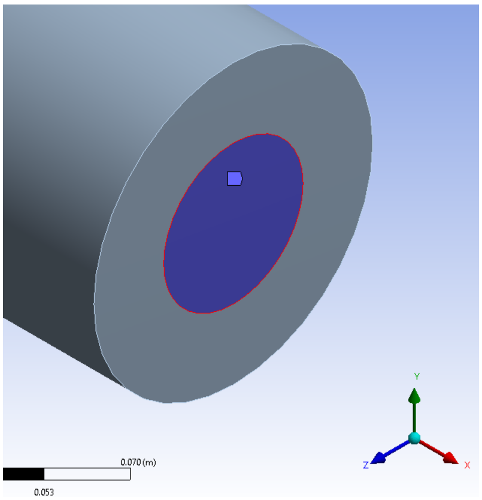


*Fig. 6.8 - Compression parameters for the **interior water** area*

Details of "Inflation" - Inflation	
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

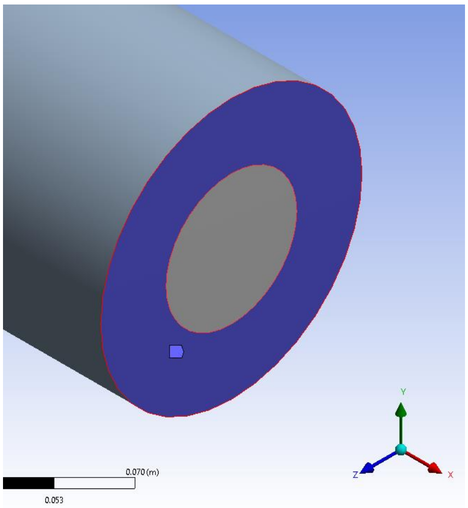


*Fig. 6.9 - Compression parameters for the **interior air** area*

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.

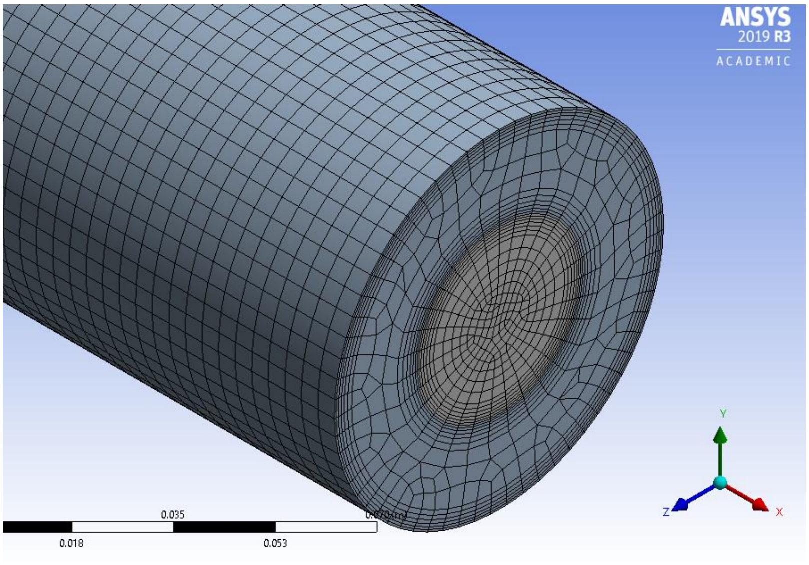


Fig. 6.10 - The resulting form of the mesh

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.

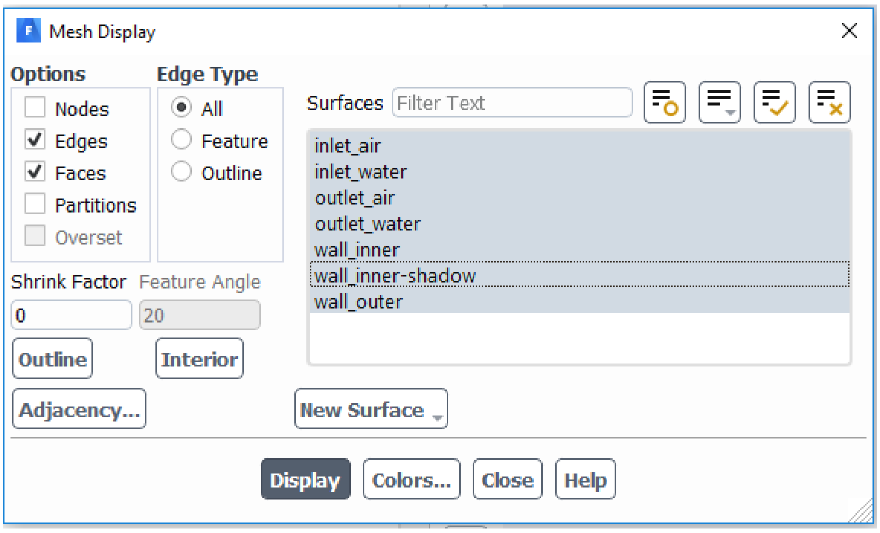


Fig. 6.11 - Checking boundary conditions

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.

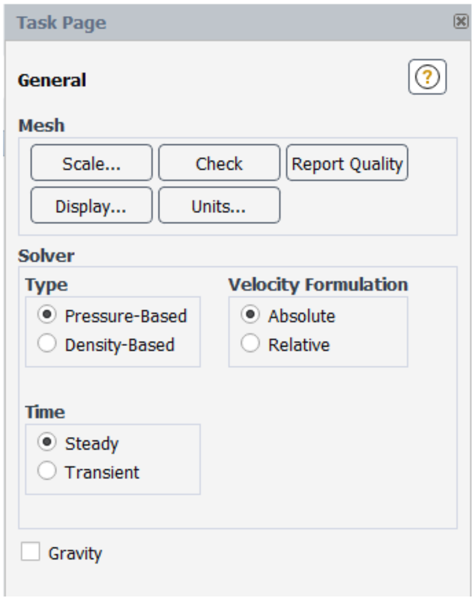


*Fig. 6.12 - Command „**General**“*

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.

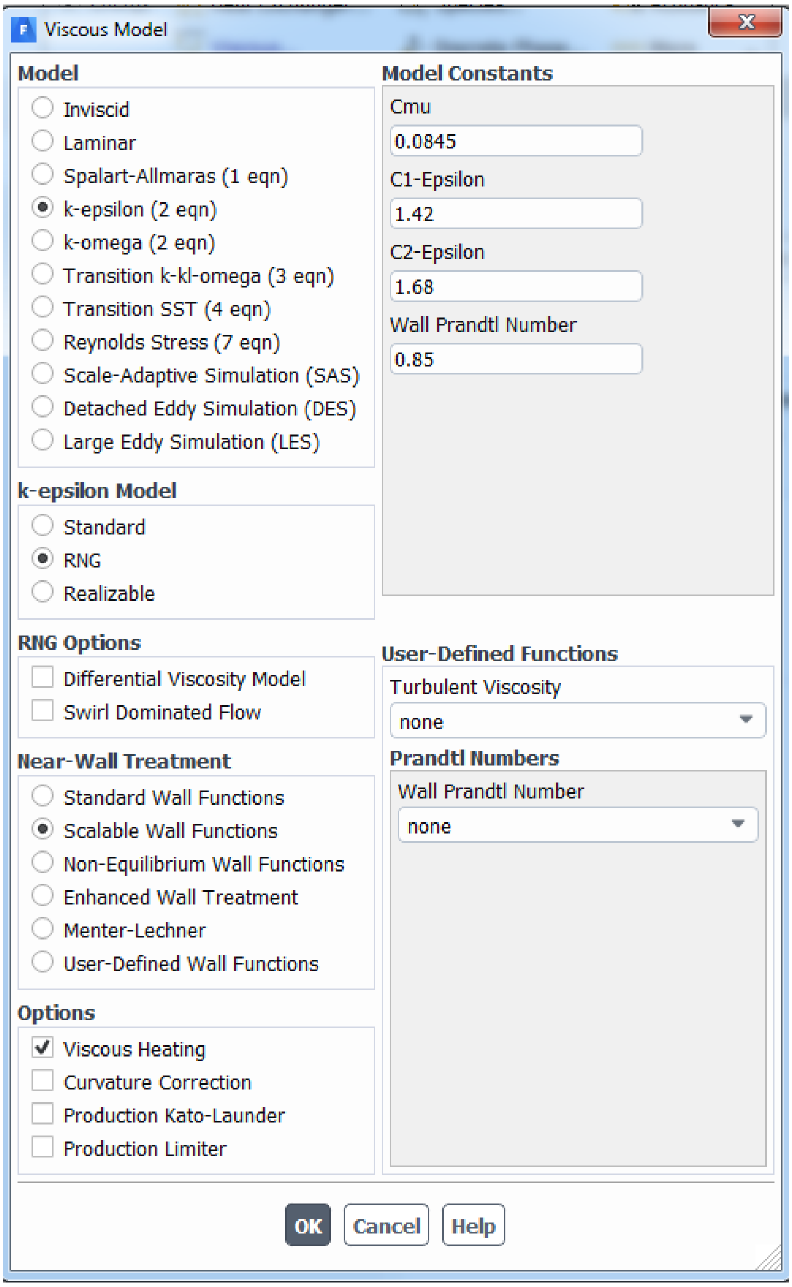


Fig. 6.13 - Setting the mathematical model of the solved problem

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.

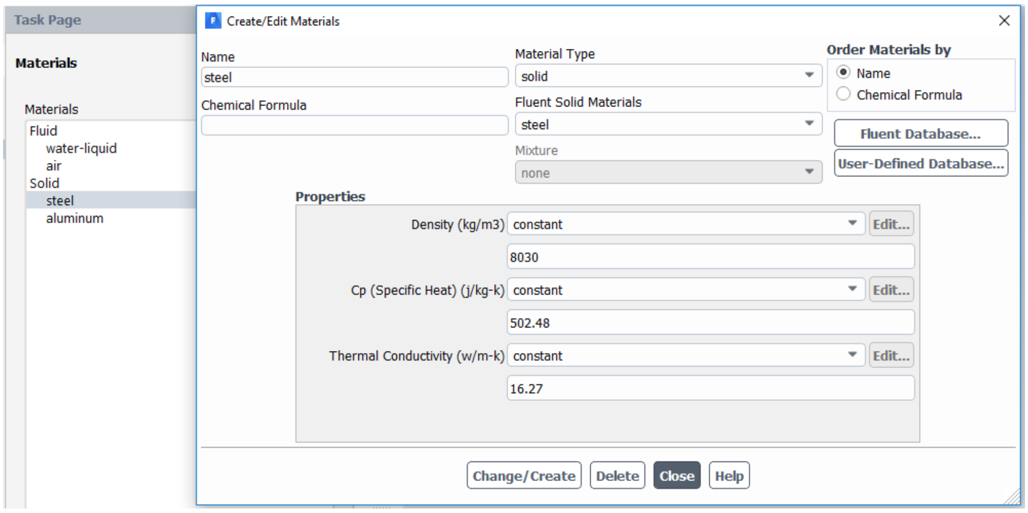


Fig. 6.14 - Required materials for the mathematical model

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.

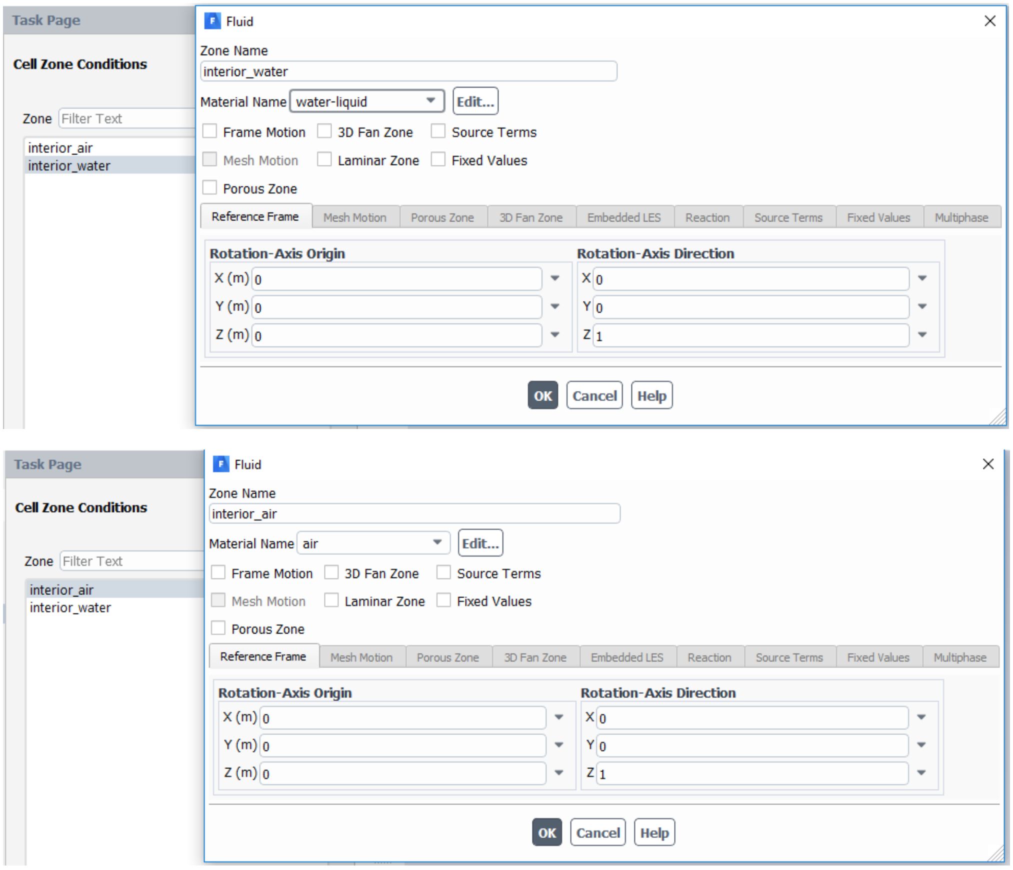


Fig. 6.15 - Defining flow media to given areas

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.

Tab. 6.4 – Types of individual boundary conditions

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

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.

