
VSB – Technical university Ostrava



**Modeling of heat, mass and momentum
transfer**

textbook

Milada Kozubková

Tomáš Blejchař

Marian Bojko

Ostrava 2019

Content

Content	2
List of labels	5
1. The hypothesis of the continuum and the physical properties of materials.....	9
1.1. The hypothesis of the continuum	9
1.2. Methods for solving of heat, mass and momentum transfer.....	10
1.3. Properties of solids and liquids	11
1.4. Dimensionless criteria	16
2. Transfer and its solution	20
2.1. Definition of transfer	20
2.1.1. Convective transfer.....	20
2.1.2. Diffusion transfer	22
2.1.3. Total transfer	22
2.1.4. Balance transfer equation	22
2.1.5. Boundary conditions.....	23
2.2. Numerical methods of solutions	25
2.2.1. Difference method.....	25
2.2.2. Finite volume method.....	28
2.2.3. Geometry and computational grid generation	30
2.2.4. Choice of interpolation scheme	32
2.2.5. Convergence and residuals	32
2.2.6. Convergence acceleration.....	33
2.2.7. Relaxation.....	33
3. Heat transfer by conduction	35
3.1. Fourier equation	35
3.2. Equation of heat transfer by conduction.....	36
3.3. Boundary conditions.....	37
3.4. One-dimensional heat conduction	38
3.4.1. Analytical solution	38
3.4.2. Numerical solution	40
3.5. Temperature distribution at the transient transfer	46
4. Mass and momentum transfer	49
4.1. Continuity equation	49
4.2. Navier-Stokes (momentum) equation.....	49
4.3. Conditions on flow boundaries and walls	52
5. Conduction and convection at laminar flow.....	55
5.1. Energy equation.....	55
5.2. Mathematical model of laminar flow with conduction and convection	56
5.3. Boundary conditions at laminar flow with conduction and convection	56
5.3.1. Boundary conditions on thin wall.....	59
5.3.2. Boundary condition on two – sided thin wall.....	60
5.4. Heat transfer coefficient	61
5.5. Heat transfer at flow around plate	61
6. Turbulence.....	70
6.1. Reynolds time averaging	72
6.2. k- ϵ two-equation model of turbulence.....	73
6.3. Boundary conditions for k- ϵ turbulent model	75
6.3.1. Turbulent variables on boundary.....	75
6.3.2. Variants of turbulent boundary conditions	76
6.3.3. Boundary conditions.....	77
6.4. Wall function, possibility of more accurate calculation	78
6.5. Influence of mesh quality on the choice of wall functions for various models of turbulence....	81

6.6. Choice of turbulent model for more accurate calculation	81
6.7. Flow around the tube in transverse direction	82
6.7.1. Flow around the tube - theory, measurement	82
6.7.2. Flow around the tube - numerical solution.....	85
6.7.3. Flow around two tubes	87
7. Conduction and convection in turbulent flow	91
7.1. Energy equation at turbulent flow	91
7.2. Mathematical model of turbulent flow with convection and conduction.....	91
7.3. Boundary conditions.....	91
7.4. Heat transfer at turbulent flow around the plate.....	93
7.5. Flow around the tube with heat transfer (without flow inside)	98
7.6. Flow around the tube with the heat transfer (with the flow inside).....	106
7.7. Flow across the tube bundle with heat transfer	106
7.7.1. The arrangement of the tube bundle in a row	109
7.7.2. The arrangement of the tube bundle to cross.....	115
8. Analysis of the heat exchangers	121
8.1. Basic types of heat exchangers and their description.....	122
8.1.1. Exchanger of fluid-fluid tubular type	122
8.1.2. Finned heat exchangers	125
8.1.3. Plate heat exchangers	127
8.2. Thermal power and pressure drop of heat exchanger.....	129
8.2.1. Thermal power	129
8.2.2. Pressure drop	132
8.3. Methods for heat calculation of heat exchanger.....	133
8.3.1. Method ϵ -NTU	133
8.3.2. Method P-NTU.....	134
8.3.3. Method MTD.....	134
8.4. Solution of co-flow and counter-flow heat exchanger	137
8.4.1. Physical properties of gases (kinetic theory).....	137
8.4.2. Co-flow heat exchanger water-water.....	138
8.4.3. Counter-flow heat exchanger water-water	144
8.4.4. Co-flow heat exchanger water-air	148
8.4.5. Co-flow heat exchanger air-water-air.....	152
9. Time dependent solution	155
9.1. Discretization of time-dependent equation.....	156
9.2. Boundary conditions.....	158
9.2.1. Table for time-dependent boundary condition	158
9.2.2. UDF for boundary condition	160
9.3. Time-dependent tasks, evaluation	160
10. Optimization (Adjoint solver)	165
10.1. Using the adjoint solver.....	166
10.2. Limitations of adjoint solver	166
10.3. Application	167
10.3.1. Minimizing the pressure drop in 180° elbow.....	167
11. Multiphase flow.....	171
11.1. Specification of multiphase materials	171
11.2. Mixture model.....	173
11.2.1. Continuity equation for mixture	173
11.2.2. Momentum equation.....	174
11.2.3. Mixture flow behind the step.....	174
12. Cavitation	177
12.1. Rayleigh -Plesset thory.....	177
12.2. Schnerr and Sauer cavitation model	178
12.3. Cavitation in the nozzle.....	179

12.3.1. Description of experiment	179
12.3.2. Results	180
13. Appendix	182
13.1. Vectors and scalars	182
13.2. Coordinate systems.....	184
13.3. Field of velocity and acceleration.....	185
14. Literature	186

List of labels

Note: label, for which there is not defined dimensions, represents a general variable.

a	temperature conductivity	$[\text{m}^2 \cdot \text{s}^{-1}]$
\vec{a}	general vector	
A, S	area	$[\text{m}^2]$
A, A_i	constant	[1]
C_D	constant	[1]
C_v	empirical constant	[1]
C_μ	constant	[1]
C_1, C_2, C	constant	[1]
c_v	specific heat capacity at constant volume	$[\text{J} \cdot \text{kg}^{-1} \cdot \text{K}^{-1}]$
c_p	specific heat capacity at constant pressure	$[\text{J} \cdot \text{kg}^{-1} \cdot \text{K}^{-1}]$
d_h	hydraulic diameter	[m]
f	frekvence	$[\text{s}^{-1}]$
f	constant	[1]
\vec{F}	force	[N]
E	specific energy	$[\text{J} \cdot \text{kg}^{-1}]$
E	empirical constant	[1]
Gr	Grashof number	[1]
g	acceleration of gravity	$[\text{m} \cdot \text{s}^{-2}]$
h	static entalpy	$[\text{J} \cdot \text{kg}^{-1}]$
h	height	[m]
l	turbulent intensity	[%]
k	turbulent kinetic energy	$[\text{m}^2 \cdot \text{s}^{-2}]$
k_p	turbulent kinetic energy in logarithmic layer	$[\text{m}^2 \cdot \text{s}^{-2}]$
k	heat transfer coefficient	$[\text{W} \cdot \text{m}^{-2} \cdot \text{K}^{-1}]$
L, l	length	[m]
m	mass	[kg]
Ma	Mach number	[1]
M	molecular weight	$[\text{kg} \cdot \text{kmol}^{-1}]$

\vec{n}	vector of outer normal direction	[1]
Nu	Nusselt number	[1]
p	pressure	[Pa]
p_{op}	operating pressure	[Pa]
p_s	static pressure	[Pa]
P_1, P_2	thermal efficiency	[1]
Pr	molecular Prandtl number	[1]
Pr_t, σ_h	turbulent Prandtl number	[1]
q	heat flux	[J·m ⁻² ·s ⁻¹]
Q	heat	[kcal, J]
Q_V	volume flow rate	[m ³ ·s ⁻¹]
Q_m	mass flow rate	[kg·s ⁻¹]
r	specific gas constant	[J·kg ⁻¹ ·K ⁻¹]
R	univerzal gas constant	[J·kmol ⁻¹ ·K ⁻¹]
R	temperature resistance	[1]
R	rezidual	
\bar{R}	normalized rezidual	[1]
Re	Reynolds number	[1]
Sc	Schmidt number	[1]
Sh	Strouhal number	[1]
t	time	[s]
t	temperature	[°C]
T	absolut temperature	[K]
\vec{u}	vector of velocity	[m·s ⁻¹]
u	mean velocity	[m·s ⁻¹]
u_i	i-th velocity component	[m·s ⁻¹]
\bar{u}_i	i-th mean velocity component	[m·s ⁻¹]
u'_i	i- th fluctuating velocity component	[m·s ⁻¹]
u^+, u^*	velocity defined by wall function	[m·s ⁻¹]
u_τ	shear velocity	[m·s ⁻¹]
U	inner energy	[J·kg ⁻¹]

v	mean velocity	$[\text{m}\cdot\text{s}^{-1}]$
\vec{v}	vector of velocity	$[\text{m}\cdot\text{s}^{-1}]$
V	volume	$[\text{m}^3]$
x_i	coordinate in Cartesian system $[x_1, x_2, x_3]$ or $[x, y, z]$	$[\text{m}]$
y	perpendicular distance from the wall	$[\text{m}]$
y^+, y^*	dimensionless distance of the first cell from the wall	$[1]$
y_v^*	dimensionless thickness of the sublayer	$[1]$
y_v	thickness of the viscous sublayer	$[\text{m}]$
Δy_P	distance of point P from the wall in normal direction	$[\text{m}]$
α	relaxing factor	$[1]$
α	heat transfer coefficient	$[\text{W}\cdot\text{m}^{-2}]$
$\tilde{\alpha}$	estimation of heat transfer coefficient	$[\text{W}\cdot\text{m}^{-2}]$
β	coefficient of thermal expansion	$[\text{K}^{-1}]$
$\overset{=}{\delta}$	Kronecker delta-tenzor	$[1]$
ε	efficiency	$[1]$
ε	rate of dissipation	$[\text{m}^2\cdot\text{s}^{-3}]$
ε_P	dissipation rate in the logarithmic layer	$[\text{m}^2\cdot\text{s}^{-3}]$
Γ	transfer	
γ	coefficient	$[1]$
κ	von Kármán constant, poměr měrných tepelných kapacit [1]	
λ	ratio of specific heat capacities	$[\text{W}\cdot\text{m}^{-1}\cdot\text{K}^{-1}]$
η	dynamic viscosity	$[\text{Pa}\cdot\text{s}]$
μ	dynamic viscosity	$[\text{Pa}\cdot\text{s}]$
μ_t	turbulent viscosity	$[\text{Pa}\cdot\text{s}]$
ν	kinematic viscosity	$[\text{m}^2\cdot\text{s}^{-1}]$
ν_t	turbulent kinematic viscosity	$[\text{m}^2\cdot\text{s}^{-1}]$
$\overset{=}{\Pi}$	total stress tensor	$[\text{Pa}]$
ρ	density	$[\text{kg}\cdot\text{m}^{-3}]$
σ_k	empirical constant	$[1]$
σ_ε	empirical constant	$[1]$

σ_h	turbulent Prandtl number	[1]
τ	time period	[s]
$\bar{\tau}$	viscous stress tensor	[Pa]
τ	stress	[Pa]
τ_w	viscous stress on the wall	[Pa]
τ_t	turbulent stress	[Pa]
ζ	general variable	
ζ'	fluctuations in general variable	
$\bar{\zeta}$	mean value of general variable	

Indexes:

i	index of the velocity component
i	summation index
C	Einstein summation index
w, e, p	index of wall of the final volume
W, E, P, N, S, F, B, NB	
ref	index of reference values
$wall$	index of the wall
c	(cool) cold, heated
h	(heat) warm, cooled
o	mass
i, n	index of iteration
I	input
O	output
P	index of cell
P	planar
s	inertial
S	wall
$stat$	static
tot	total
L	index of raw
T	index of column

1. The hypothesis of the continuum and the physical properties of materials

Fundamentals of thermomechanics is to investigate the heat transfer between a given system with its surroundings. This interaction is called the work and heat. However thermomechanics must deal with the process, during which the heat transfer takes place depending on changing conditions and time. Thus, we will consider not only the heat transfer and its resulting effect, but also the transmission rate.