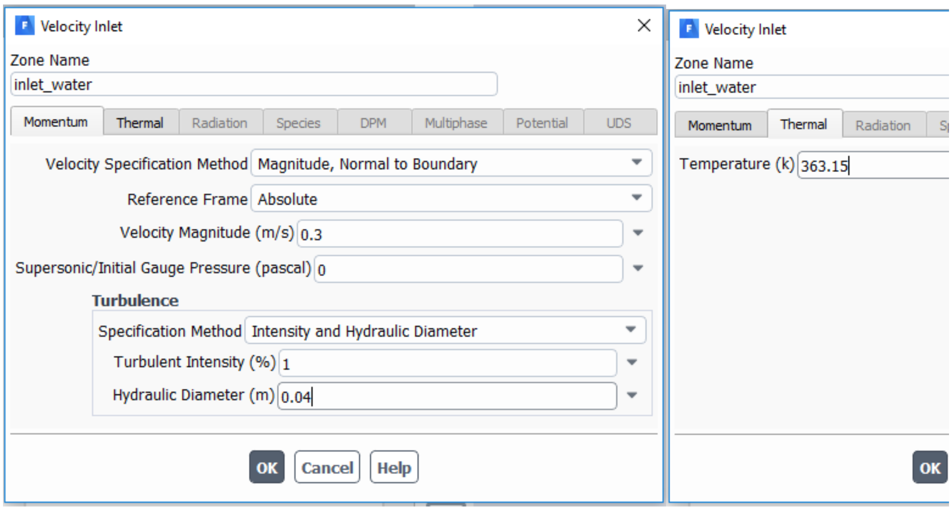


Fig. 6.16 - Parameters of boundary condition ***inlet water***

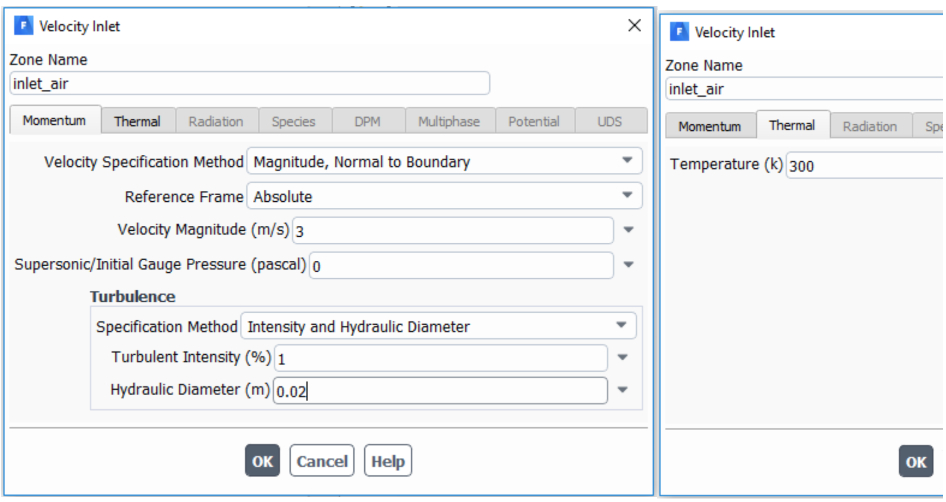


Fig. 6.17 - Parameters of boundary condition ***inlet air***

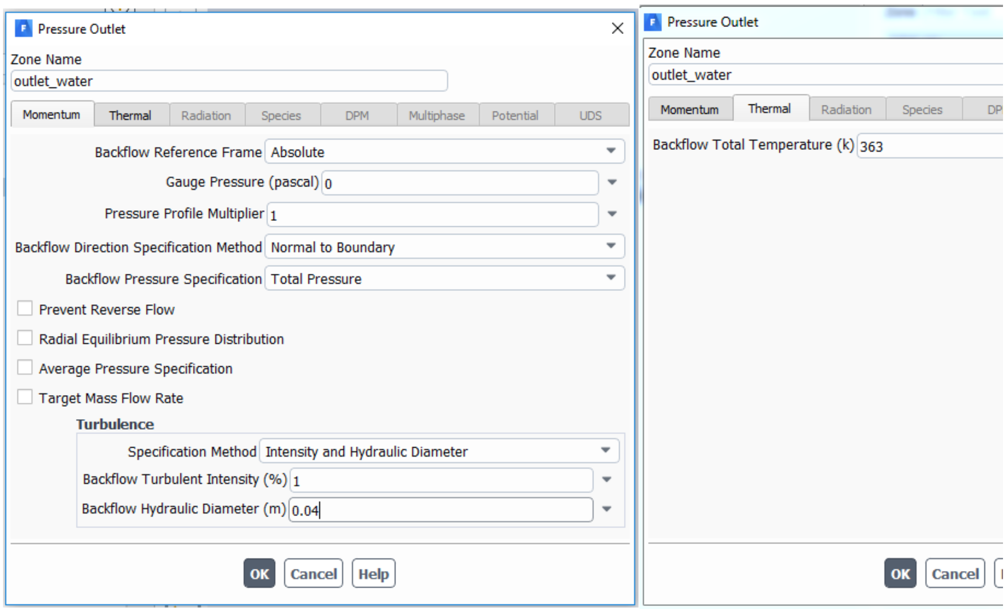


Fig. 6.18 - Parameters of boundary condition **outlet water**

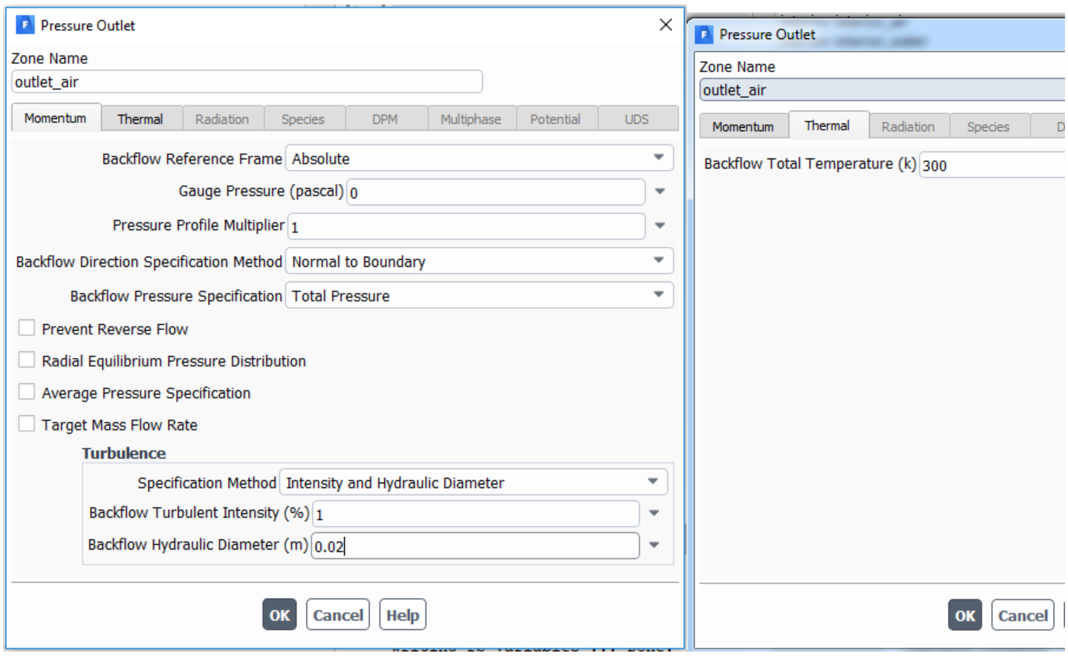


Fig. 6.19 - Parameters of boundary condition **outlet air**

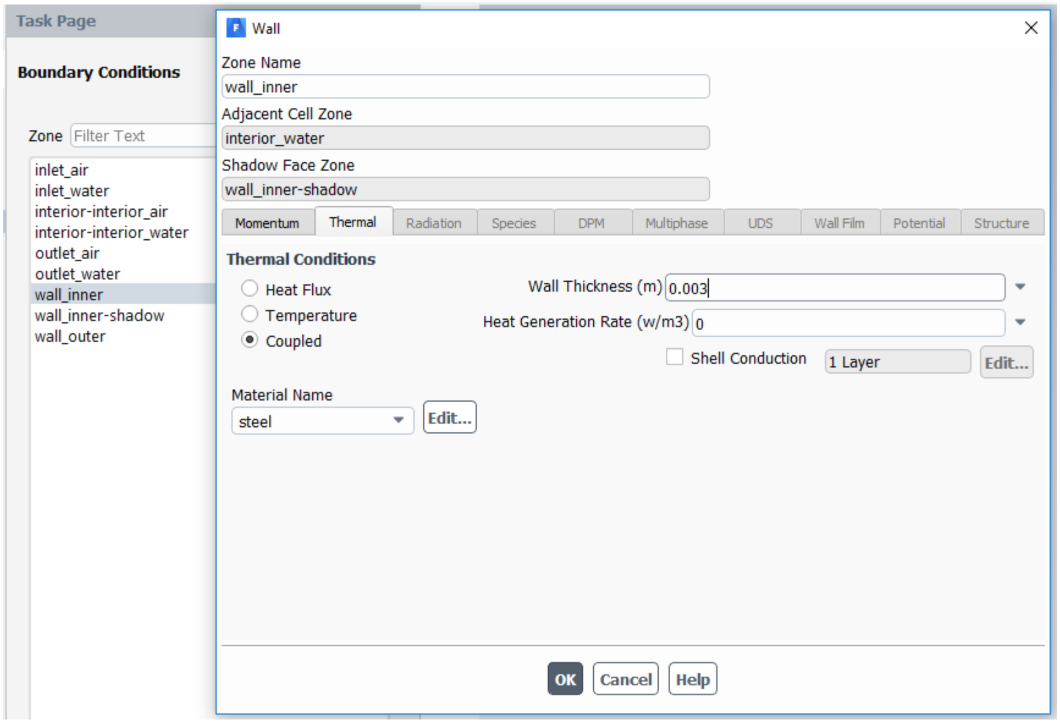


*Fig. 6.20 - Parameters of boundary condition **wall inner***

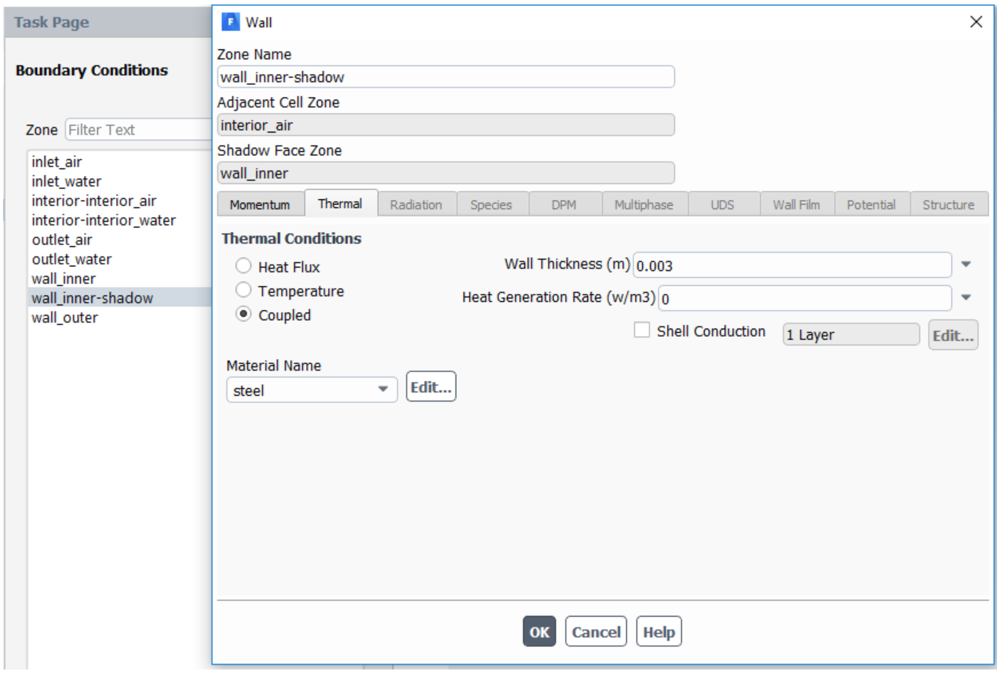


*Fig. 6.21 - Parameters of boundary condition **wall inner-shadow***

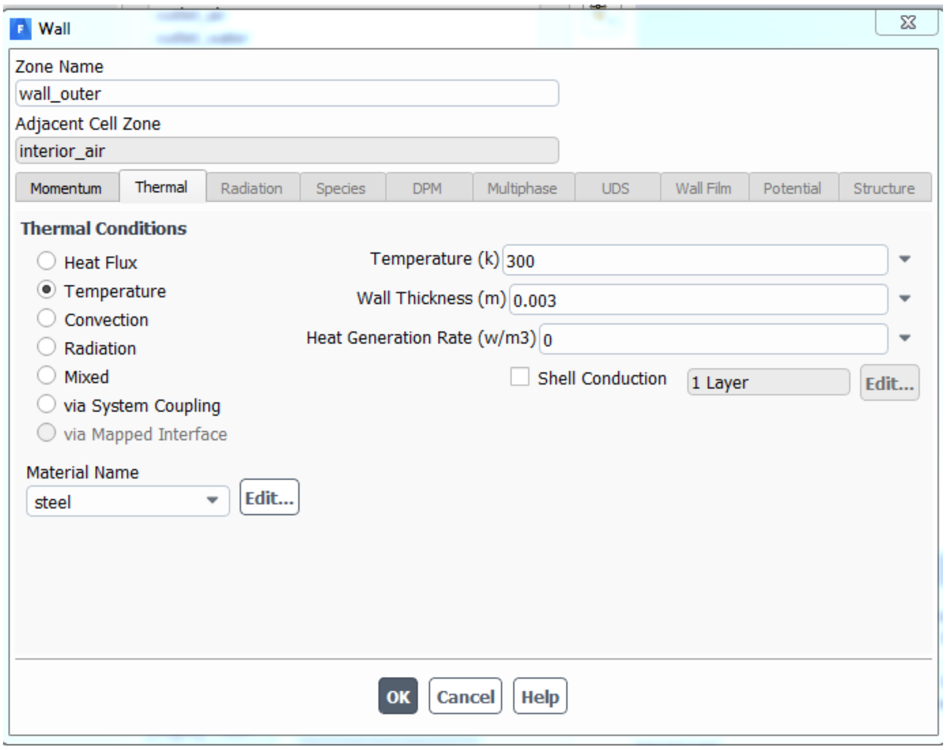


*Fig. 6.22 - Parameters of boundary condition **wall outer***

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.

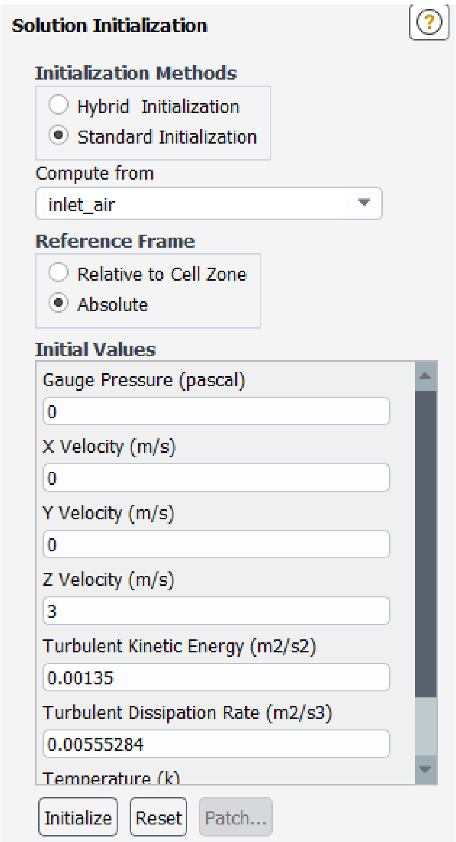


*Fig. 6.23 - Initialization of the computation area („**Solution Initialization**“)*

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.

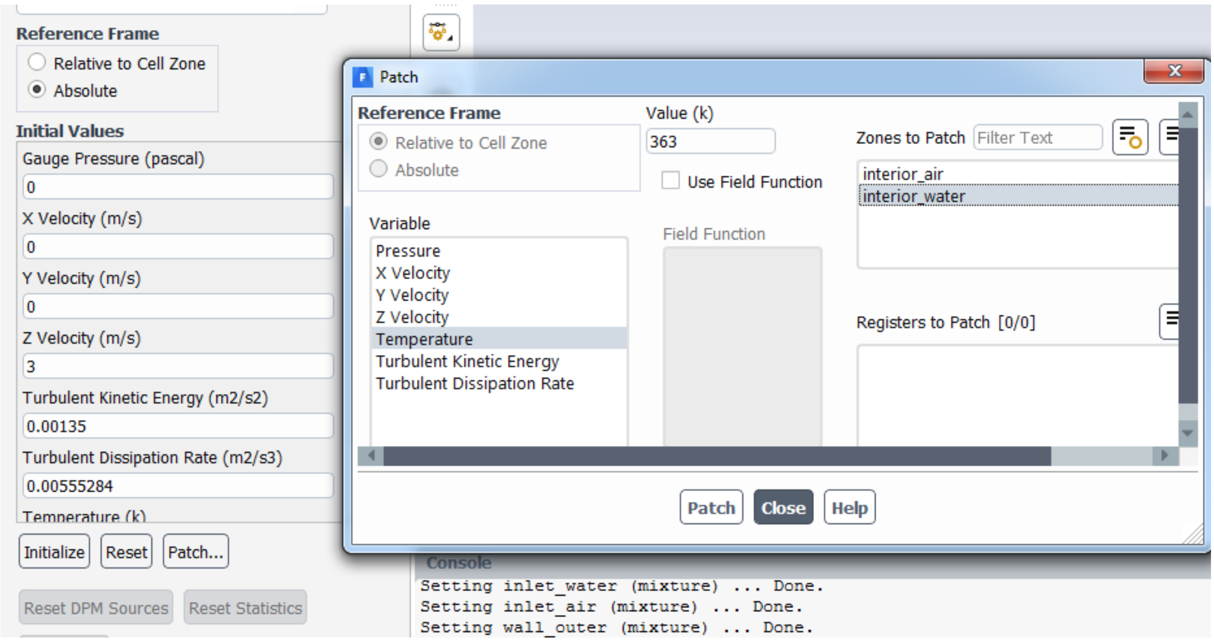


Fig. 6.24 - Initialization of water flow area by command **Patch**

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.

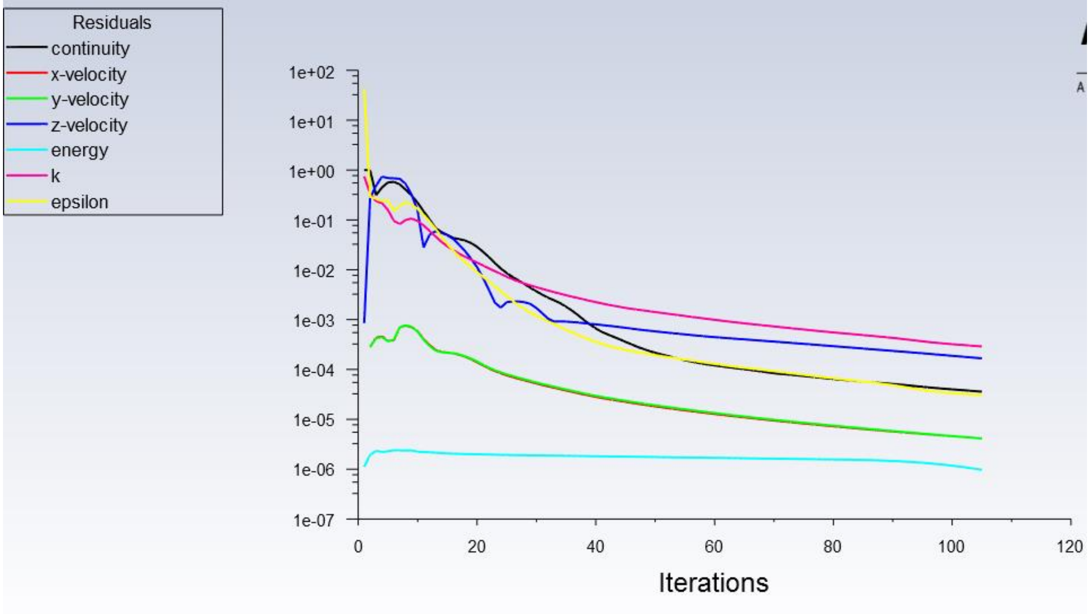


Fig. 6.25 - The course of residuals

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**.

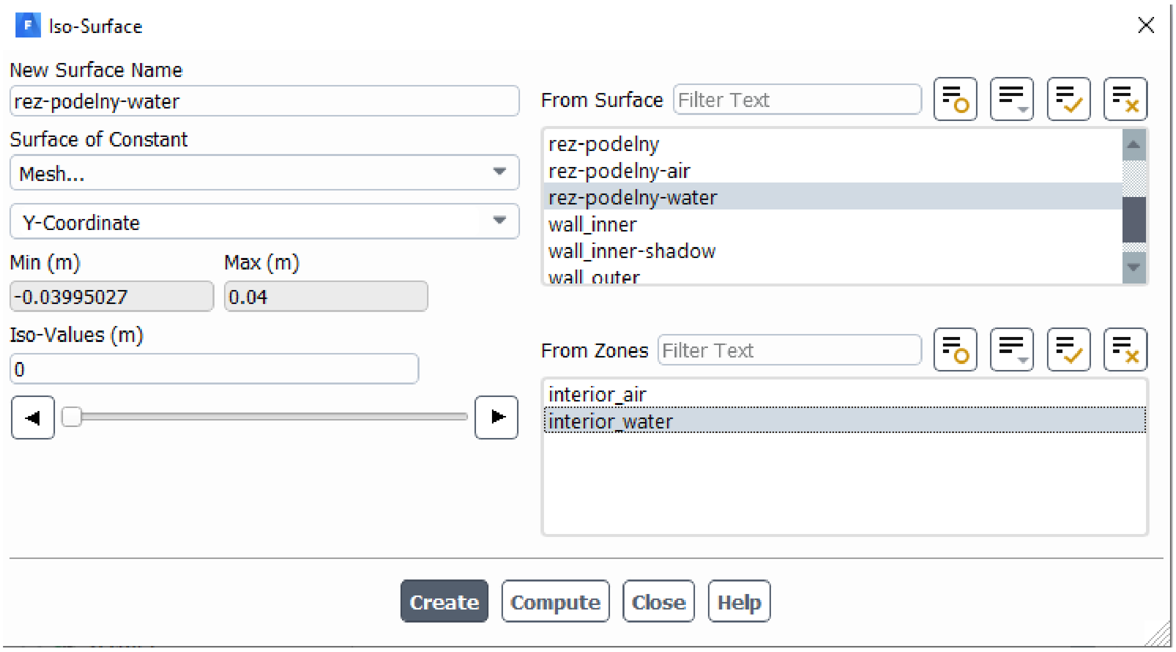


Fig. 6.26 - Creating a longitudinal section in the interior water area

The resulting section is shown in Figure 6.27.

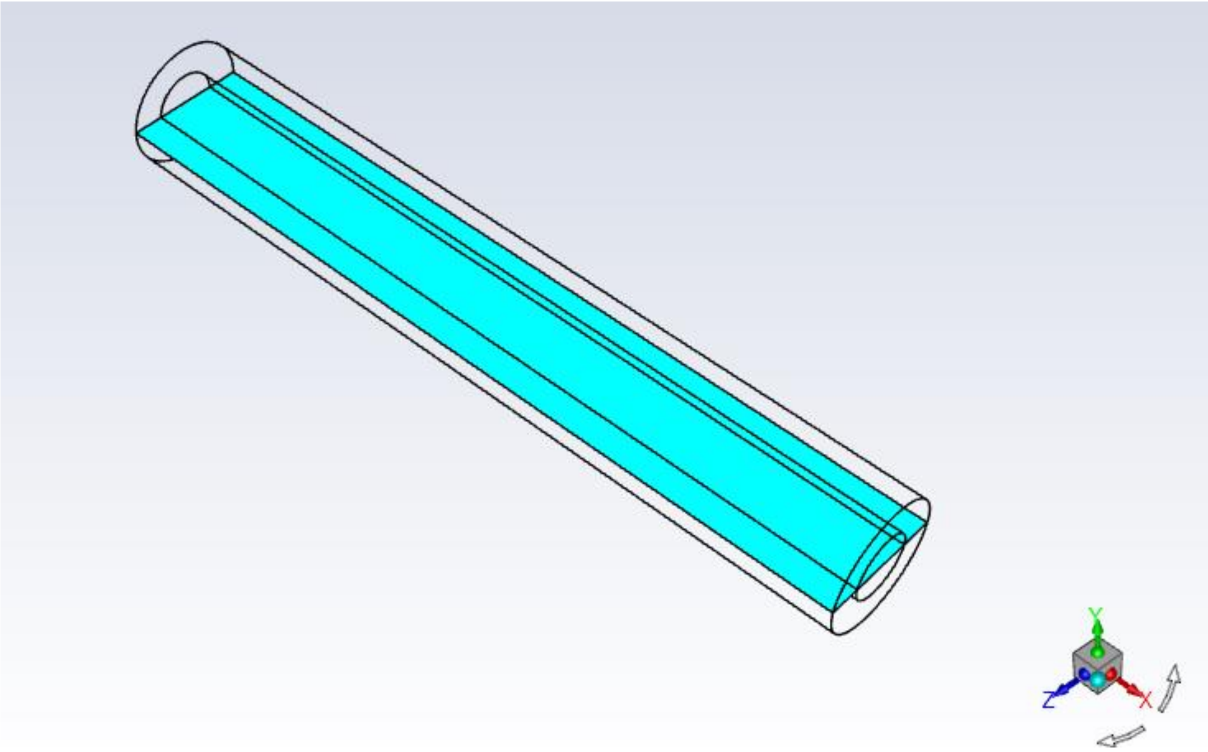


Fig. 6.27– Longitudinal section through the center of the computing area

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.

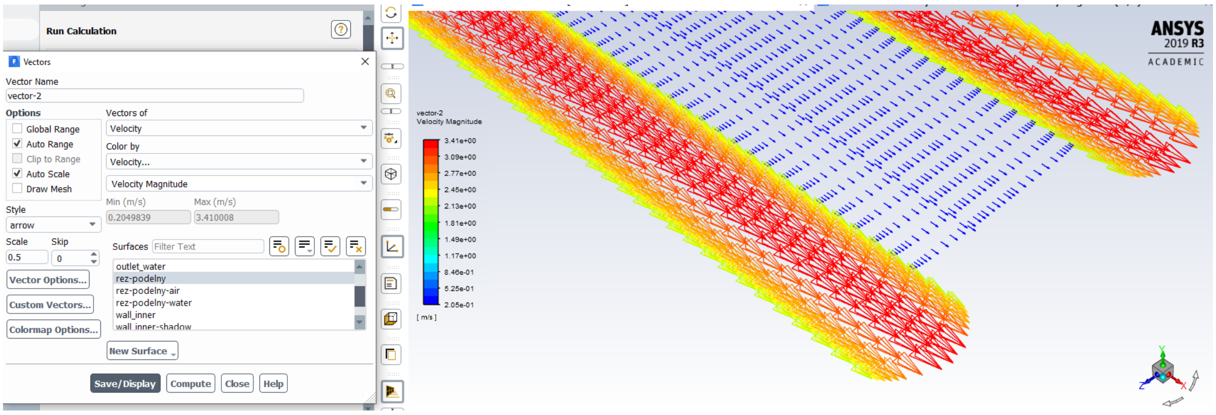


Fig. 6.28 - Velocity vectors ($u[m.s^{-1}]$)

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.

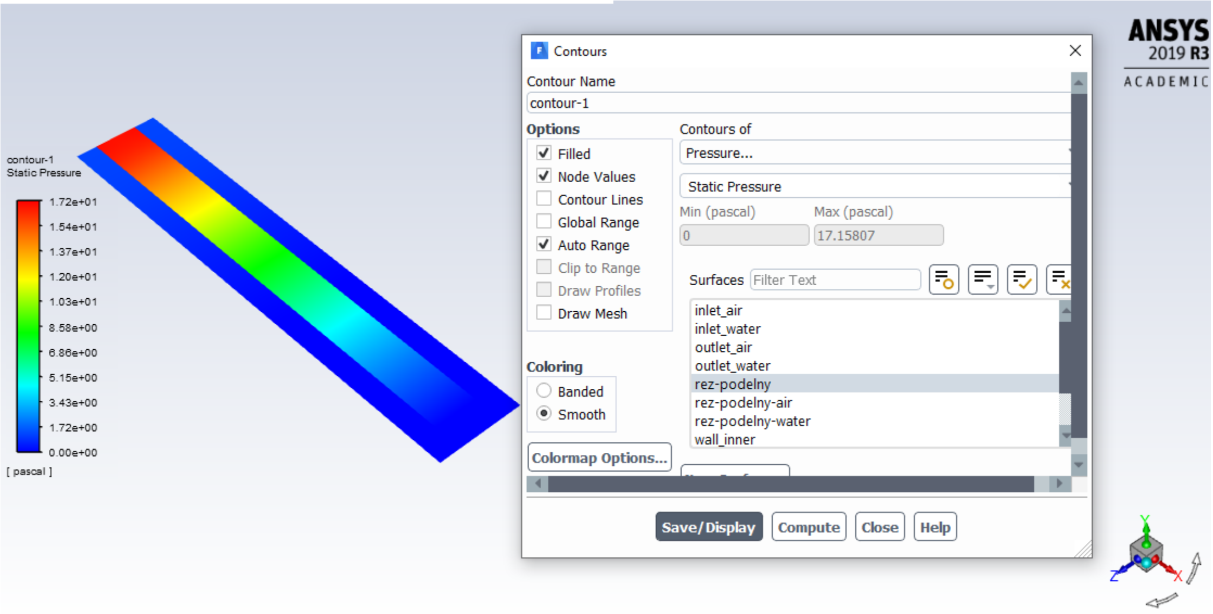


Fig. 6.29 - Static pressure contours (Pa)

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.