What is heat transfer? Heat transfer is a change in thermal energy due to the existence of temperature difference. However, the temperature difference exists within a single medium (media) or between multiple media. We discuss three types of heat transfer [2]:

- conduction that occurs in the solid or stationary fluid with a temperature gradient (gradient)
- convection, defined between the solid surface and the flowing fluid, if they have different temperatures
- radiation arising between the surfaces which emit energy in the form of electromagnetic waves,

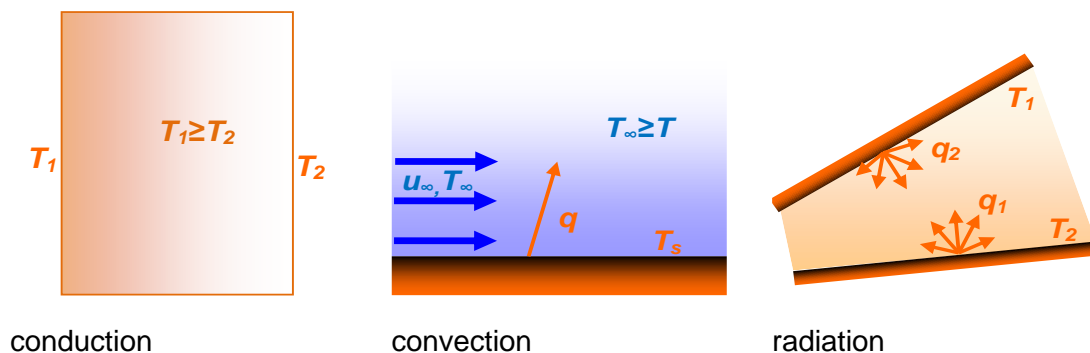


Fig. 1.1 Conduction, convection a radiation [2]

In a complex context, it is necessary to consider not only the heat transfer, but also mass and momentum transfer, that deal with the flow of gases and liquids (fluids).

1.1. The hypothesis of the continuum

Each substance consists of molecules that exist in the environment, they may also move. But this environment is not considered as a discrete environment at the molecular level. Thus, it has a molecular structure, but it is not always optimal to include this molecular structure of the model. The intentional release of the molecular structure is known as a

hypothesis of continuum, when the molecular structure of fluid is replaced by a set of properties such as density, pressure, temperature and velocity [4], defined in fluid points (very small volume) and varies continuously in the transition from one to another volume. These properties are therefore described by continuous functions of position and time. It has been proven that this approach can replace the problem at a molecular level in a certain sense.

Similarly, as in the general mechanics the concept of a mass point is introduced, the concept of "elementary volume of fluid and solid" performs in the tasks of transfer. It is a volume very small compared to the dimensions of the liquid stream but sufficiently large with respect to the mean length of the free path of the molecule. It can be assumed, therefore, that the statistical mean values of the kinetic theory apply to the number of molecules contained in this volume.

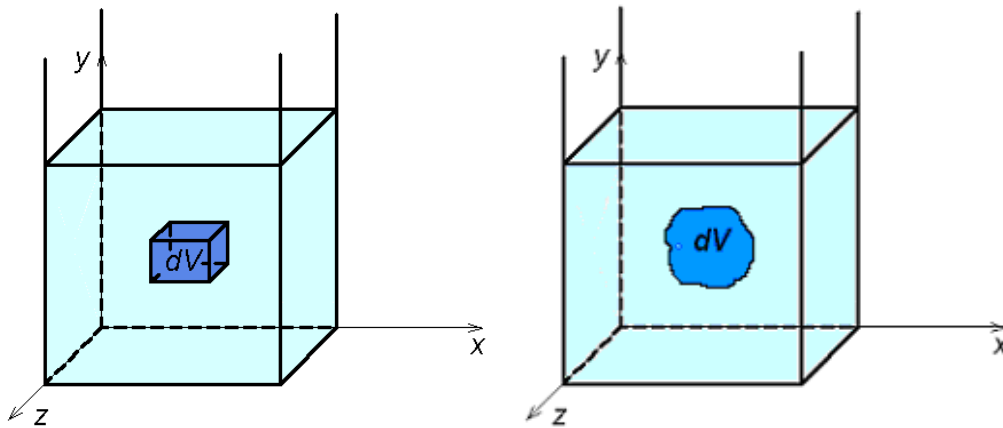


Fig. 1.2 Elementary volume of fluid [11]

For this "elementary volume" the equilibrium conditions of forces and energy will be derived and the basic laws will be defined, i.e. the law of mass and energy conservation.

1.2. Methods for solving of heat, mass and momentum transfer

Basic laws of mass, momentum and energy conservation are described by partial differential equations together with the boundary and initial conditions. Their analytical solution is very difficult and is only possible for several significantly simplified applications. Therefore, numerical methods are currently used on a large scale.

Numerical modeling of many physical phenomena is closely linked to modeling a form of motion mathematically. Movement of fluids is associated with the solution of various problems of the physical model:

- laminar and turbulent flow in both simple and complex geometries
- compressible and incompressible flow
- steady, unsteady and transient flow

- heat transfer, natural and mixed convection, radiation
- transfer of chemical admixtures, including chemical reactions
- multiphase flow, free surface flow, flow with solid particles, bubbles, respectively. drops
- combustion and chemical reactions
- porous flow, etc.

Because phenomena are generally three-dimensional and time-dependent, they are described by a set of partial differential equations that must be solved by numerical methods. The flow solution is possible to use commercial software systems CFD (Computational Fluid Dynamics) program systems, such as Ansys-Fluent, Ansys-CFX, Fidap, Flow 3D, Rampant, Fluidyn-Panache, and others. The task of the user is to build the correct calculation model, which contains some mathematical, physical and technical principles. For such a model it is necessary to find all input data in the existing standards, to build the input data for a program that can solve computational model, solving by the terminal, correctly interpret the results for further use in all phases and carry out effective monitoring of all inputs and outputs. The user must safely divide all the information on the geometric data (two-dimensional or three-dimensional features, topology), data on the effect of external forces and physical data (information about flowing media, its physical properties). Thus, an essential task is to know hydromechanics, thermodynamics and other sciences to the complexity of the problem.

As regards the computational methods underlying the use of the program, the designer should know the methods principle for reliable use in standard cases. For program ANSYS Fluent or CFX it is a need to know in what shape the final volume will work, it follows the choice of network density, approximation schemes, the nature of the time dependence of the quantities and the resulting time step size, etc.

1.3. Properties of solids and liquids

State of substances found in equilibrium can be determined by **density, temperature, pressure and velocity**.

Density ρ is equal to the ratio of mass of elemental particles of a substance dm to its elementary volume dV

$$\rho = \frac{dm}{dV} \quad [\text{kg}\cdot\text{m}^{-3}] \quad (1.3.1)$$

Temperature T is a variable that gives information about internal energy of the substance. It is expressed in degrees of Celsius or Kelvin.

$$T[\text{K}] = t[^\circ\text{C}] + 273.15 \quad (1.3.2)$$

The temperature change of substances is often associated with convection or conduction of heat.

The density of solids and liquids varies with pressure and temperature only slightly and most of the calculations will be considered as constant $\rho = konst$. Yet the liquids have the ability to reduce their fluid volume at increasing pressure and therefore their **bulk compressibility** can be defined. **Thermal expansion** [11] is the ability of a substance increasing its volume when heated. It is expressed as a coefficient of thermal expansion β

$$\beta = \frac{1}{V} \left(\frac{\Delta V}{\Delta t} \right)_{p=konst} \quad [^{\circ}\text{C}^{-1}] \quad (1.3.3)$$

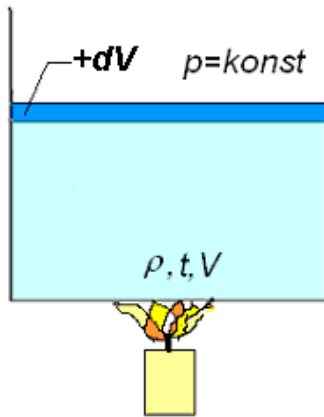


Fig. 1.3 Thermal expansion of liquids

Let at the beginning the liquid of density ρ and the volume V is placed in the container, see Fig. 1.3. After heating the liquid, its temperature is higher by Δt and the liquid occupies a volume $V_0 = V + \Delta V$. The volume, temperature and density of the liquid after heating are V_0, t_0, ρ_0 . Substituting the volume and temperature difference after heating and before heating in equation (1.3.4) it gives equation (1.3.4), which expresses the change of liquid volume $\Delta V = V_0 - V$ per unit of original volume when the temperature change is $\Delta t = (t_0 - t)$.

$$\beta = \frac{1}{V} \frac{V_0 - V}{t_0 - t} = \frac{\Delta V}{V \Delta t} \quad [^{\circ}\text{C}^{-1}] \quad (1.3.4)$$

The previous equation shows the relationship of the volume of liquid when heated

$$V_0 = V + \Delta V = V + \beta V \Delta t = V(1 + \beta \Delta t) \quad [\text{m}^3] \quad (1.3.5)$$

The density after heating is given by the following equation

$$\rho_0 = \frac{m}{V_0} = \frac{m}{V(1 + \beta \Delta t)} = \frac{\rho}{(1 + \beta \Delta t)} \quad [\text{kg} \cdot \text{m}^{-3}] \quad (1.3.6)$$

The liquid **pressure** is determined by the amplitude of the pressure force acting perpendicularly to the unit area. When a pressure force is uniformly distributed, the pressure is given by the ratio of the amplitude force and the area.

$$p = \frac{|\vec{F}|}{S} \quad \text{resp.} \quad p = \frac{d|\vec{F}|}{dS} \quad [\text{Pa}] \quad (1.3.7)$$

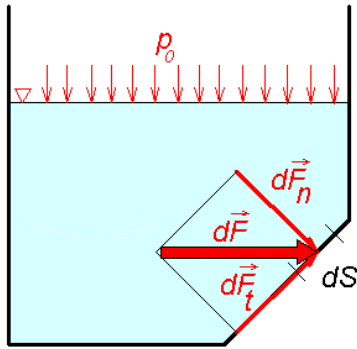


Fig. 1.4 Effect of pressure forces on the wall of the container

Pressure force in hydrostatics always acts perpendicular to the surface. This statement can be proved by negation, see Fig. 1.4. If the force $d\vec{F}$ was applied to area dS in non-normal direction, it could be broken down into normal and tangential component. The tangential component of the force would require the movement of liquid particles, that do not resist to relative movement. Because the fluid is at rest, the tangential component is zero and the pressure force must act in the direction normal to the surface.