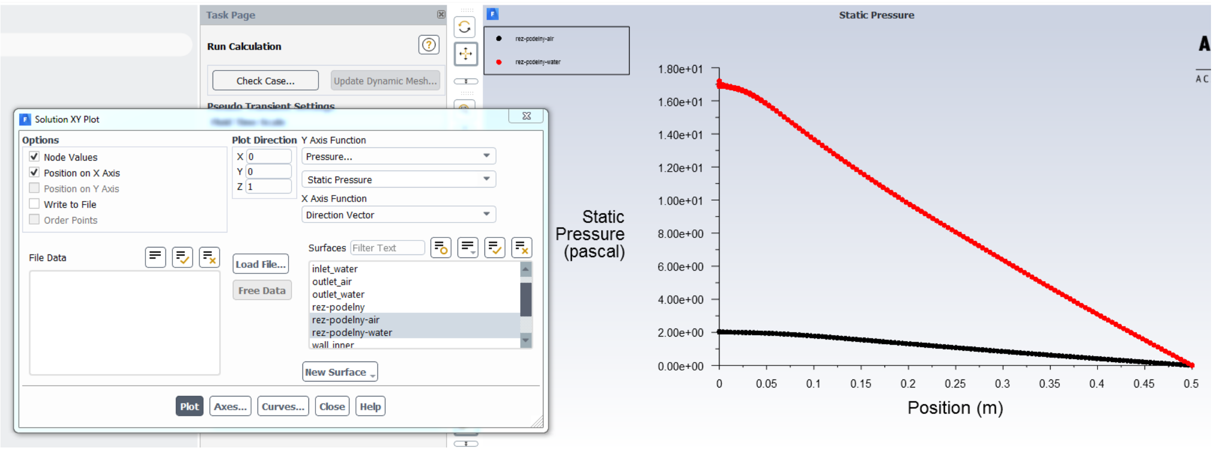


*Fig. 6.30 - Progression of static pressure along length in area **interior water** and **interior air***

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

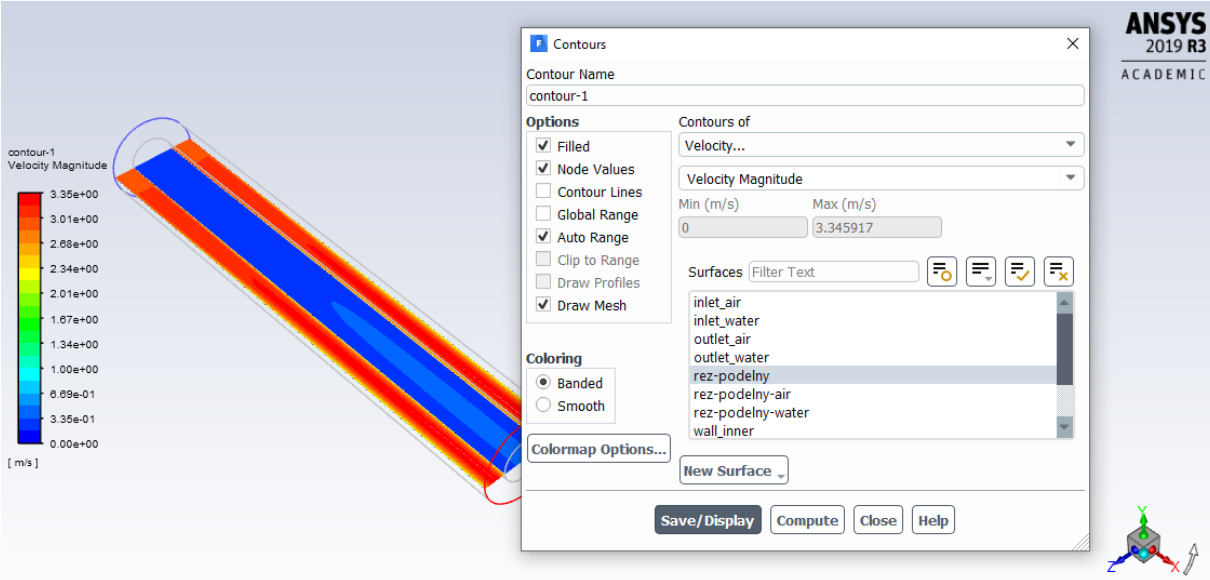


Fig. 6.31 - Velocity contours [m/s]

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

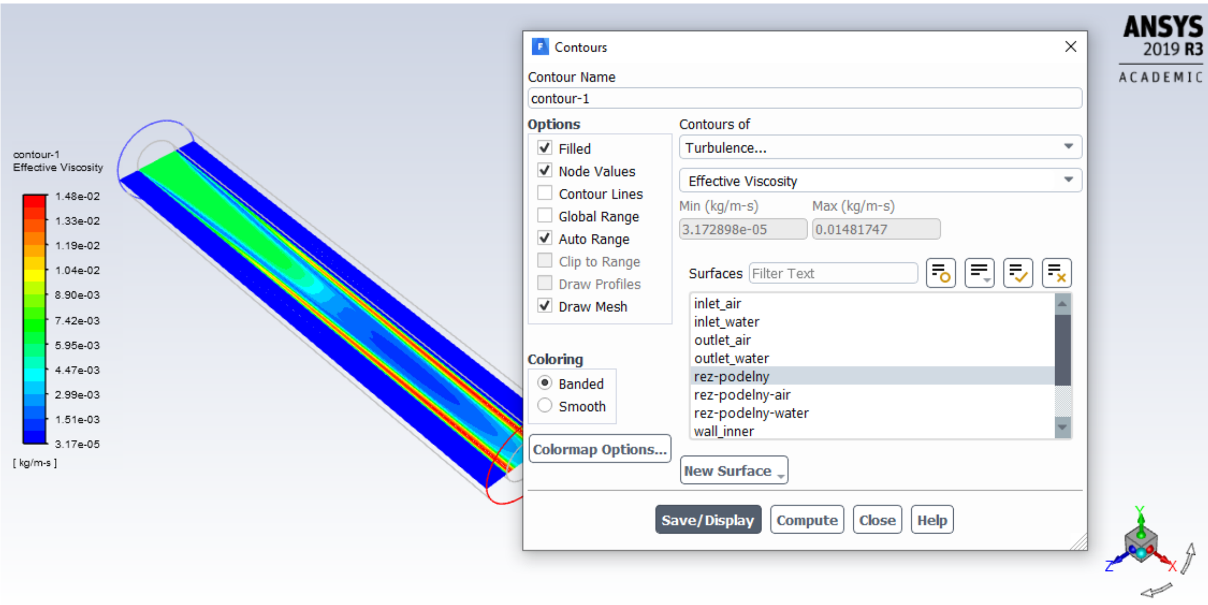


Fig. 6.32 - Effective viscosity [$kg.m^{-1}.s^{-1}$]

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

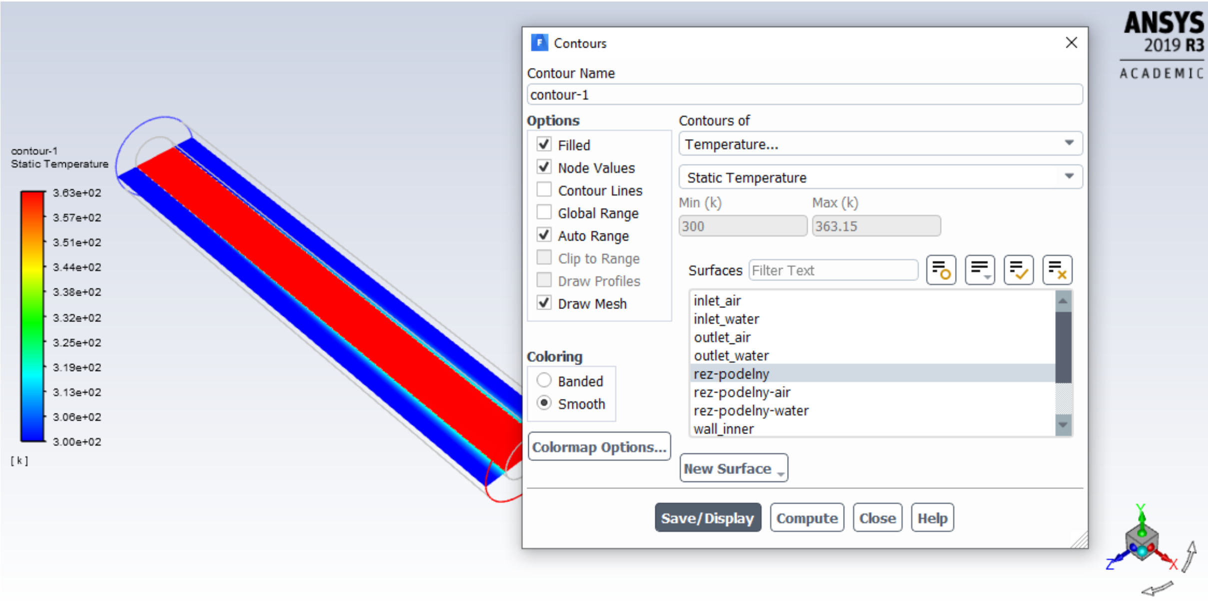


Fig. 6.33 - Temperature field [K]

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)).

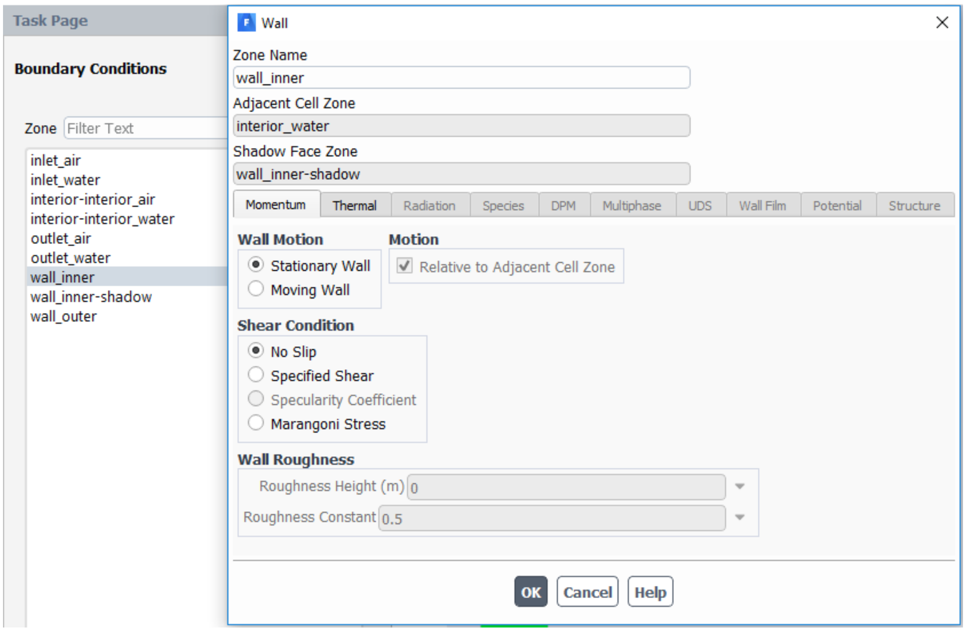
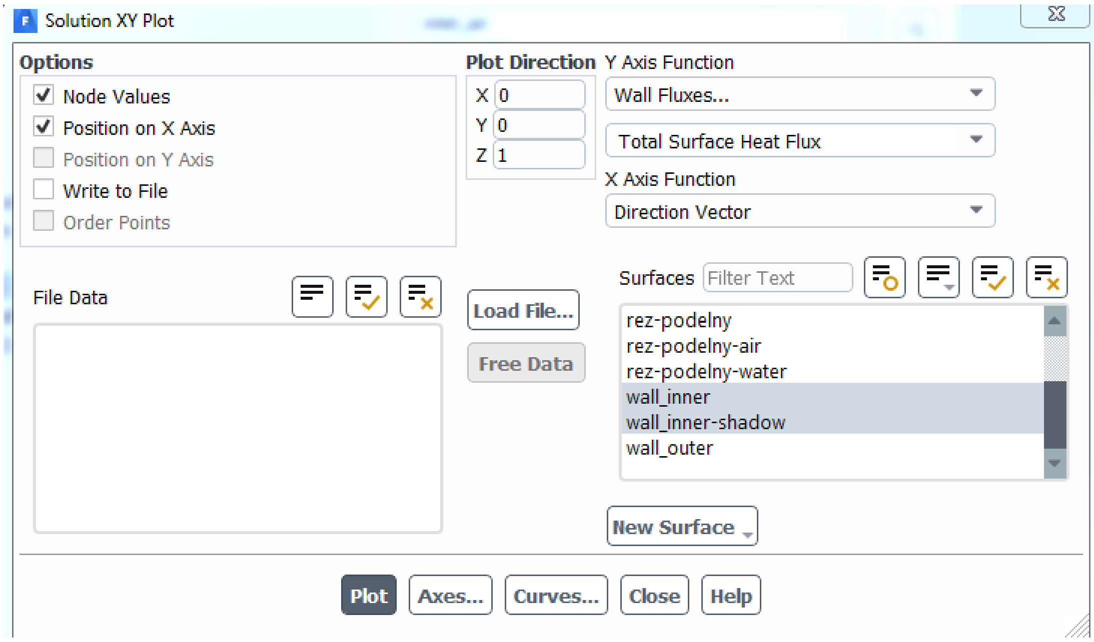


*Fig. 6.34 - Identification of the **wall inner** adjacent to the surrounding water*

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..



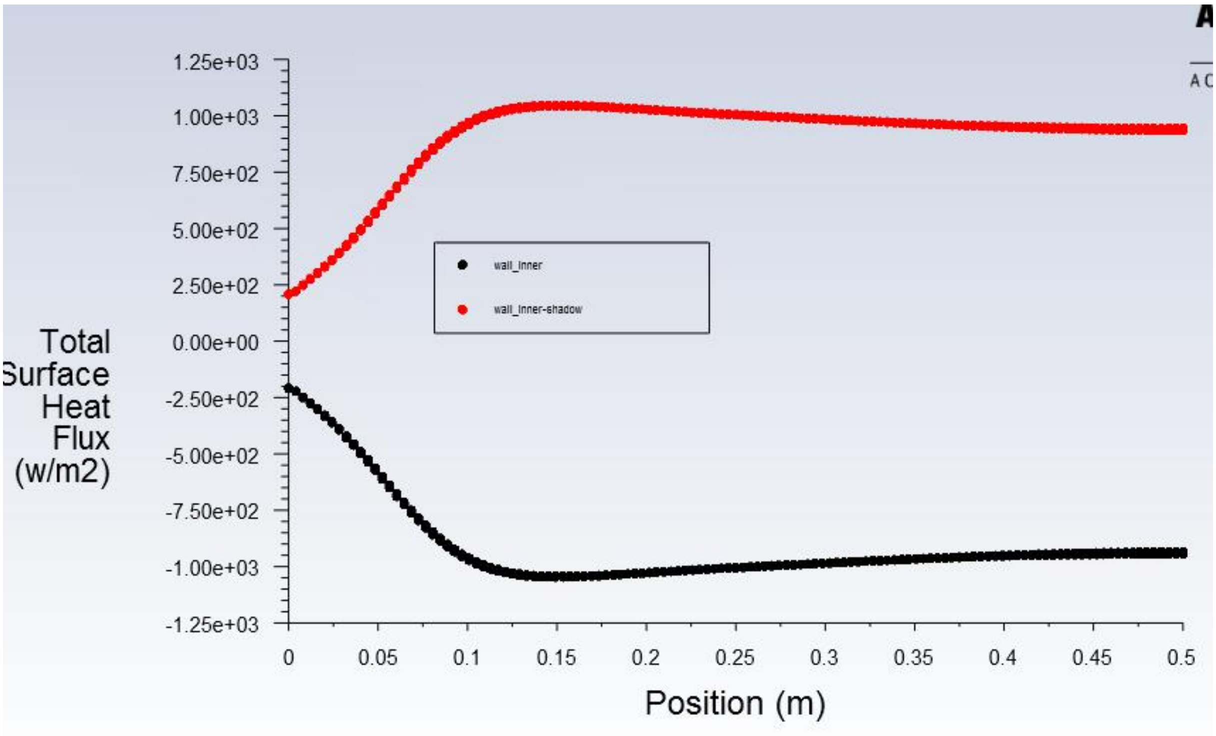


*Fig. 6.35 - Heat flow (W / m²) through the wall inner (**wall inner-shadow**)*

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

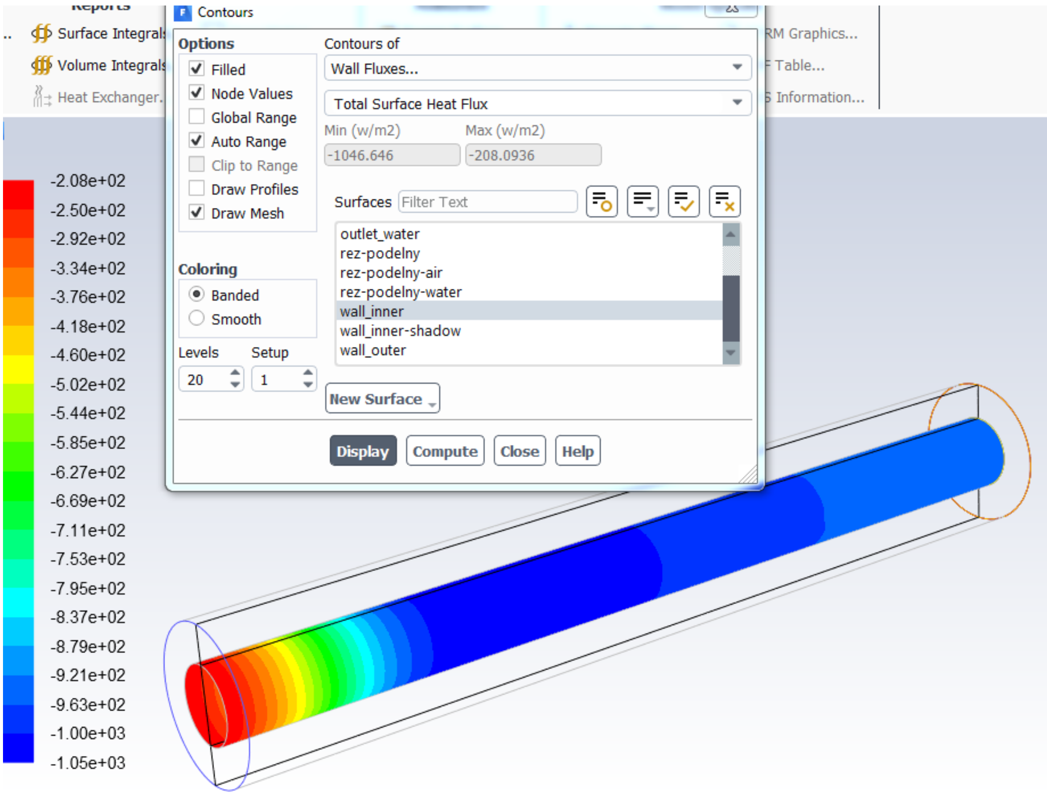


*Fig. 6.36 - Heat flow (W / m^2) through the **wall inner**)*

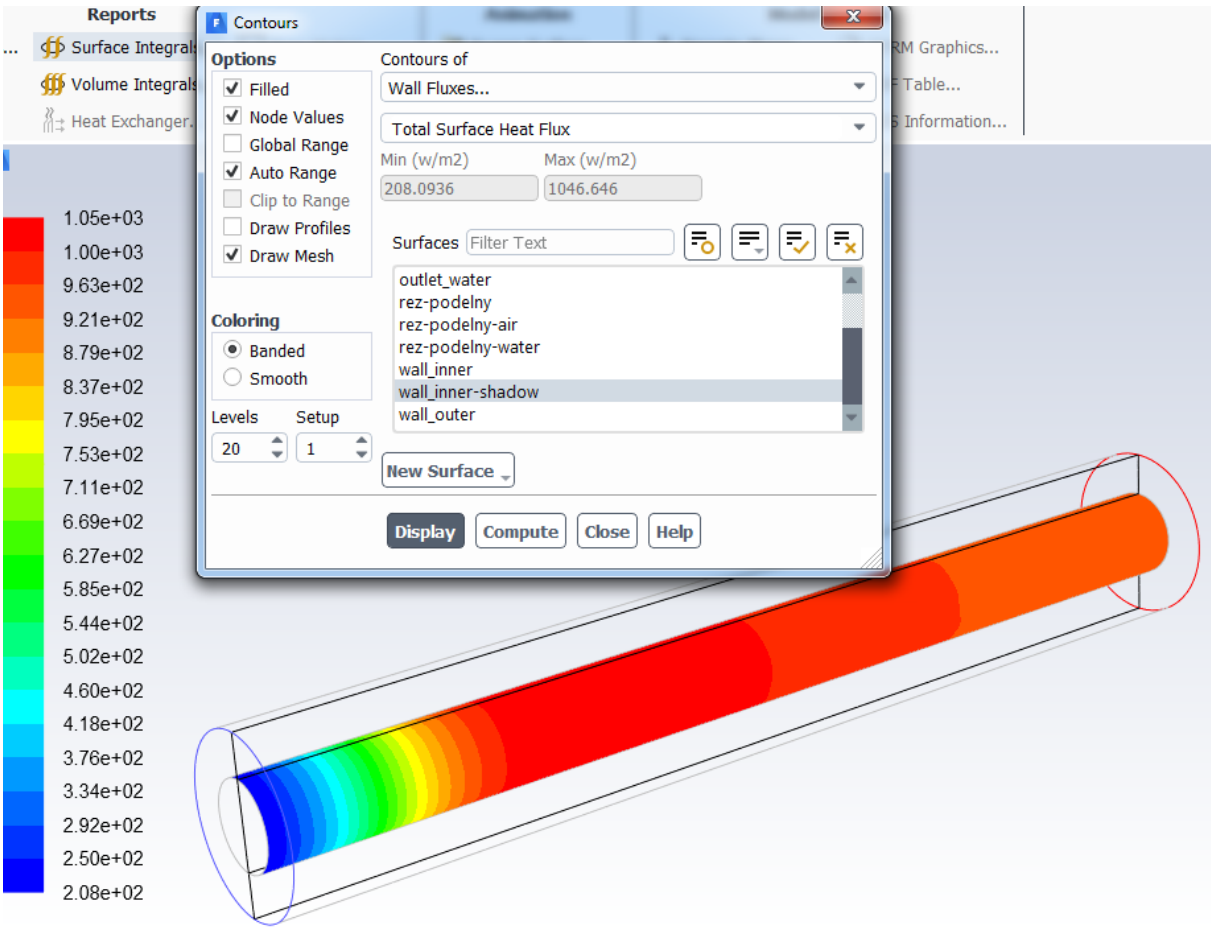


*Fig. 6.37 - Heat flow (W / m^2) through the wall **inner-shadow**)*

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

<u>Evaluation for water</u>

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.

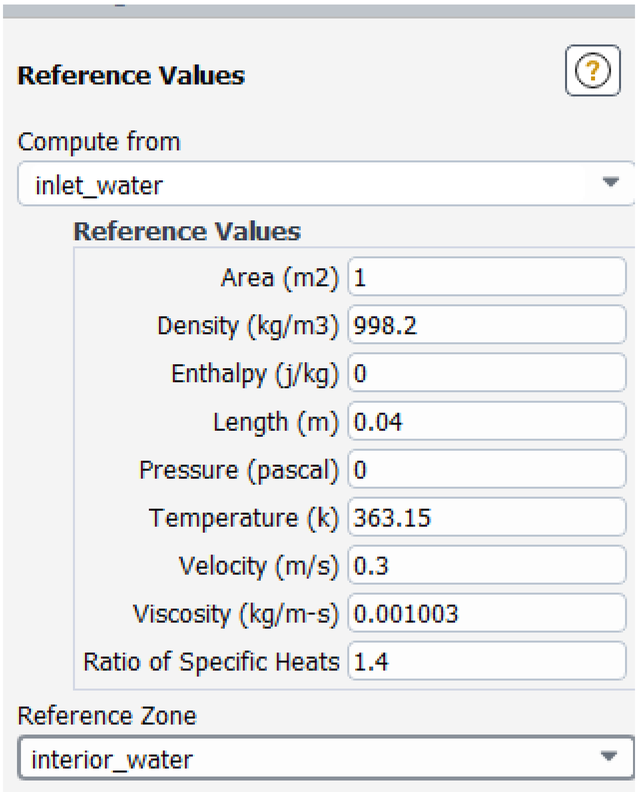


*Fig. 6.38 - Reference values for evaluation into the water for the wall **wall inner***

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.

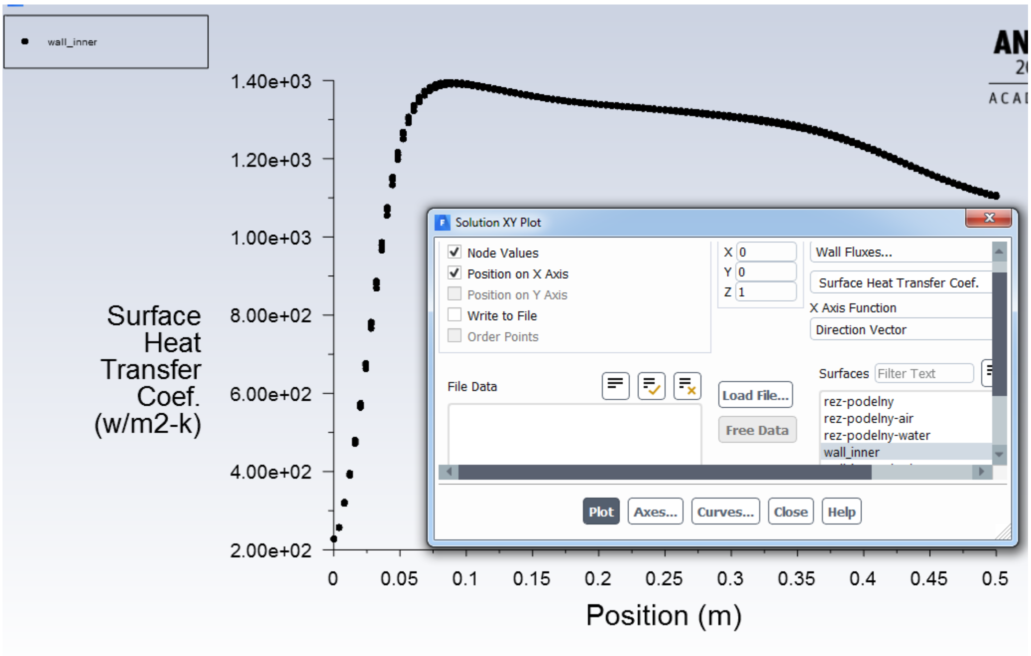
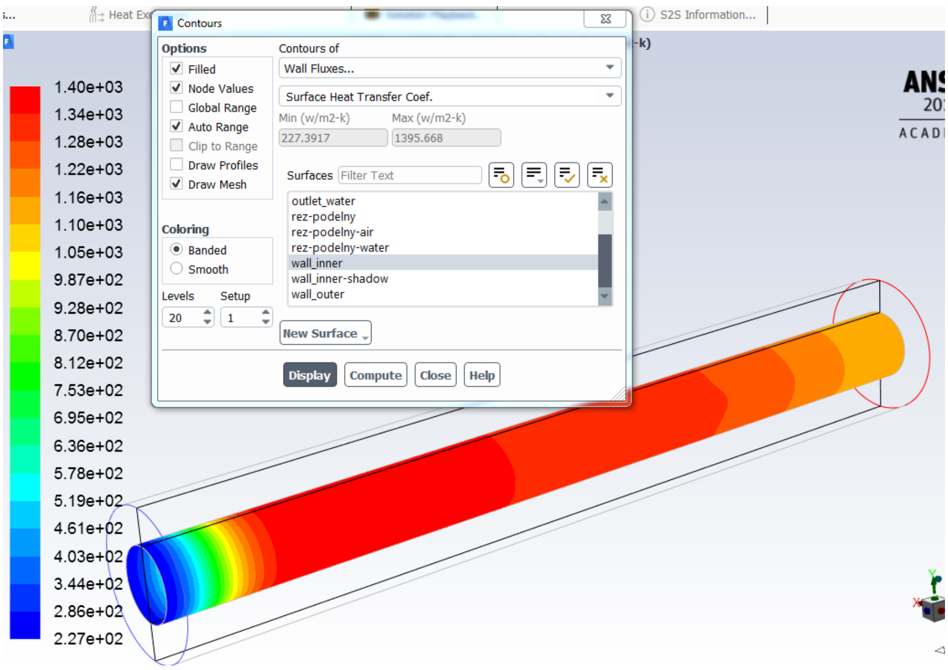


*Fig. 6.39 - Heat transfer coefficient through the interface wall (**wall inner**) [W.m^{-2}.K^{-1}]*

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 odkazů.**).