The density of gases and vapors is a function of state variables, i.e. pressure p and temperature T [K]. For its calculation there will be use a simple **equation of state** of ideal gas

$$\rho V = mrT \Rightarrow \frac{\rho}{\rho} = rT \quad (1.3.8)$$

where r is the specific gas constant [$\text{J}\cdot\text{kg}^{-1}\cdot\text{K}^{-1}$], whose value depends on the type of gas.

The viscosity of fluid is reflected by the movement of real fluids. If the adjacent layers of fluid move at different velocity, their interface shear stress arises and it prevents movement. Slower layer is accelerated and in turn faster layer being held back. Tangential (shear) stress is caused by internal friction or viscosity of the fluid. It is proportional to the change in velocity in the direction perpendicular to the direction of motion according to Newton's relationship

$$\tau = \eta \frac{dv}{dy} \quad [\text{Pa}] \quad (1.3.9)$$

where η is dynamic viscosity and $\frac{dv}{dy}$ is gradient of velocity in the direction perpendicular to the moving direction, see **Fig. 1.5**. This formula was formulated in 1687 by English physicist Isaac Newton for laminar flow. Shear stress causes angular deformation of the elementary volume of fluid (Fig. 1.5).

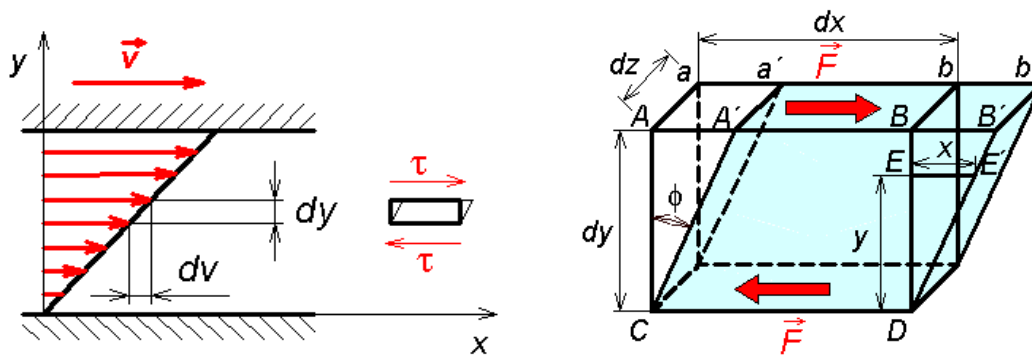


Fig. 1.5 Shear stress under laminar flow [11]

The unit of dynamic viscosity η is defined by the equation for shear stress

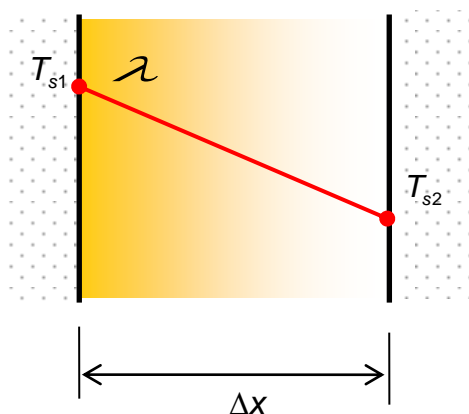
$$[\eta] = \frac{[\tau][y]}{[v]} = \frac{\text{N} \cdot \text{s}}{\text{m}^2} = \frac{\text{kg}}{\text{m} \cdot \text{s}} = \text{Pa} \cdot \text{s}$$

Kinematic viscosity is calculated by dividing the dynamic viscosity and density according to the relation

$$\nu = \frac{\eta}{\rho} \quad [\nu] = \frac{\text{kg} \cdot \text{m}^3}{\text{m} \cdot \text{s} \cdot \text{kg}} = \text{m}^2 \cdot \text{s}^{-1} \quad (1.3.10)$$

The dimension of kinematic viscosity does not contain units of weight and force. In practice, the still important unit of kinematic viscosity in the technical system is Stokes for which applies $1\text{S} = \text{cm}^2 \cdot \text{s}^{-1} = 10^{-4} \text{m}^2 \cdot \text{s}^{-1}$.

Heat Q [J] (incorrectly used term the thermal energy) [12], [2] is part of the inner energy which system exchanges (i.e. to accept or surrender) in contact with another system, without causing to produce work. The heat exchange between systems per unit time defines the thermal power P [$\text{J} \cdot \text{s}^{-1} = \text{W}$]. The heat passing through a surface determines so called **heat flux**. The density of heat flux (specific heat flux) is the amount of heat that passes through the surface per unit time.



The basic law of heat distribution is known Fourier's law, which gives the relationship between density of heat flux q and temperature gradient $\text{grad } T$:

$$\vec{q} = \frac{dQ}{dS dt} = \frac{dP}{dS} = -\lambda \nabla T \quad (1.3.11)$$

$$[\text{J} \cdot \text{s}^{-1} \cdot \text{m}^{-2} = \text{W} \cdot \text{m}^{-2}]$$

where λ [$\text{W} \cdot \text{m}^{-1} \cdot \text{K}^{-1}$] is thermal conductivity, which

Fig. 1.6 The principle of heat conduction

depends on type of material and temperature. The negative sign on the right side of the equation expresses the fact that the density of heat flux and temperature gradient as vectors have opposite sense (heat propagates in the direction of decreasing temperature).

The **specific heat** (heat capacity) is then defined as the amount of heat required to increase the temperature by 1 °C of substance amount 1 kg.

$$c = \frac{dQ}{m dT} \quad [\text{J}\cdot\text{kg}^{-1}\cdot\text{K}^{-1}] \quad (1.3.12)$$

In heat transfer by conduction the **temperature conductivity** is defined by equation

$$a = \frac{\lambda}{c_p \rho} \quad [\text{m}^2\cdot\text{s}^{-1}] \quad (1.3.13)$$

Heat transfer coefficient is defined by the equation

$$\alpha = \frac{q}{T_{wall} - T_{ref}} \quad [\text{W}\cdot\text{m}^{-2}\cdot\text{K}^{-1}] \quad (1.3.14)$$

where q je convective heat flux, T_{wall} is wall temperature and T_{ref} is reference temperature, which should be representative of the problem.

The heat transfer through the planar wall in fluid flow

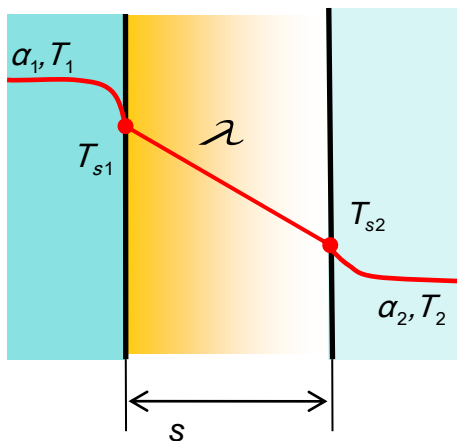


Fig. 1.7 Heat transfer through the wall

The simplest case of heat transfer is stationary heat transfer through a homogeneous isotropic unlimited planar wall [3]. The condition is, that the fluid surrounding wall on both sides, does not move significantly and thus avoid the convective heat transfer. To calculate the heat flux density in this case, the fundamental relationship holds:

$$q = \frac{1}{\frac{1}{\alpha_1} + \frac{s}{\lambda} + \frac{1}{\alpha_2}} (T_2 - T_1) = k(T_2 - T_1) \quad (1.3.15)$$

where α_1 and α_2 represent heat transfer coefficient at the interface of boundary walls and fluid, T_1 and T_2 represent the temperature of both the fluid surrounding the walls and s is the wall thickness. This method can not be used for composite walls. **The heat transfer**

coefficient k [$\text{W}\cdot\text{m}^{-2}\cdot\text{K}^{-1}$] characterizes the heat transfer from one working fluid into the second one via solid obstacle. In the heat transfer coefficient there is included thermal conductivity λ of solid wall that separates the two fluids and the heat transfer coefficient α for the interface between the solid wall and the two fluids. Determination of thermal conductivity is relatively easy because it is a material property. Heat transfer coefficient, as already mentioned, specifies the intensity of heat transfer from the fluid to the solid walls, and vice versa. This coefficient, however, is dependent on both the material properties of the flowing fluid and on flow character around the fixed wall.

1.4. Dimensionless criteria

Reynolds number (Re) defines the ratio of inertial and viscous forces and is determined from the boundary conditions and physical conditions as dimensionless criterion in order to specify a laminar or turbulent flow. Its value characterizes the flow in the transition region between laminar and turbulent flow [3].

$$\text{Re} = \frac{u d_h}{\nu} \quad (1.4.1)$$

where so called hydraulic diameter d_h represents in case of the flow in the pipe the diameter of the pipe, in case of flow around the pipe as well as its diameter, u is the mean velocity of the flowing medium. For the flow in the pipe, if the value is $\text{Re} < 2320$ it is a laminar flow (particles move in layers). At higher $\text{Re} > 2320$ it is a turbulent flow (swirl particles) [4].

Prandtl number is the ratio of the viscous and thermal diffusion, and is dependent on the material properties of the fluid. It refers to the thickness of boundary layers, the reference velocity and temperature.

$$\text{Pr} = \frac{\rho c_p \nu}{\lambda} = \frac{\nu}{a} \quad (1.4.2)$$

For air it is possible to assume a constant value 0.7.

Grashof number is the ratio of buoyancy and viscous forces. Its value so indicates whether the fluid flow is significantly affected by gravity (thus lifting members)

$$\text{Gr} = \frac{g \beta (T_s - T_{ref}) d_h^3}{\nu^2} \quad (1.4.3)$$

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

$$\text{Fo} = \frac{\lambda \tau}{c_p \rho d_h^2} \quad (1.4.4)$$

τ is the time constant.

Nusselt number expresses the influence of the flow on the wall heat flux, depending on the geometric reference parameter (which is well-defined).

$$\text{Nu} = \frac{\alpha d_h}{\lambda} \Rightarrow \alpha = \frac{\text{Nu} \lambda}{d_h} \quad (1.4.5)$$

The value of Nusselt number so specifies the ratio of convection to conduction (heat transfer to conduction). Heat transfer coefficient, as already mentioned, specifies the intensity of heat transfer from the fluid to the solid walls, and vice versa. This factor, however, is dependent on both the material properties of the flowing fluid and on flow character around the walls

Second definition of Nusselt number contains more measurable variables, such as thermal power P , characteristic dimension d_h , area S on which the heat transfer is determined, the temperature gradient between the wall temperature and the reference ambient temperature $\Delta T = T_s - T_{ref}$. The temperature gradient can be specified as a mean logarithmic difference.

$$\text{Nu} = \frac{P d_h}{S \Delta T \lambda} \quad (1.4.6)$$

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

$$\text{Nu} = f(\text{Re}, \text{Pr}, \text{Gr}, \text{Fo}) \quad (1.4.7)$$

Heat transfer is divided by gravity into two modes:

- **Natural (free) convection** - is dominantly controlled by buoyancy forces (gravity). The flow of fluid is then invoked only by changing the density (warm liquid rises, cold drops)
- **Forced convection** - is dominantly controlled by the fluid flow through heat exchanger and around the plates. The flow is caused by an external force acting on the fluid (pumps, fans, etc.). Gravity is negligible in this case.