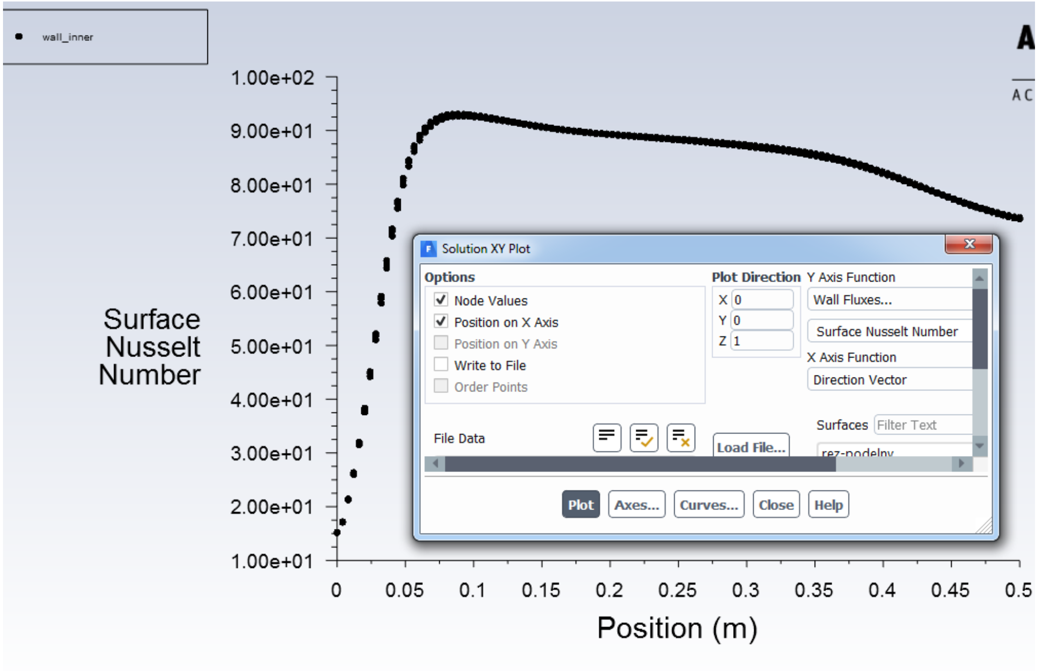


*Fig. 6.41 - Nusselt number evaluated on the interface wall (**wall inner**)*

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.

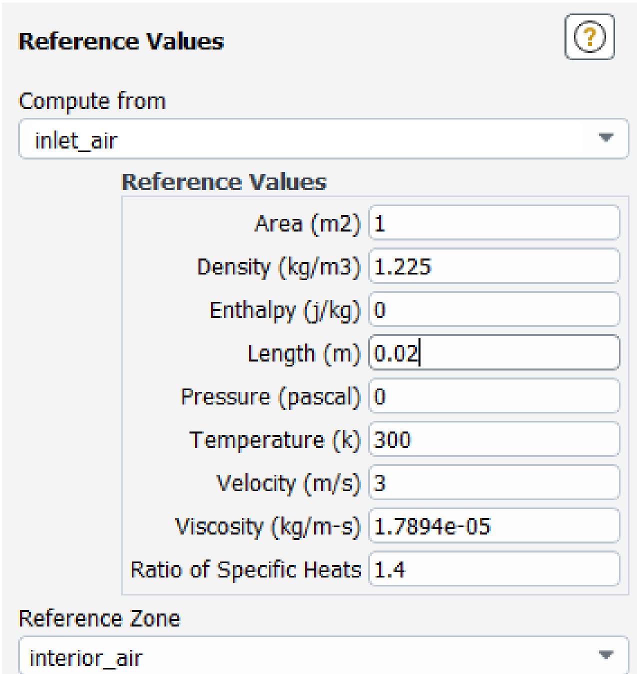


*Fig. 6.43 - Reference values for the wall **wall inner-shadow***

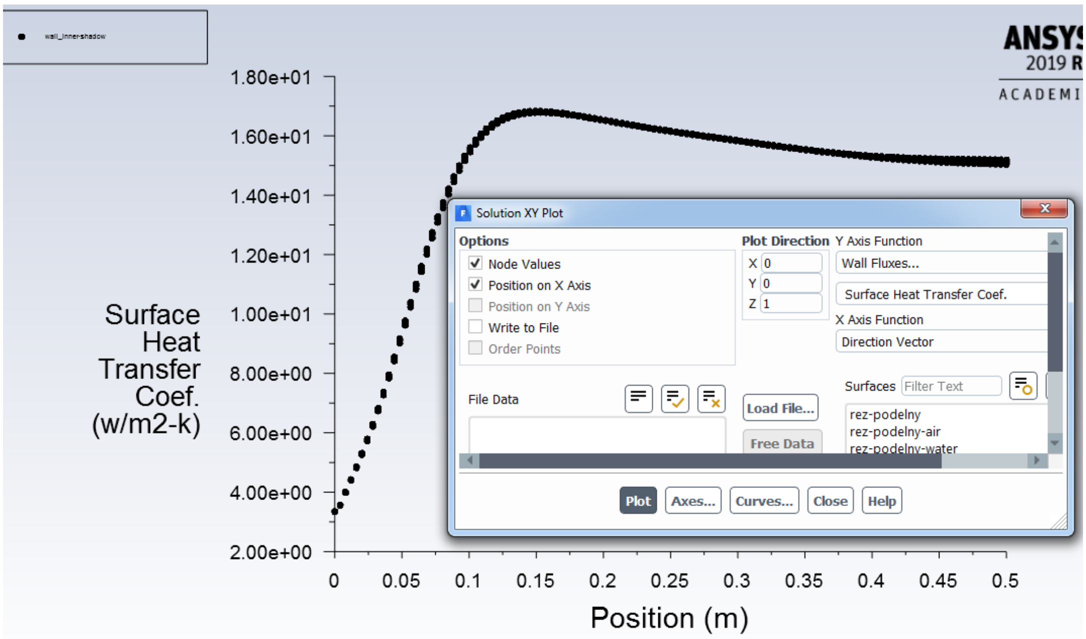


*Fig. 6.44 - Coefficient of heat transfer to air for the interface (**wall inner-shadow**) [W.m^{-2}.K^{-1}]*

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

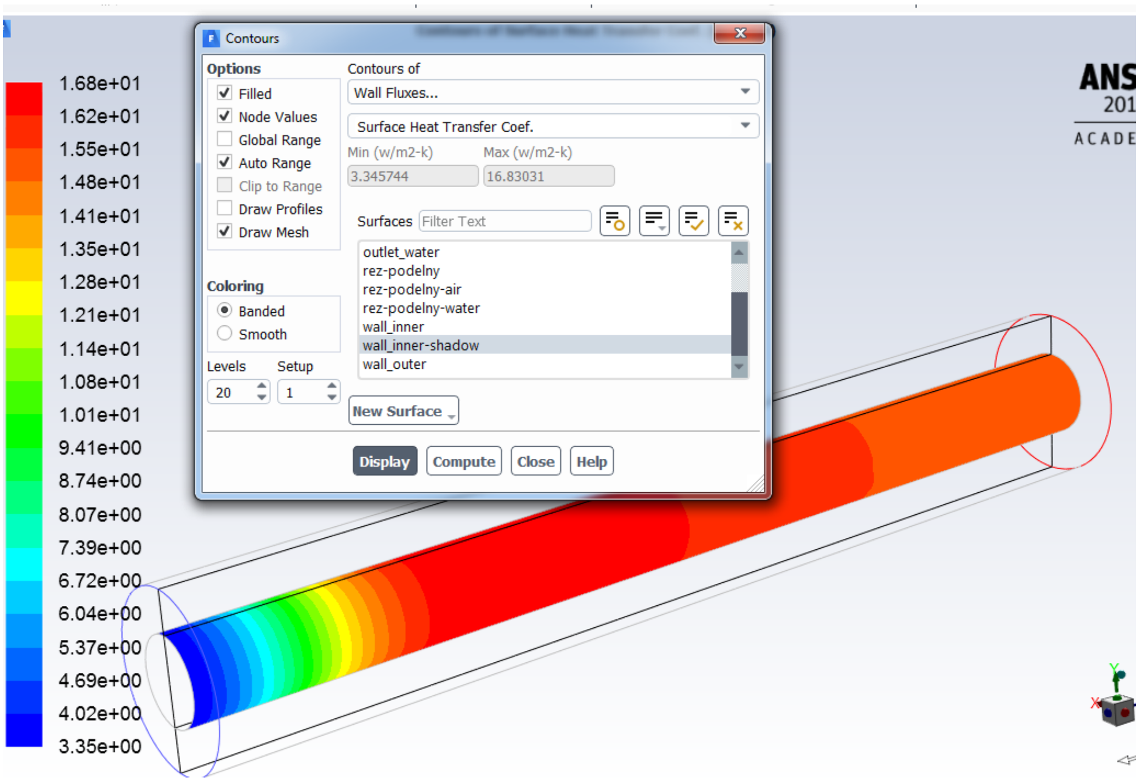


*Fig. 6.45 - Heat transfer coefficient for air for interface (**wall inner-shadow**) [W.m^{-2}.K^{-1}]*

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.

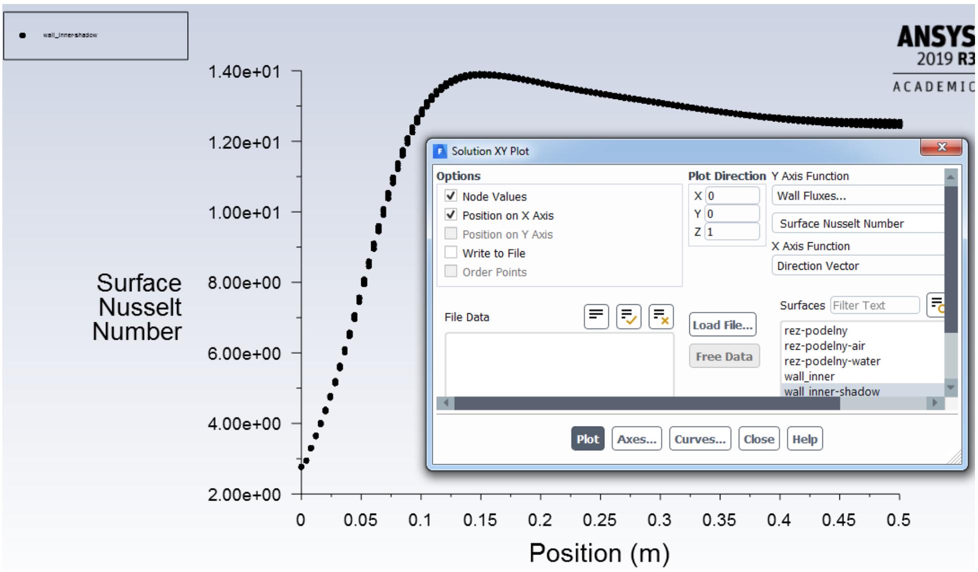


*Fig. 6.46 - Nusselt number evaluated on the interface wall (**wall inner-shadow**)*

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.

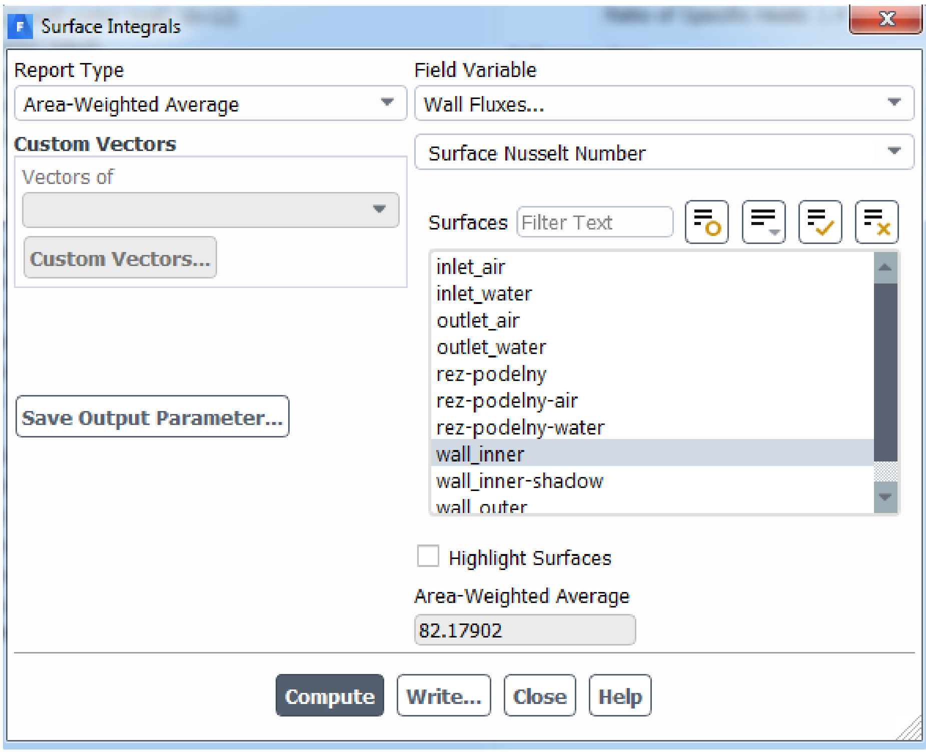


*Fig. 6.48 - Evaluation of the average value of the Nusselt number on the **wall inner** for the **water flow** area*

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.

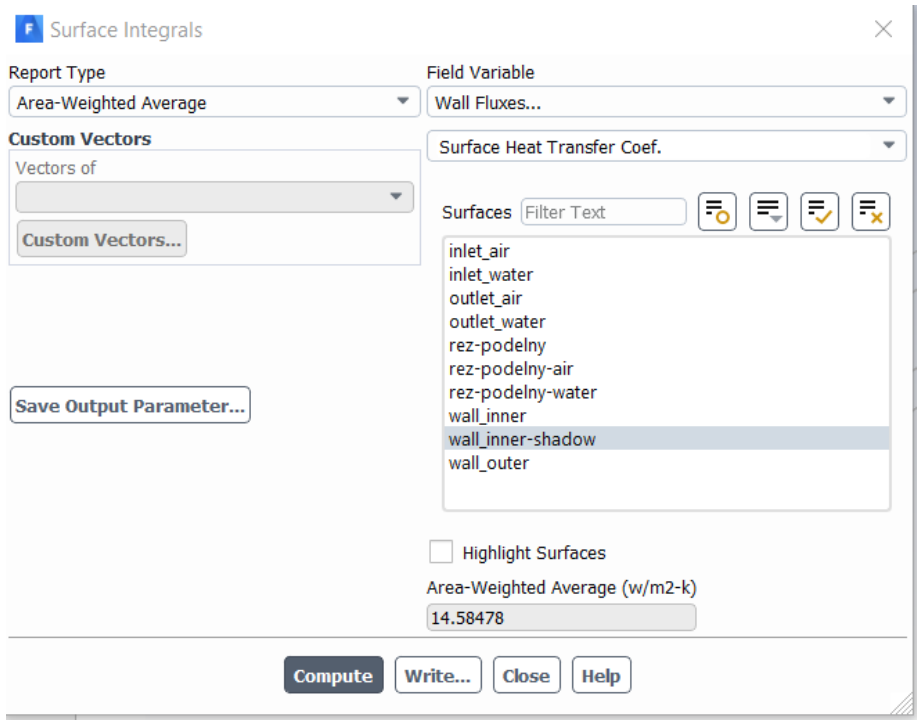


*Fig. 6.49 - Evaluation of the average value of the **Nusselt number** on the **wall inner-shadow** for the area of **air** flow*

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).