In the case of forced convection, the value of Nusselt number is determined as a function of the value of the Re number. There are a number of empirical relations for calculating the Nusselt number in various simple geometries [24], see Tab. 1.3

Tab. 1.1 Forced convection

laminar flow around the plate,	$\text{Nu} = 0,664 \text{Re}_L^{1/2} \text{Pr}^{1/3}$	$0,6 \leq \text{Pr}$
T_s is constant	$\text{Re}_L = \frac{uL}{\nu}$, $10^4 \leq \text{Re}_L \leq 5 \cdot 10^5$, L plate length	

laminar flow around the plate, q is constant	$Nu = 0,908 Re_L^{1/2} Pr^{1/3}$ $Re_L = \frac{uL}{\nu}, 10^4 \leq Re_L \leq 5 \cdot 10^5, L$ plate length	$0,6 \leq Pr$	
turbulent flow around the plate, T_s is constant	$Nu_x = 0.0405 Re_L^{4/5} Pr^{1/3}$ $5 \cdot 10^5 \leq Re_L \leq 10^8$	$0,6 \leq Pr \leq 60$	
laminar flow in the tube	$Nu = 4.36$ pro $q = \text{const.}$ on the wall $Nu = 3.66$ pro $T = \text{const.}$ on the wall		
turbulent flow in the tube	$Nu = 0,023 Re^{0.8} Pr^m,$ $3 \cdot 10^4 \leq Re_L \leq 10^6$	$m = 0.3$ for cooling $m = 0.4$ for heating	
laminar, transverse and turbulent flow across the tube	$Nu = C_1 Re^{C_2} Pr^{0.38}$ Re 0,4 ÷ 4 4 ÷ 40 40 ÷ 4 000 4 000 ÷ 40 000 40 000 ÷ 400 000	C1 0,989 0,911 0,683 0,193 0,0266	C2 0,330 0,385 0,466 0,618 0,805
laminar, transient and turbulent flow across the tube bundle, N_L is a number of tube column	$Nu_D = C_1 Re_{D,\max}^m$ pro $N_L > 10,$ $Pr = 0.7,$ constants C_1 and m are given in table S_L – horizontal distance of tubes, S_T – vertical distance of tubes	$2000 \leq Re_{D,\max}^m \leq 40000$	

Tab. 1.2 Constants for determining the Nusselt number when flowing across the tube bundle

tubes in row	$S_T/D = 1.25$	$S_T/D = 1.50$	$S_T/D = 2.00$	$S_T/D = 3.00$
S_L/D	C_1 m	C_1 m	C_1 m	C_1 m
1.25	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
tubes in 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

Tab. 1.3

laminar flow inside tube	$\text{Nu} = 1,615 \left(\text{Re} \text{Pr} \frac{d}{l} \right)^{1/3}$
Turbulent flow inside tube	$\text{Nu} = 0,023 \text{Re}^{0,8} \text{Pr}^m, \quad m=0.3 \text{ for cooling}$ $m=0.4 \text{ for heating}$
Transverse flow around tube	$\text{Nu} = 0,59 \text{Re}^{0,47} \text{Pr}^{0,38} \left(\frac{\text{Pr}_f}{\text{Pr}_w} \right)^{0,25} \quad \text{pro } 10 \leq \text{Re} \leq 1000$ $\text{Nu} = 0,21 \text{Re}^{0,62} \text{Pr}^{0,38} \left(\frac{\text{Pr}_f}{\text{Pr}_w} \right)^{0,25} \quad \text{pro } 1000 \leq \text{Re} \leq 200000$

2. Transfer and its solution

Understanding the transfer of variables is the basis for many engineering applications including mechanical equipment, such as motors, pumps and transport systems (transport of oil, chemicals, food, etc.), energy systems and equipment [2]. In order to calculate the transfer of mass, momentum, energy, and other properties and substances through the surface, it is necessary to distinguish the movement of the fluid at various length scales - macroscopic scale (particles) and a microscopic scale (molecules). At macroscopic Eulerian approach it is necessary to determine the velocity field. Transfer of fluid particles via a surface is called a **convective transfer**.

Transfer defined at the molecular level is called **diffuse transfer**. Convection is zero if the fluid is not moving, but diffusion transport can also be zero at rest, eg. the existence of temperature gradient is determined only by diffusive transport of heat. During fluid flow both transfers are present, but one of them can be significantly higher than the second. For example at turbulent flow the convective mass transfer of momentum and energy can be surprisingly large. The surface over which the transfer is in progress, can be a real wall bounding the volume of fluid or fictitious wall located inside the fluid (the inner surface of the flow). To clarify the difference between both transfers, in Fig. 2.1 and Fig. 2.2 the convection of heat from the walls and diffusion (conduction) heat between two walls of different temperatures is shown.

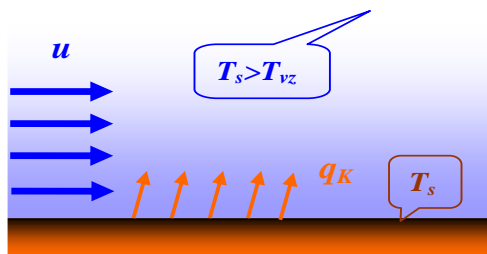


Fig. 2.1 Convective heat transfer

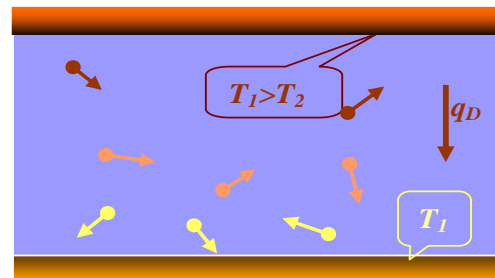


Fig. 2.2 Heat transfer by diffusion

2.1. Definition of transfer

2.1.1. Convective transfer

Transfer Γ_k at some point of flow area is defined by the rate, at which the given quantity is transferred through the surface. In differential form it is defined

$$d\Gamma_k = \zeta \left(\vec{u} \cdot \vec{n} \right) dS \quad (2.1.1)$$

where ζ general variable (scalar)

dS element size of surface

\vec{n} normal vector to the surface element dS

$(\vec{u} \cdot \vec{n})$ creates a normal component of the velocity vector to the surface dS

$\zeta \vec{u}$ is called flux density of variable ζ .

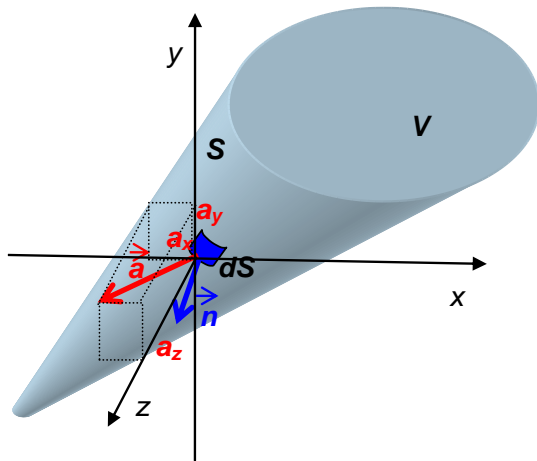


Fig. 2.3 Coordinate system and surface definition dS

Convection of scalar ζ through

surface S is scalar defined by the surface integral

$$\Gamma_{\kappa} = \int_S \zeta (\vec{u} \cdot \vec{n}) dS \quad (2.1.2)$$

Surface integral is often called **convective integral of flow** or flux. The result of the integral, ie. convective transport is the value of the unit $\frac{[\zeta]}{s}$ (e.g.

volumetric and mass flow rate) and

is used more often than the **flux density** defined by unit $\frac{[\zeta]}{m^2 \cdot s}$. Flux can be visualized, see

Fig. 2.4. It is proportional to the density of the vector field, it is changing by the setting of direction of flow area and its size. Arrows coming from the area are sources (positive divergence), while ending on the area are sinks (negative divergence). In the case of three-dimensional space the area is oriented so that the flux coming from the area is regarded as positive (in the direction of outward normal), and the flow entering the area is considered negative.

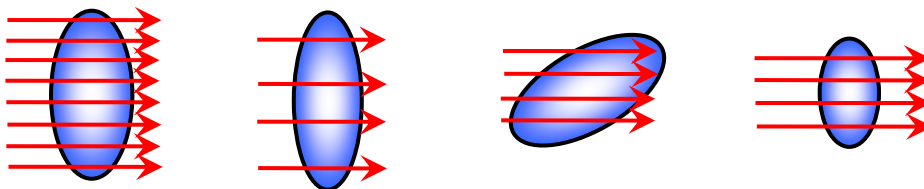


Fig. 2.4 The value of the flux depending on the density of the vector field, the direction of flow area and its size

If you mark the enthalpy $h = U + \frac{1}{2}(\vec{u})^2$, then the heat flux is generally defined as

$$\Gamma_K = \int_S h(\vec{u} \cdot \vec{n}) dS \quad (2.1.3)$$

Significant role in momentum transfer is the determination of momentum flux, i.e. the flow of the velocity vector through the surface which is defined as

$$\vec{\Gamma}_K = \int_S \vec{u}(\vec{u} \cdot \vec{n}) dS \quad (2.1.4)$$

At each point of area the transfer has a different value.

2.1.2. Diffusion transfer

The diffusion transfer arises from the microscopic movement of molecules, it depends on the orientation and shape of the surface and on the distribution of properties at a given point. It is useful to define a diffusion transfer flux at a given point, which has a dimension of the transported quantity per unit area and unit time. For fluids such as air and water, the relationship between the flow and gradient of transported quantities is modeled by linear relationship, which is sufficiently accurate for engineering applications.

When determining the heat conduction according to the Fourier law, for example, the heat flux density vector is:

$$\vec{q}_D = -\lambda \nabla T \quad (2.1.5)$$

The situation is similar for concentration.

The total diffusion transfer is analogous to the total convective transfer given by the surface integral

$$\Gamma_D = \int_S (\vec{q}_D \cdot \vec{n}) dS \quad (2.1.6)$$

2.1.3. Total transfer

The total transfer is then expressed as the sum of convective and diffusion transfer

$$\Gamma = \Gamma_K + \Gamma_D \quad (2.1.7)$$

2.1.4. Balance transfer equation

The physical laws describing transfer are the conservation laws of mass, momentum, heat or other scalar variables. They are expressed by the energy equation, Navier Stokes

equations coupled with the continuity equation in general conservative form and describe laminar and turbulent flow regime.

$$\iiint_V \frac{\partial(\rho\zeta)}{\partial t} dV + \iint_S (\rho\zeta \vec{u} \cdot \vec{n}) dS = \iint_S [\alpha_\zeta \nabla \zeta] dS + \iiint_V S_\zeta dV \quad (2.1.8)$$

$$\text{accumulation} + \text{convection} = \text{diffusion} + \text{source}$$

where ζ is variable and the members in the equation are sequentially accumulation, convection, diffusion and the source member, so the equation is also called **convection - diffusion equation**.

This equation can be expressed in differential form (more common in textbooks of Hydromechanics and Thermodynamics). We use the divergence theorem to convert the surface integral to the volume integral.

Equation (2.1.8) has the form

$$\iiint_V \frac{\partial(\rho\zeta)}{\partial t} dV + \iiint_V \nabla \cdot (\rho \vec{u} \zeta) dV = \iiint_V \nabla \cdot [\alpha_\zeta \nabla \zeta] dV + \iiint_V S_\zeta dV \quad (2.1.9)$$

$$\text{accumulation} + \text{convection} = \text{diffusion} + \text{source}$$

Since the equation is true for any integral applied to any volume, so it is true for the expression under the integral

$$\frac{\partial(\rho\zeta)}{\partial t} + \nabla \cdot (\rho \vec{u} \zeta) = \nabla \cdot [\alpha_\zeta \nabla \zeta] + S_\zeta \quad (2.1.10)$$

$$\text{accumulation} + \text{convection} = \text{diffusion} + \text{source}$$

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

The task to find a solution of equation (2.1.10) following boundary and initial conditions is called mixed problem. If the boundary conditions equal to zero they are called homogeneous boundary conditions, if the initial conditions equal to zero they are called homogeneous initial conditions. Instead of boundary conditions, conditions of another type may also be given, which are also called boundary. Consideration on the boundary and initial conditions for temperature is valid for general variable ζ .

2.1.5. Boundary conditions

At heat transfer flow modeling there are defined the areas filled with flowing fluid or solids. Areas are bounded by flow boundaries and walls. The walls can also separate the flowing fluids and are washed from both sides. At flow boundary and wall, the state values are entered based on available measurements (pressure, velocity, flow rate, temperature,

heat flow, etc.). A typical example is the tubular heat exchanger, which is measured and theoretically tested in the literature, see Fig. 2.5.

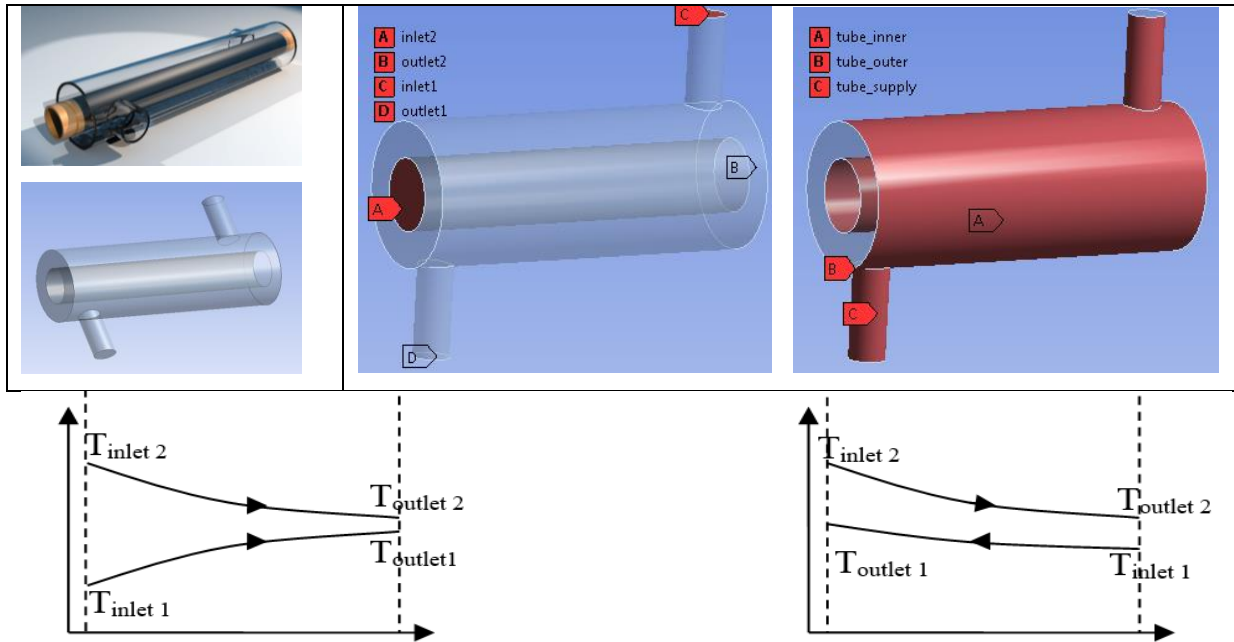


Fig. 2.5 Tubus exchanger, simplified geometry and typical temperature patterns for both co-flow and counter-flow exchangers.

The flow surfaces are shown in the diagram as input1, input2, outlet1 and outlet2. The walls are simply defined by surfaces such as the inner, outer and supply tubes, but can also be defined in the model by a real volume (eg a tube wall of 3 mm thickness). From the scheme the following boundary conditions can be defined:

- **flow areas - input1, input2, outlet1, and outlet2**
- **walls - tube inner, outer and supply**

In terms of geometry simplification, other significant areas of the region can be considered

- **symetry** (the region is assumed to be symmetric by plane)
- **axi-symetry** (the region is rotationally symmetrical).

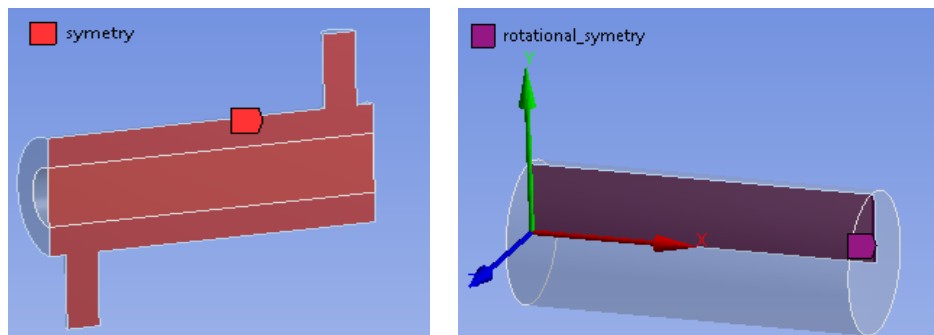


Fig. 2.6 Tubus exchanger, symmetry plane (left), solution area for rotationally symmetry (right)

Boundary conditions need not be a constant value, but can be defined by functions, tables, etc.:

- constants $y = konst$
- polynomial function

$$y(x) = A_0 + A_1x + A_2x^2 + \dots,$$

where the coefficients are entered only of five significant digits

- derivatives with respect to normal (OUTLET, temperature flux)

$$\frac{\partial y(x)}{\partial x} = konstl.$$

- piecewise linear function $(x_1, y_1), (x_2, y_2), (x_3, y_3), \dots (x_N, y_N)$
- combination of polynomial and piecewise linear function

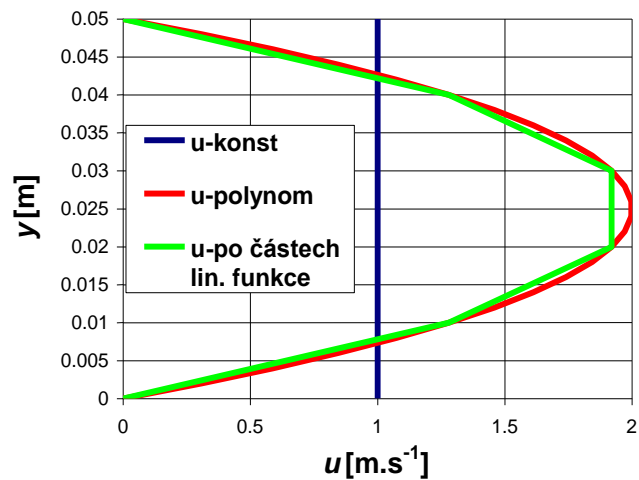


Fig. 2.7 Velocity profiles

2.2. Numerical methods of solutions

The aim of numerical methods for solution of partial differential equations is looking for isolated solution defined in a sufficiently small subarea using so-called discretized (algebraic) equations in basic points

- dividing the area into isolated geometric elements - establishing a mesh
- balancing the unknown variables in finite volumes or nodes and discretization
- numerical solution of discretized equations in general form

while the discretization error is defined as the difference between the solution of differential and discretized equations. The basic properties of numerical methods are:

- level of accuracy of the discretization error and residual
- level of stability

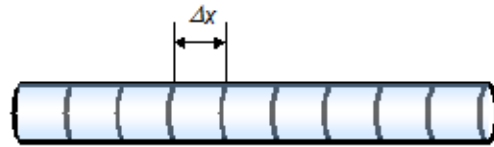
There are some developments in the numerical solution of the equations defining the fluid flow and heat transfer.

2.2.1. Difference method

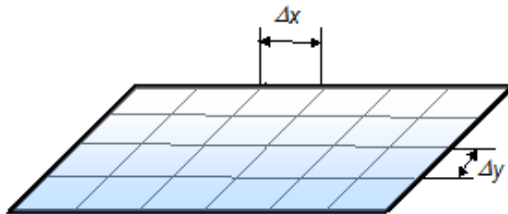
The oldest classical method is the **difference method**. The principle of difference methods for solving of differential equations can be described as follows

- the area, in which a solution is looking for, is covered with mesh composed from a finite number of non-overlapping elements. The simplest mesh is:

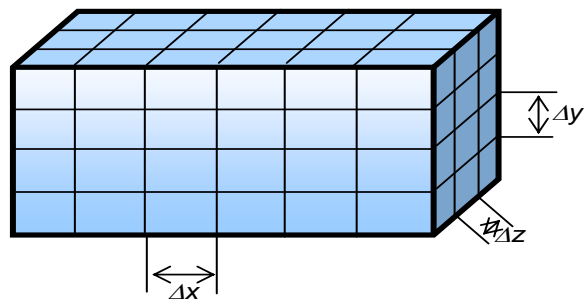
the lines in one-dimensional case



rectangles in two-dimensional case



hexahedrons in three-dimensional case



- at these points the derivative will be replaced by differences of various precision (eg. $\left(\frac{\partial T}{\partial x}\right)_i \approx \left(\frac{\Delta T}{\Delta x}\right)_i = \frac{T_{i+1} - T_i}{\Delta x}$), relationships necessary for derivative are derived from Taylor series by specific labeling associated with heat conduction, convection, etc.
- differential equation goes into a system of algebraic equations with unknowns, which determine the approximate value of the unknown function in all mesh nodes
- system of algebraic equations is solved numerically

Solved example

Solve the heat equation in the rod given by the parabolic differential equation $\frac{\partial T}{\partial t} = a \frac{\partial^2 T}{\partial x^2}$.

The solution is found in a rectangle $D(t, x)$ and must satisfy the conditions:

Initial conditions $T(x, 0) = T_0(x) = 20^\circ C \quad 0 < x < L$.