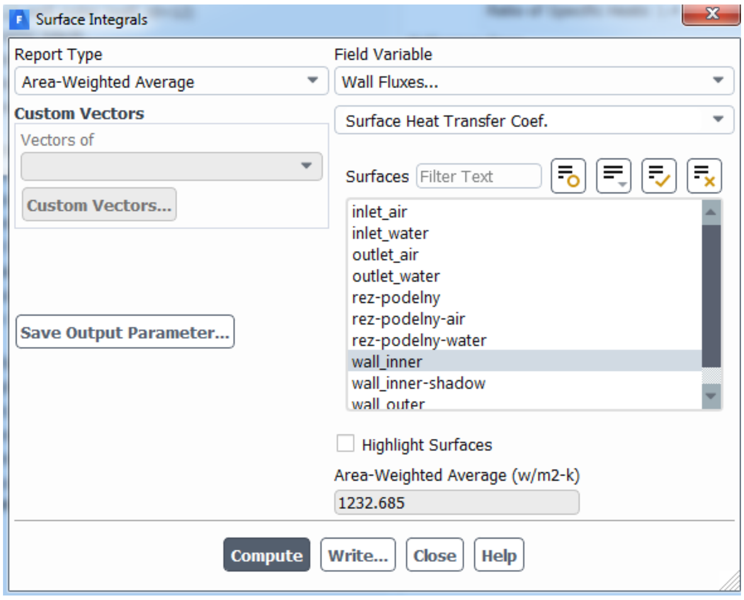


*Fig. 6.50 - Evaluation of the mean value of the **heat transfer coefficient** α on the **wall inner** for the **water** flow area*

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.

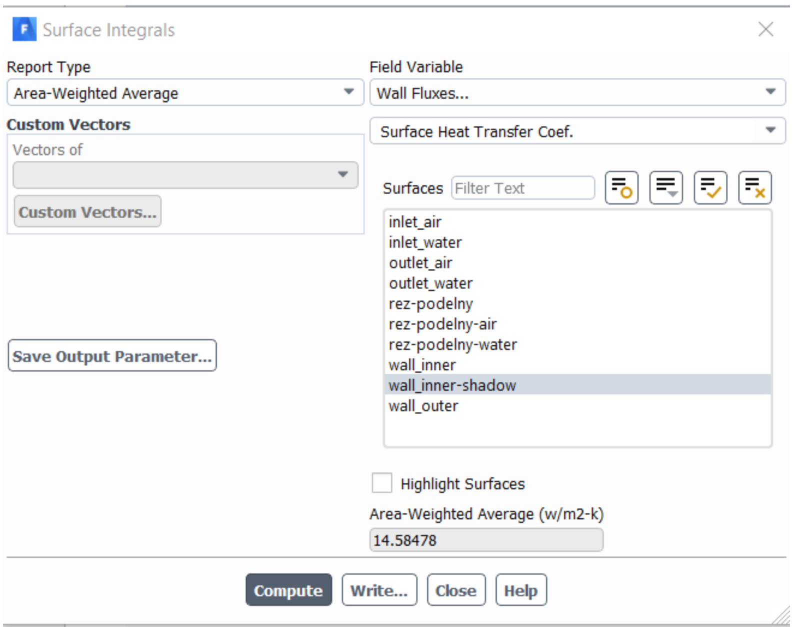


*Fig. 6.51 - Evaluation of the mean value of the **heat transfer coefficient** α on the **wall inner-shadow** for the **air** flow area*

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.

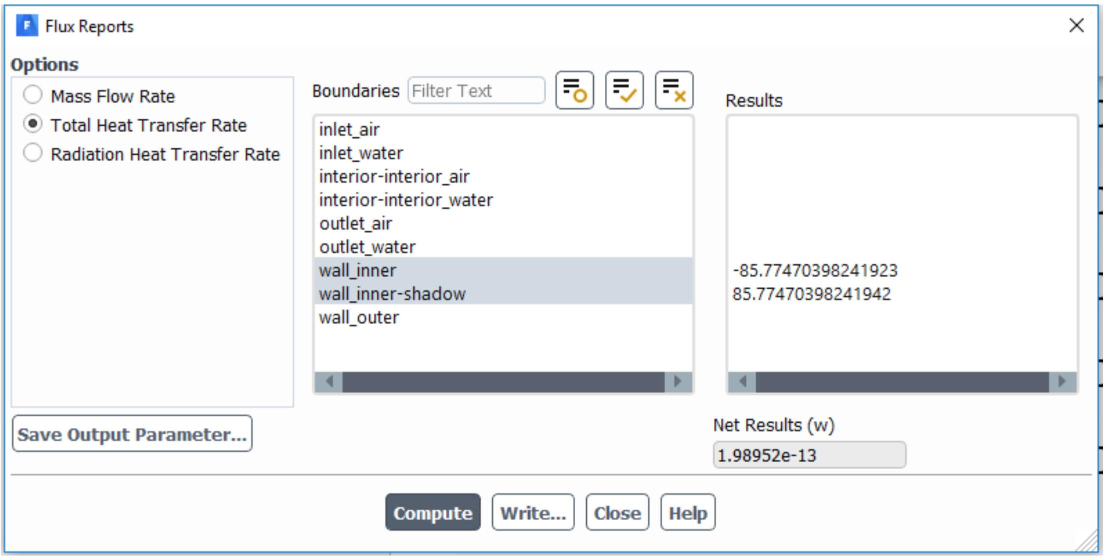
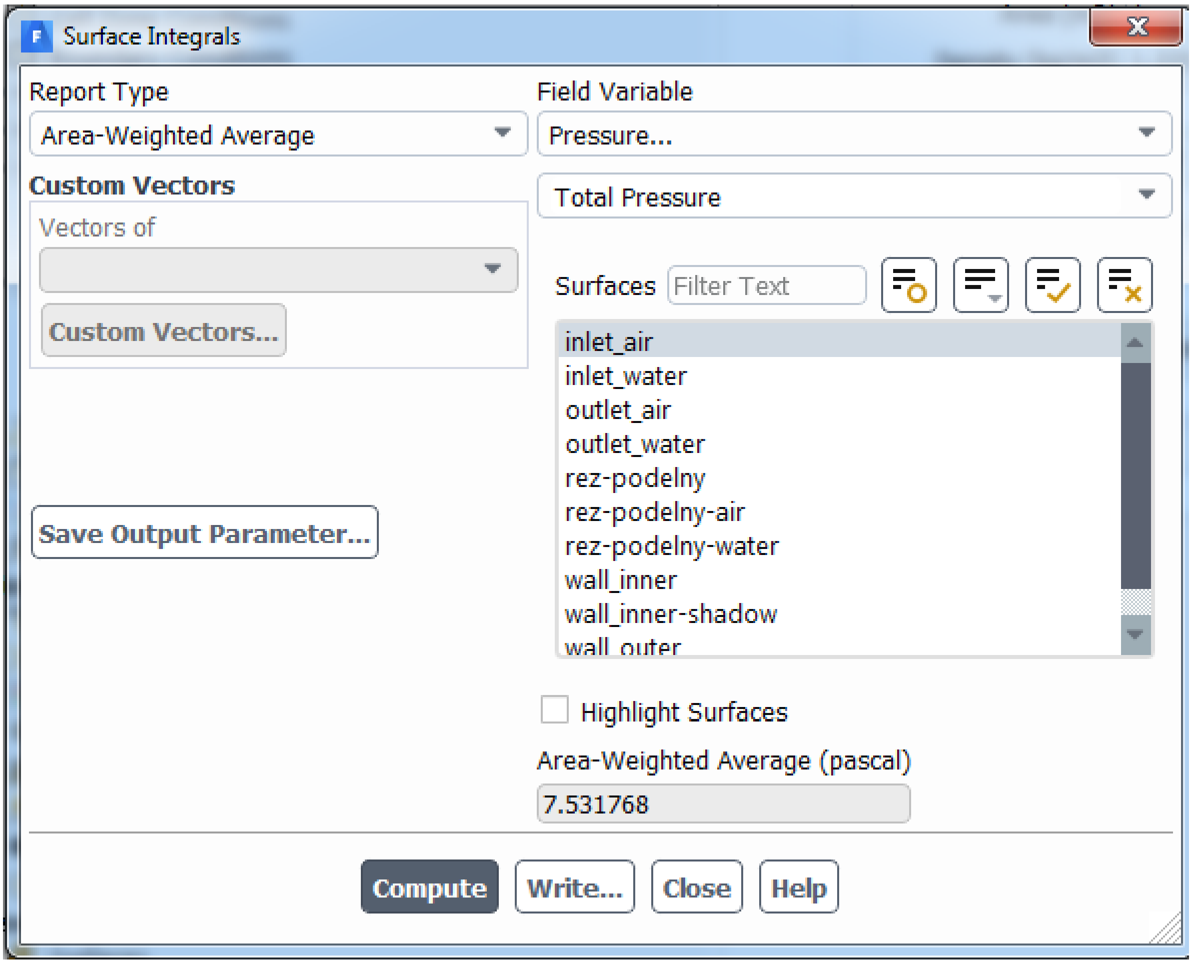


Fig. 6.52 - Heat flow evaluation ***P* [W]**

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}***).



obr. 6.2 – Evaluation of $\boldsymbol{p_{1tot}}$ *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:

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

Tab. 6.5 – Final comparison of average results

	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}]
P			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]

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.

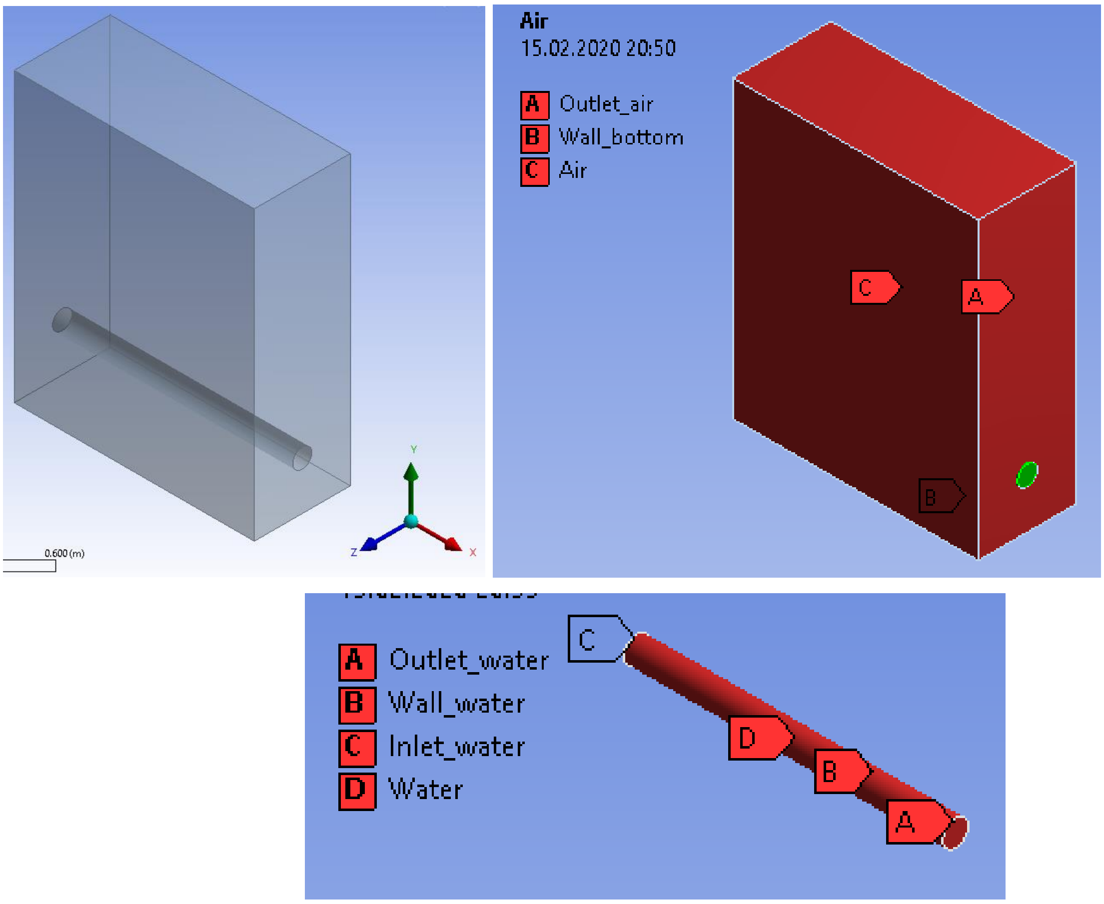


Fig. 7.1 - Geometry and boundary conditions.

Tab. 7.1 – Area dimensions

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

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.

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

Material	**Steel**	**Water**	**Air**	
density ρ	8030	998.2	1.225	[kg.m^{-3}]
specific heat capacity c_p	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. 7.3 – Boundary conditions

	Inlet water	**Outlet water**	**Wall water**	**Outlet air**	**Wall 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	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:

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

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 $\tilde{\tilde{\alpha}} = \frac{\mathrm{Nu}.\lambda}{d}$ [2].

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

$$\mathrm{Pr} = \frac{\rho \cdot c_p \cdot \nu}{\lambda} = \frac{998.2 \cdot 4182 \cdot 1.01e-6}{0.6} = 6.99 \tag{7.2}$$

$$Nu = 0{,}023 \cdot Re^{0{,}8} \cdot Pr^{0{,}3}$$

$$Nu = 0.023 \cdot 12000^{0{,}8} \cdot 6.99^{0.3} = 75.5 \tag{7.3}$$

Then the heat transfer coefficient is

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

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.