Boundary conditions (BC) $T(0, t) = T_1(t) = 80^\circ C, T(L, t) = T_2(t) = 20^\circ C$

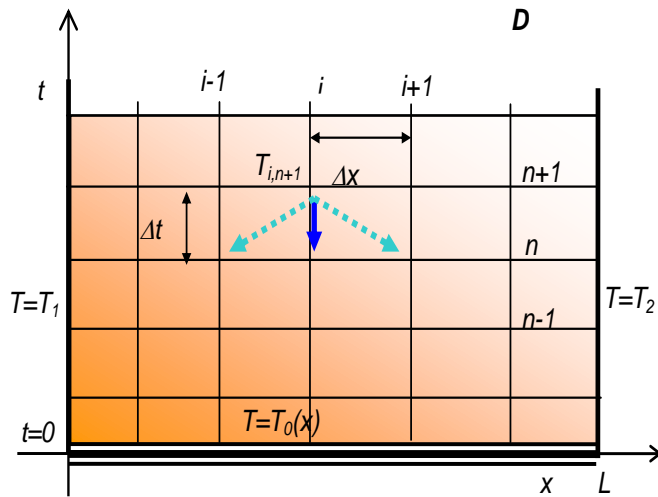


Fig. 2.8 The area geometry, boundary conditions, grid

Discretized equation of heat conduction has a form

$$\frac{T_{i,n+1} - T_{i,n}}{\Delta t} = a \frac{T_{i+1,n} + T_{i-1,n} - 2T_{i,n}}{\Delta x^2}$$

and after modification it applies

$$T_{i,n+1} = T_{i,n} + a\Delta t \frac{T_{i+1,n} + T_{i-1,n} - 2T_{i,n}}{\Delta x^2}$$

Therefore, we can express $T_{i,n+1}$ explicitly by using the values in the previous time step n . In this case we can find a solution in Excel.

In the following Tab. 2.1 it is shown (using Excel) definition of task and solution. The gray values can be changed, that is the size of area, the number of mesh elements, heat transfer coefficient and boundary conditions.

Tab. 2.1 Table of setting parameters for iterative calculation

a=	0.1	T(x=0)=	80	koef=	0.5
L=	1	T(x=L)=	20	Δx=	0.1
n=	10	T(t=0)=	20	Δt=	0.05

time	0.00	0.05	0.10	0.15	0.20	0.25	0.30	0.35	0.40	0.45	0.50
BC 0	80.00	80.00	80.00	80.00	80.00	80.00	80.00	80.00	80.00	80.00	80.00
	0.1	20.00	50.00	50.00	57.50	57.50	61.25	61.25	63.59	63.59	65.23
	0.2	20.00	20.00	35.00	35.00	42.50	42.50	47.19	47.19	50.47	50.47
	0.3	20.00	20.00	20.00	27.50	27.50	33.13	33.13	37.34	37.34	40.63
x	0.4	20.00	20.00	20.00	20.00	23.75	23.75	27.50	27.50	30.78	30.78
	0.5	20.00	20.00	20.00	20.00	20.00	21.88	21.88	24.22	24.22	26.56
	0.6	20.00	20.00	20.00	20.00	20.00	20.00	20.94	20.94	22.34	22.34
	0.7	20.00	20.00	20.00	20.00	20.00	20.00	20.00	20.47	20.47	21.29
	0.8	20.00	20.00	20.00	20.00	20.00	20.00	20.00	20.00	20.23	20.23
	0.9	20.00	20.00	20.00	20.00	20.00	20.00	20.00	20.00	20.00	20.12
BC 1	1	20.00	20.00	20.00	20.00	20.00	20.00	20.00	20.00	20.00	20.00

Convergence of problem depends on the choice of time and space step. Another problem is the efficient solution of this algebraic equations systems.

Fig. 2.9 shows changes in the temperature distribution along the length of the rod in dependence on time. After converged task the temperature would be spread linearly from left to right boundary conditions. Unfortunately, the graph would be unreadable.

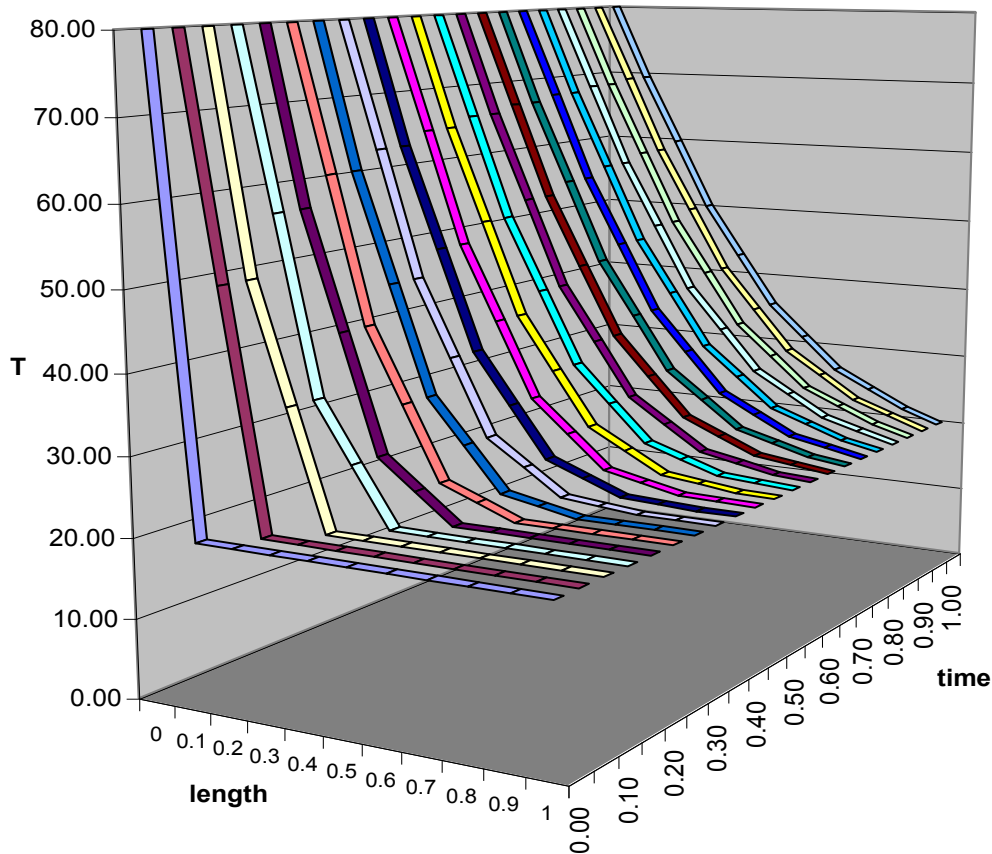


Fig. 2.9 Graphical presentation of solution in Excel

2.2.2. Finite volume method

Finite volume method (FVM) [1] , [13] consists briefly in three basic steps

- dividing area into discrete volumes using general curvilinear grid
- balancing unknown quantities in individual finite volume discretization
- numerical solution of discretized equations

Fluent defines discrete final volumes using **non-staggered scheme**, where all variables are stored in the centers of finite volumes.

The physical laws of transfer are the conservation laws of mass, momentum, heat or other scalar quantities and describe the laminar and turbulent flow regime. The balance equation for a general variable has a shape (see eq. (2.1.8))

$$\iiint_V \frac{\partial(\rho\zeta)}{\partial t} dV + \iint_S (\rho\zeta \vec{u} \cdot \vec{n}) dS = \iint_S [\alpha_\zeta \nabla \zeta] dS + \iiint_V S_\zeta dV$$

accumulation + convection = diffusion + source

After discretization of this general transfer equation in the control volume we obtain:

$$\frac{\partial(\rho\zeta)}{\partial t} V + \sum_f^{N_f} \rho_f \zeta_f \vec{u}_f \cdot \vec{S}_f = \sum_f^{N_f} [\alpha_\zeta \nabla \zeta_f] \vec{S}_f + S_\zeta V \quad (2.2.1)$$

where N_f is the number of areas surrounding the cells, ζ_f is value of variable ζ passing through the surface f , $\rho_f \vec{u}_f \cdot \vec{S}_f$ is mass flow through the surface f , \vec{S}_f is the size of the surface f , $\nabla \zeta_f$ is gradient of variable ζ on surface f , V is cell volume.

All the equations solved in ANSYS Fluent have the same general shape on any mesh. The discretized transmission balance equation contains a general variable in the center of cells in its vicinity. This equation is generally nonlinear. Its linearized form is

$$a_p \zeta = \sum_{nb} a_{nb} \zeta_{nb} + b \quad (2.2.2)$$

where the sum is performed over the neighboring cells (in 1D case is i=E, W; in 3D case i=N, S, E, W, F, B). a_{nb} are coefficients, which contain contributions from convection, diffusion and source members. The sign is visible from z Fig. 2.10.

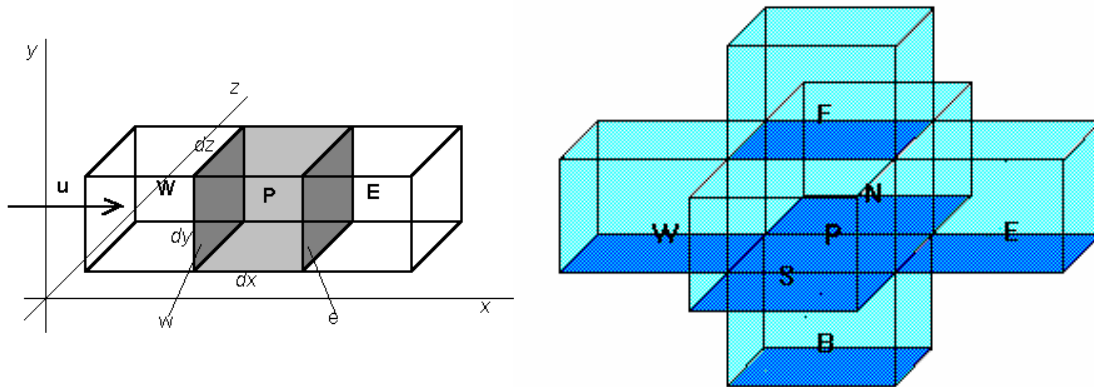


Fig. 2.10 Coordinate scheme using special notation of cells for 1D and 3D model instead of subscripts, where N – North, S – South, E – East, W – West, F – Front, B – Back

Each iteration consists of the steps that are shown in diagram on **Fig. 2.11** and are described as follows

- equations of motion for unknown velocity components are solved using the pressure values in order to update the velocity field
- velocity specified in the previous step can not satisfy the continuity equation, thus the

pressure correction and subsequent correction of velocity field shall be determined

- using new values of velocity the equation for the turbulent energy k and dissipation ε is solved
- another equation to determine the temperature and other scalar variables are solved
- physical properties of fluids (eg. viscosity) must be updated
- checking the convergence

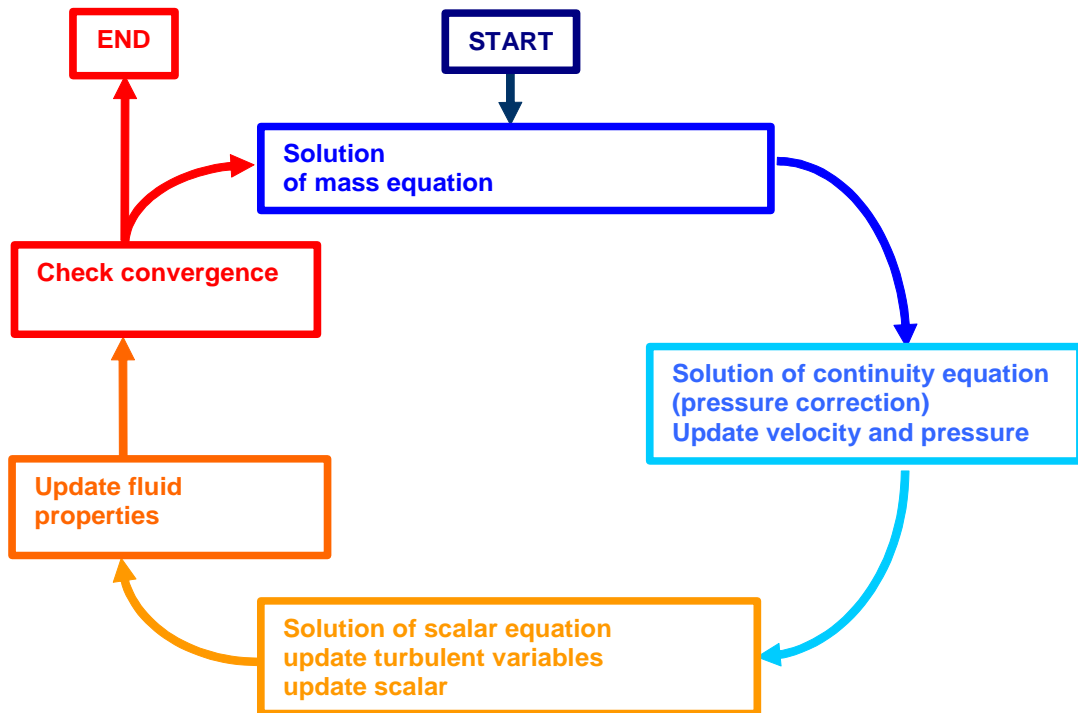


Fig. 2.11 Diagram of solution algorithm in Fluent [1]

2.2.3. Geometry and computational grid generation

Numerical finite volume method is based on a system of non-overlapping elements, finite volumes. Originally, the finite volume method was based on finite volumes of curvilinear rectangles, rectangles in two-dimensional case and blocks or general hexahedron in three-dimensional problems (see **Fig. 2.12**).

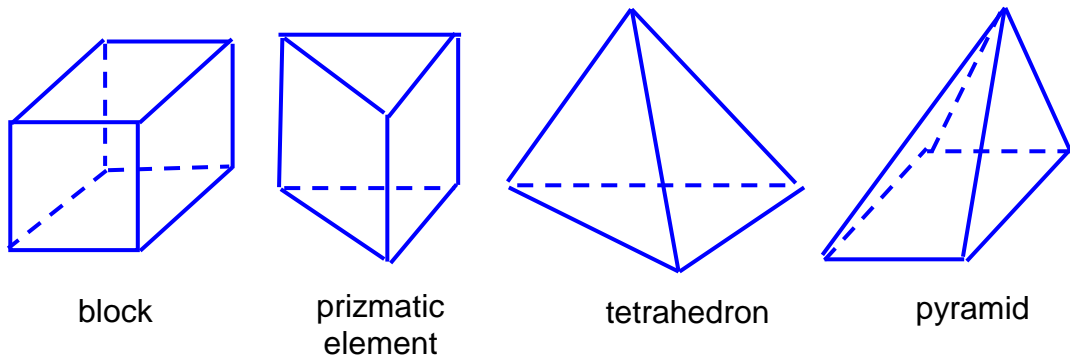


Fig. 2.12 Shape of finite volume

Such a grid is called a **structured grid**. The fundamental rule is that the elements boundary must adjacent with the single element boundary, so we can not arbitrarily refine grid (it is analogous to the method of finite differences, including the possibility of using indexing). Also, the resulting computational region is then a block or a rectangle. Today there is starting to take a new approach of building a so-called **unstructured grid**. The final volume is a 3D cuboid, tetrahedron, and pyramid prismatic element. The benefits have been validated in problems of elasticity, solved by finite element method.

The above listed elements can now be combined to obtain the optimal grid near the walls where the rectangles and blocks are used (to get more accuracy) and in other areas where there are no large gradients of solved variables due to existence of the boundary layer, we apply the remaining elements. They provide an easy change of grid density, see Fig. 2.13.

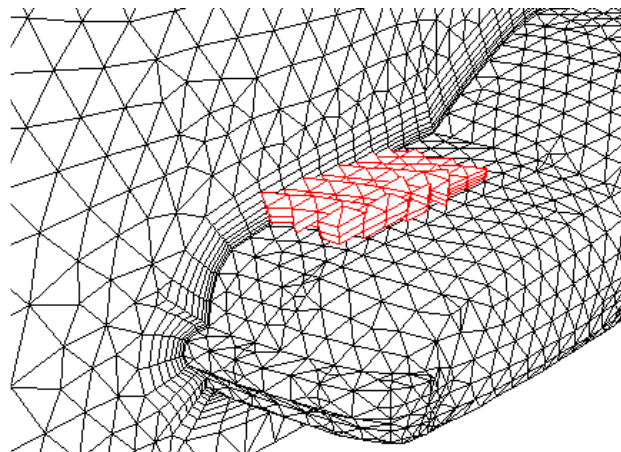


Fig. 2.13 Using the different types of elements [1]

To create geometry and grid the different CAD software to create a grid are used. It should be noted that it is appropriate to use the programs recommended in the manuals Ansys-

Fluent, because the grid which is created in software solving the problem of deformation or thermal conduction, is quite different from grid generated for the flow problem.

2.2.4. Choice of interpolation scheme

FLUENT stores the components of velocity and scalar quantities in geometric centers of finite volumes defined by grid. Because the calculation process, the required values of these variables on the border of finite volume are used. These values are obtained by interpolation, while you can choose between the following variants differing in order of accuracy (ascending)

- power interpolation
- quadratic upwind interpolation (QUICK)
- second-order interpolation / central difference
- QUICK

During large changes in pressure and flow it should be available to compute with the lowest order of accuracy (which is predefined) and after a few iterations to use higher order of accuracy (for vortex flow with heat transfer, dissipation, etc.)

2.2.5. Convergence and residuals

During flow simulation using Fluent it is very important to obtain convergent solutions. The measure of convergence are residuals, which represent the maximum difference between two corresponding values at the same grid point in two consecutive iterations. Residuals are evaluated for all values computed in each iteration step and are displayed for the selected variables.

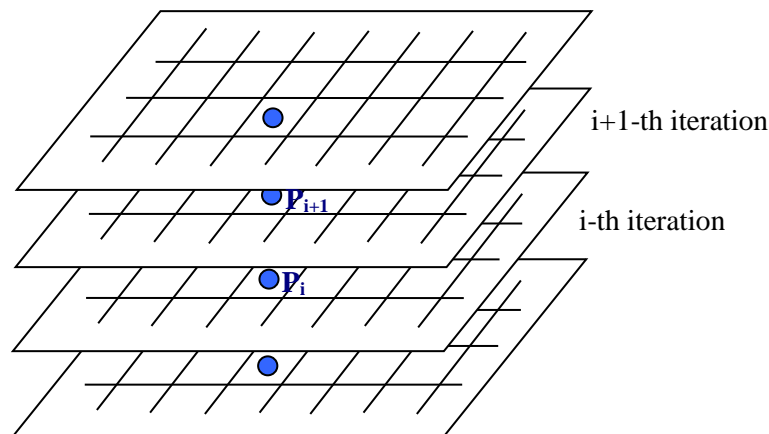


Fig. 2.14 Iteration in numerical steady calculation

It is also possible to assess at what point it is the highest residual value. Residuals are used to evaluation convergence. Generally, the solution converges very well when the normalized residuals are reduced to the value of the order of $1 \cdot 10^{-3}$ and enthalpy residuals to the value of the order $1 \cdot 10^{-6}$.

2.2.6. Convergence acceleration

Convergence is influenced by many factors such as initial conditions, a large number of cell, relaxing factor, etc.

To accelerate convergence, it is proposed to use an initial estimation of the variables important for flow which is the best way to start solving task successfully. Otherwise, all variables defined by initialization, are often considered to be zero at the beginning of the calculation. The most important examples of setting the initial conditions are:

- temperature for solution of heat transfer problems when using the equation of state
- velocity at a large number of cells
- temperature and velocity in solution of natural convection
- flow with chemical reaction, when it is available to set the temperature and the mass fraction of species

An important technique to accelerate convergence is the step by step technique (step by step from simple to more complex tasks). To solve the problem of heat transfer it is good to start the calculation from isothermal flow, in case of reacting flow to start the solution without the inclusion of additives. The problem is defined at first complex and then would be necessary to select the variables for which initial state will be resolved .

2.2.7. Relaxation

Due to nonlinearity of differential equations it is not generally possible to obtain values of all variables by solution of originally derived approximations of difference schemes. Convergence can be achieved using relaxation, which reduces the changes of each variable in each iteration. Simply said, the new value $\zeta_{P,i+1}$ in the finite volume containing point P

depends on the old value from the previous iteration $\zeta_{P,i}$, the new value from the current

iteration $\zeta_{P,i+1,vyg}$ (or calculated changes $\Delta\zeta_P = \zeta_{P,i+1,vyg} - \zeta_{P,i}$ and relaxation parameter

$\alpha \in \langle 0,1 \rangle$ follows

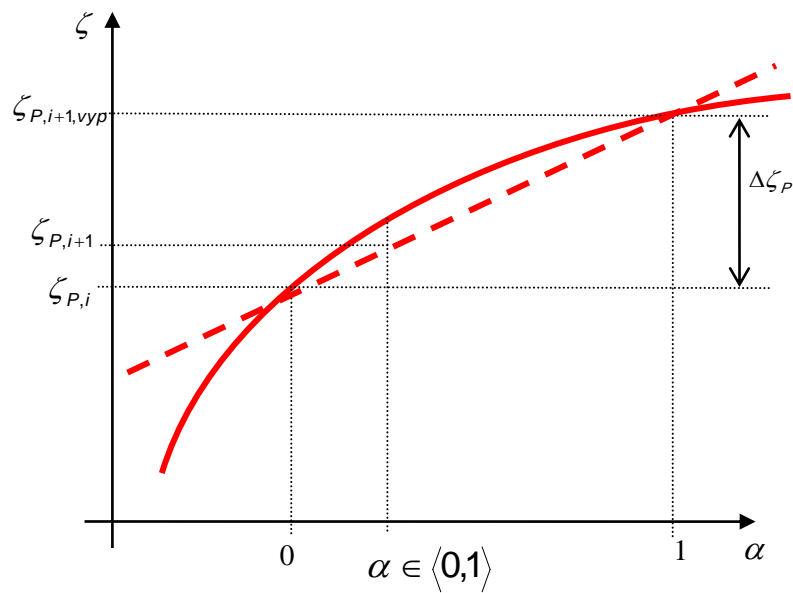


Fig. 2.15 Definition of relaxation parameter

$$\zeta_{P,i+1} = (1 - \alpha)\zeta_{P,i} + \alpha \cdot \zeta_{P,i+1,vyp} \quad (2.2.3)$$

These relaxation parameters can be set for all computed variables. Especially for velocity they are defined as very small, the order of tenths to hundredths. It is desirable during the calculation to change these values and accelerate the convergence, ie. if the residual changes are large in the transition from one iteration to another, than set a small relaxation factor and thus to damp non-linearity, if the residual changes become constant, you should increase the relaxation factors.