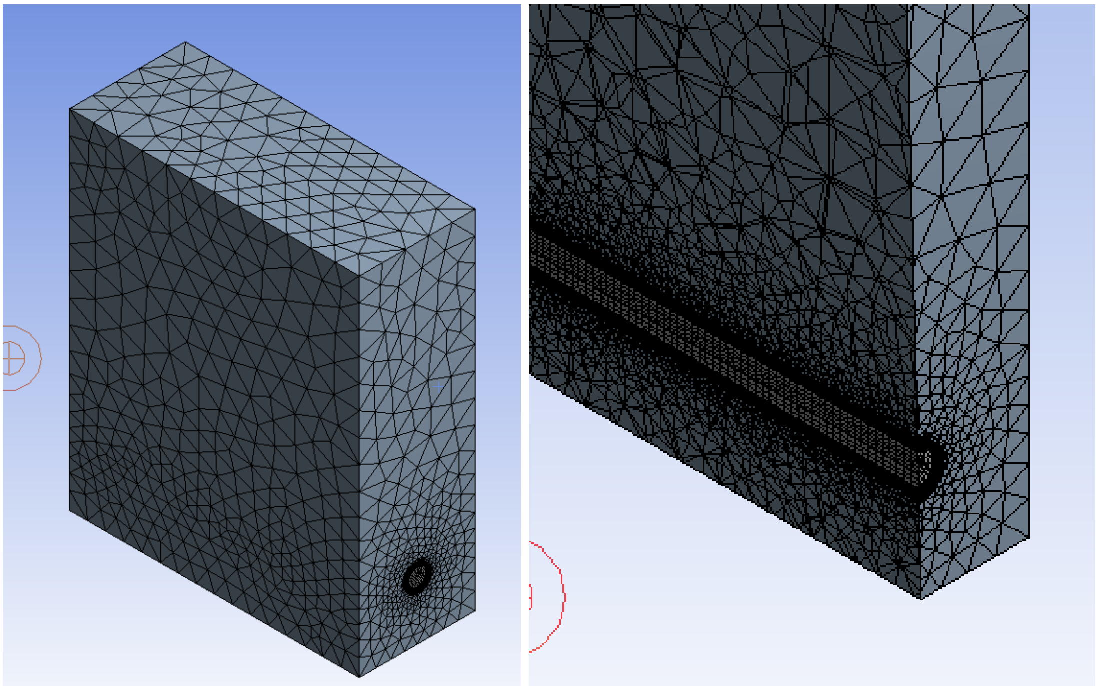


Fig. 7.2 - Surface mesh and detail with inflation.

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.

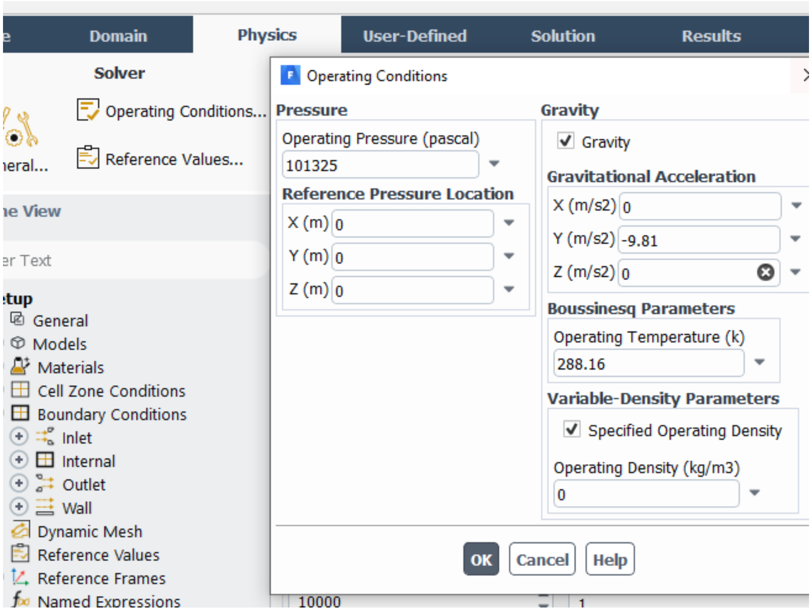


Fig. 7.3 Operating Conditions

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.

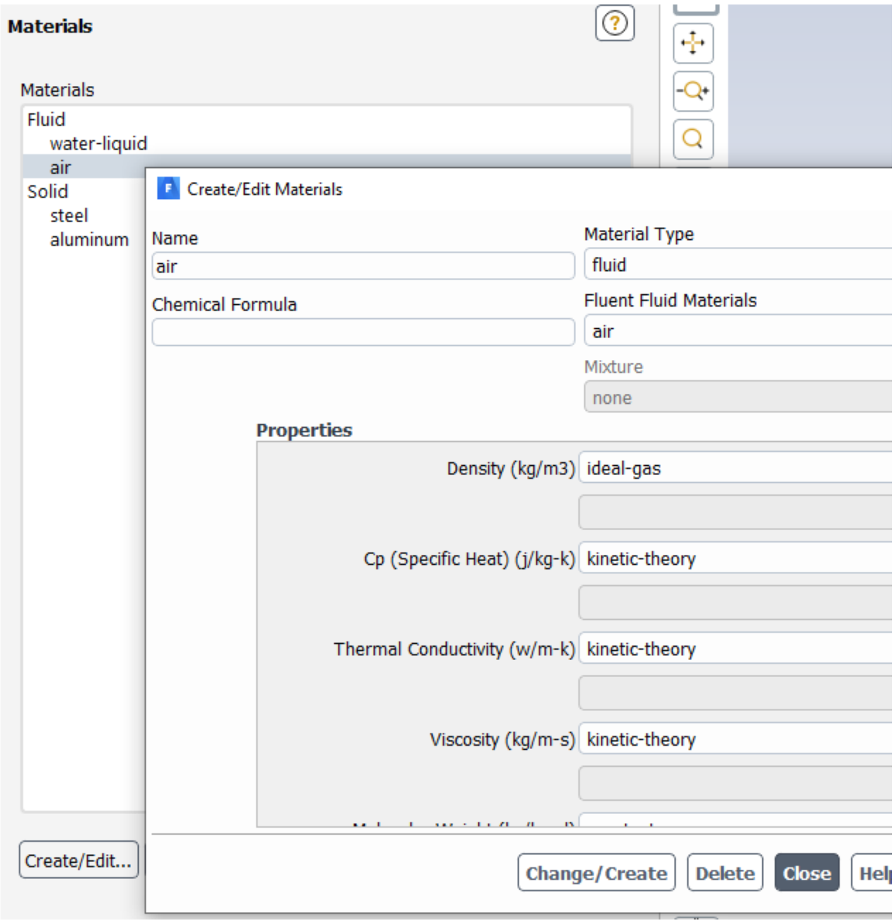


Fig. 7.4 Properties of Air

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.

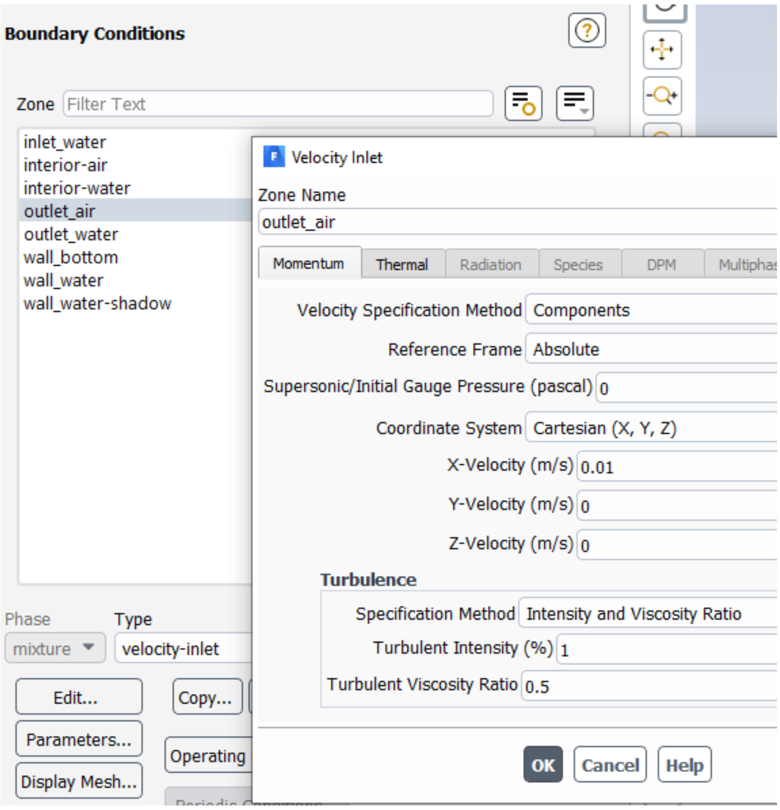


Fig. 7.5 Velocity inlet for air

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.

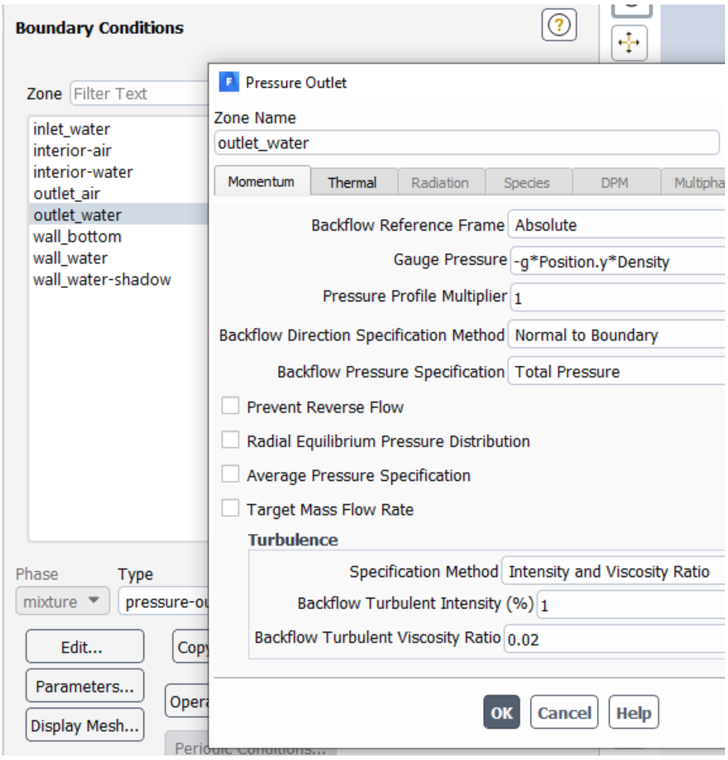


Fig. 7.6 Pressure outlet for water

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.

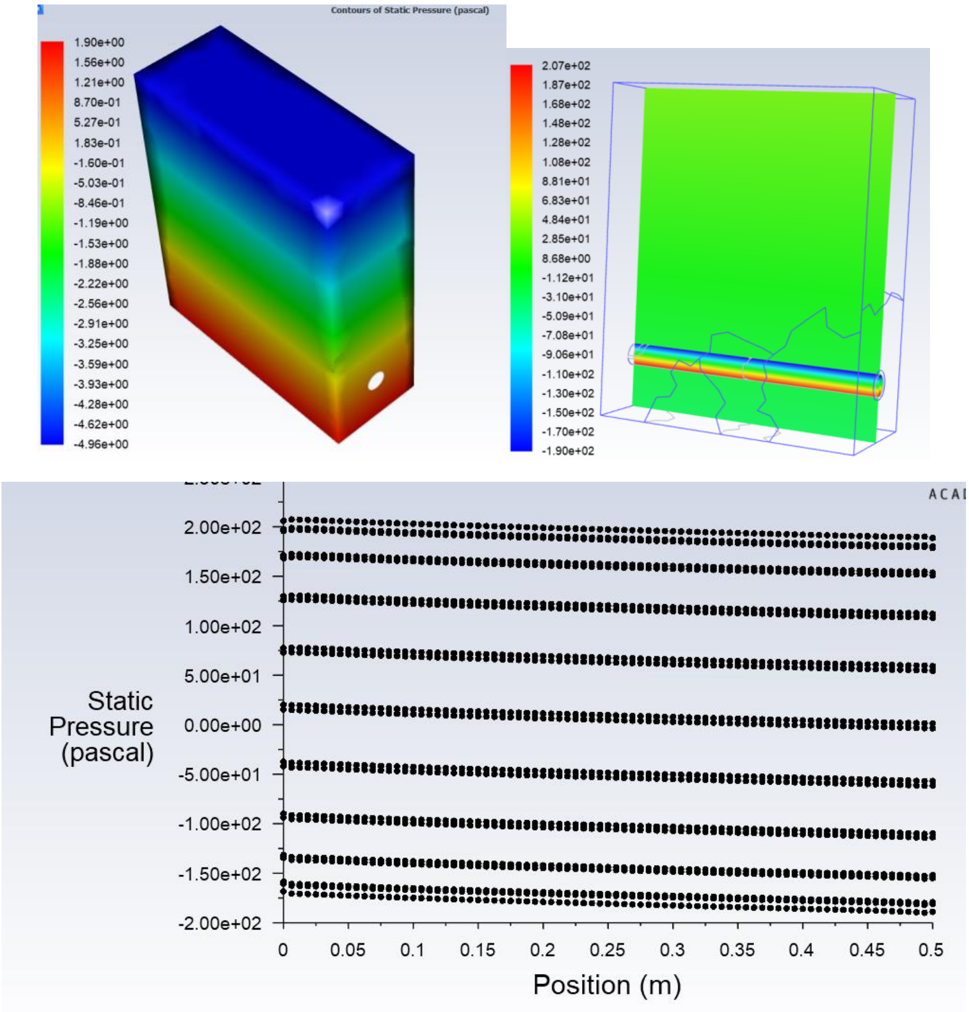


Fig. 7.8 - Contours of the hydrostatic pressure at the air / pipe boundary and pressure drop in the pipe (PLOT XY)).

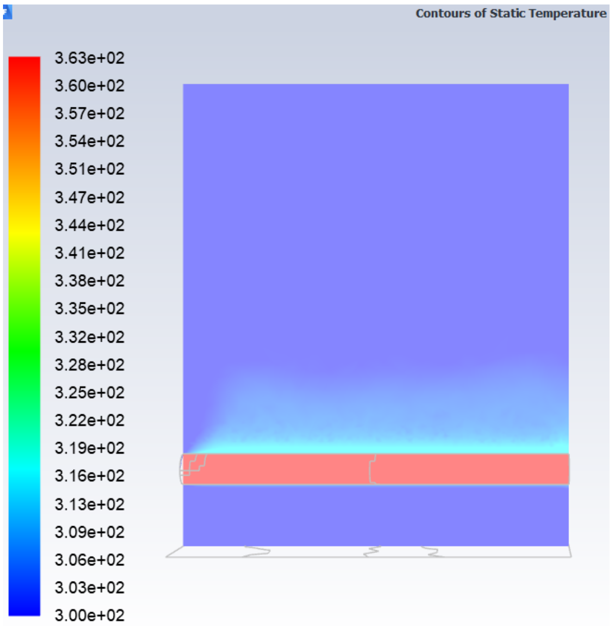


Fig. 7.9 - Temperature distribution in axial section of the area with air heating above the pipe.