3. Heat transfer by conduction

3.1. Fourier equation

Heat conduction in solids is described by the first law of thermodynamics, ie the Fourier equation, which has the form:

$$\frac{\partial T}{\partial t} = \nabla \cdot (a \nabla T)$$

respectively using components

$$\frac{\partial T}{\partial t} = a \left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} \right)$$

(3.1.1)

If the area is geometrically simple (for example, heat conduction in the plate) and if the initial and boundary conditions are simple, the problem can be solved analytically (1D) and numerically (1D, 2D, 3D) due to the linearity of the equation. If the plate is of large size, the influence of the plate ends can be neglected, then the solution can be simplified to 1D resp. 2D task, see **Fig. 3.1**.

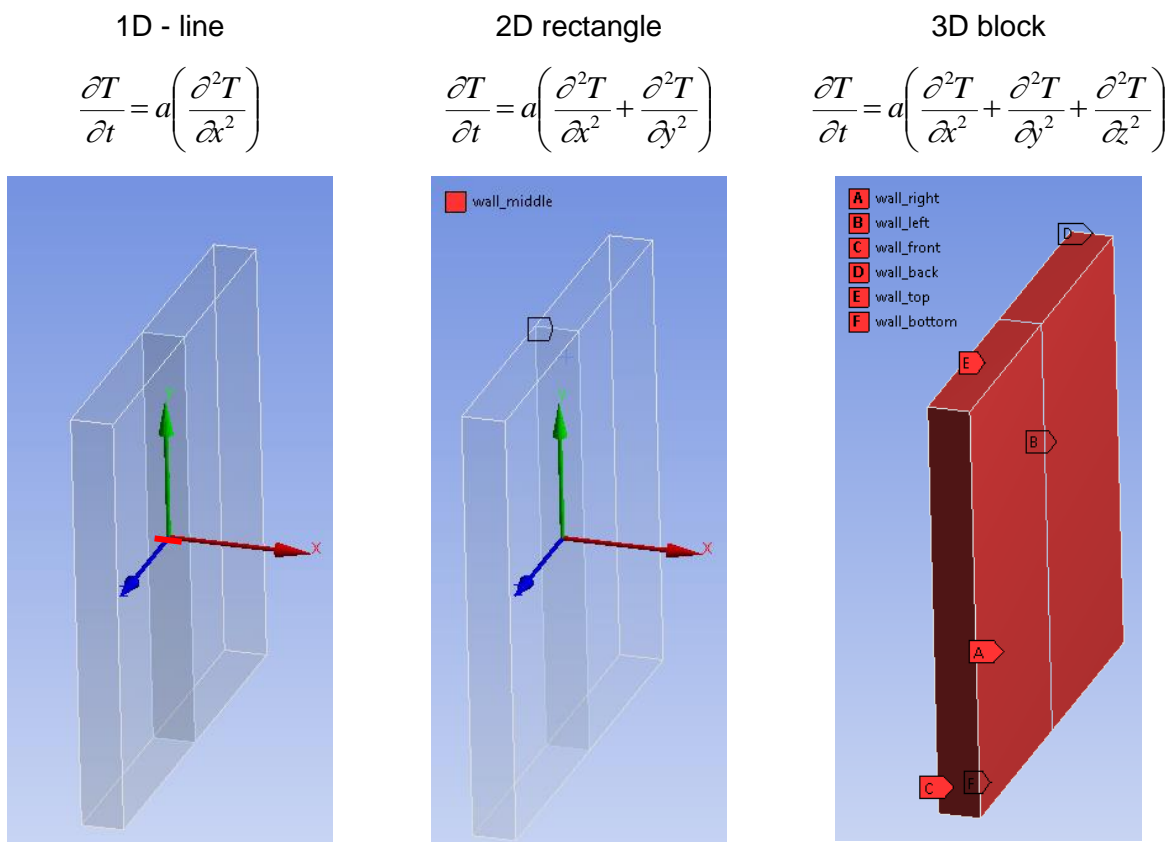


Fig. 3.1 1D, 2D, 3D computational area for large-size plate

In the case of the axially symmetric task (heat conduction in the rod), the simplification is in **Fig. 3.2** and the equations are then defined in the cylindrical coordinate system.

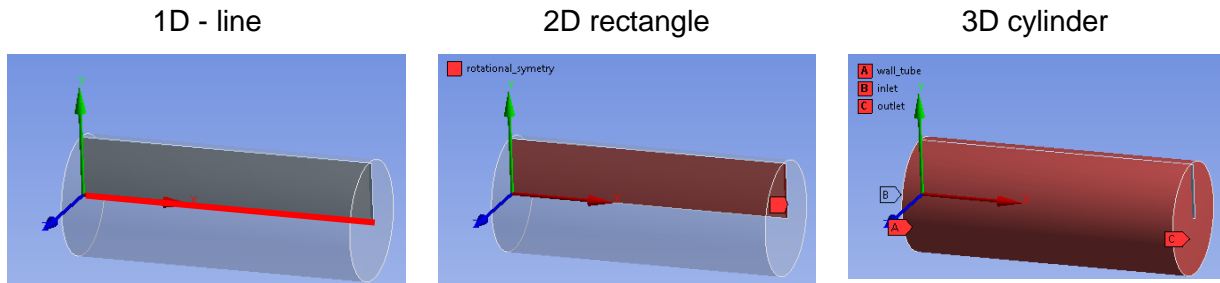


Fig. 3.2 1D, 2D, 3D computational area for cylinder

The images also show the meaning of boundary conditions, which must be in agreement with the experiment.

3.2. Equation of heat transfer by conduction

To determine the temperature distribution the Fourier's law expressing the law of energy conservation is used:

$$\frac{\partial}{\partial t}(\rho h) = \nabla \cdot (\lambda \nabla T) + S_h \quad (3.2.1)$$

where ρ density of the solid material
 h enthalpy of conductive material, $c_p(T - T_{ref})$
 λ thermal conductivity
 T teperature
 S_h heat source

In the above formulas the calculating enthalpy is defined for the reference temperature (e.g. $T_{ref} = 298.15 \text{ K}$), which can be changed according to the situation.

When problem solving, where there is still movement or rotation of the object, then these effects are included in the solution of the energy equation:

$$\frac{\partial}{\partial t}(\rho h) + \nabla \cdot (\vec{v} h) = \nabla \cdot (\lambda \nabla T) + S_h \quad (3.2.2)$$

Convection of heat is included in the energy equation for flow-limiting regions due to the wall's velocity movement.

Specifying the thermal conductivity allows to solve problems in which the solid conductive region is formed by separated walls of different materials and characteristics. The density and specific heat of the wall are important in solving the time-dependent problems.

Typical examples are the solutions of conveyor belts, moving rolled steel strip in furnaces, problems with rotating machine parts etc.

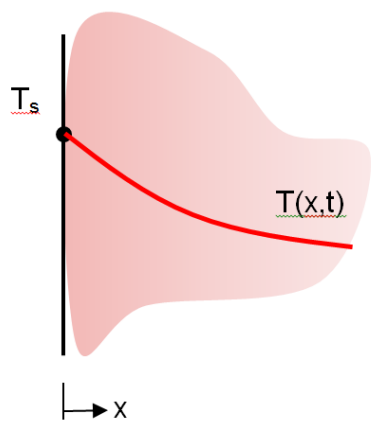
All physical properties can be constant or dependent on the temperature or on the pressure. The most important variable in this sense is the density. The above formula is generally assumed in three dimensional space. All variants such as

- heat transfer predominant in one or two direction
 - heat transfer in axisymmetric (rotating, cylindrical) coordinate system (pipes)
- are special, simplified case.

3.3. Boundary conditions

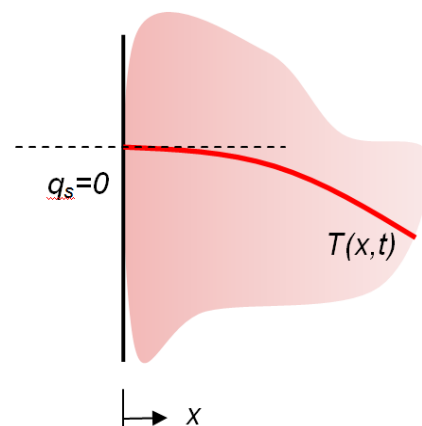
Heat conditions can be defined by four variants, see Fig. 3.3.

- **Constant temperature**
- **Constant heat flux**
- **Zero heat flux - adiabatic or isolated wall**
- **Convection – heat transfer coefficient and reference temperature** (Surface temperature influenced by convection)



Constant temperature surface

$$T(t, S) = T_s$$



Adiabatic or isolated wall

$$\frac{\partial T(t, S)}{\partial n} = 0$$

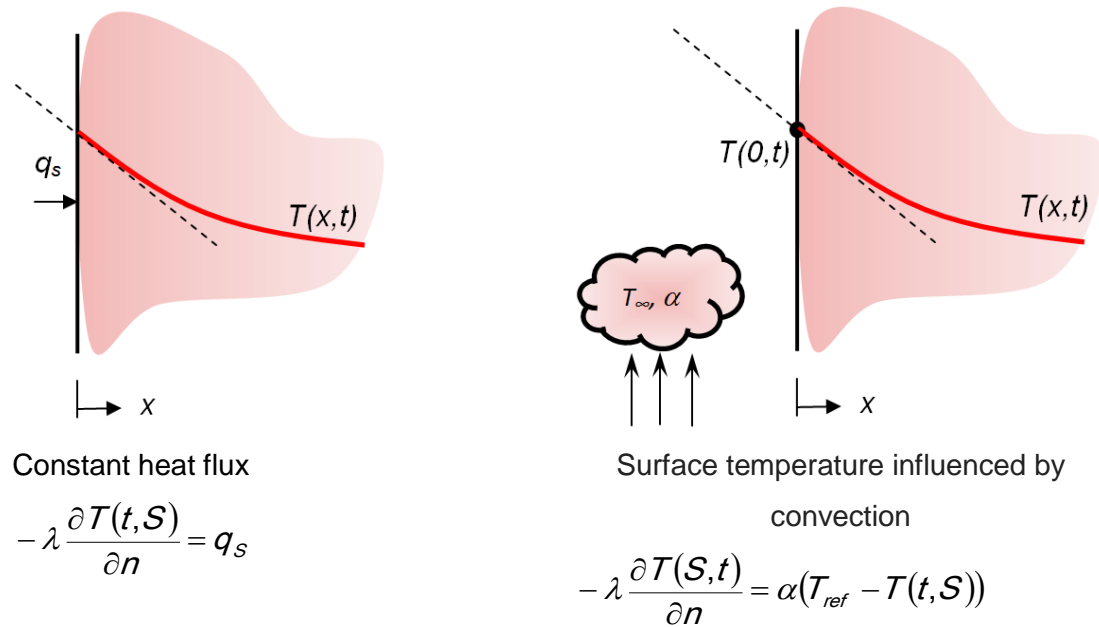


Fig. 3.3 Types of boundary condition

The last boundary condition is complicated because it involves the influence of fluid flow around the walls. Determination of the external heat transfer coefficient is given empirically and varies due to different fluids and flow velocities. The temperature on the outer wall is the result of calculation.

All types of conditions can be time-dependent, if required by their character.

3.4. One-dimensional heat conduction

3.4.1. Analytical solution

At any given simplification there is considered time independent (stationary) task of heat propagation in an infinitely large plate of thicknesses l , see Fig. 3.4.

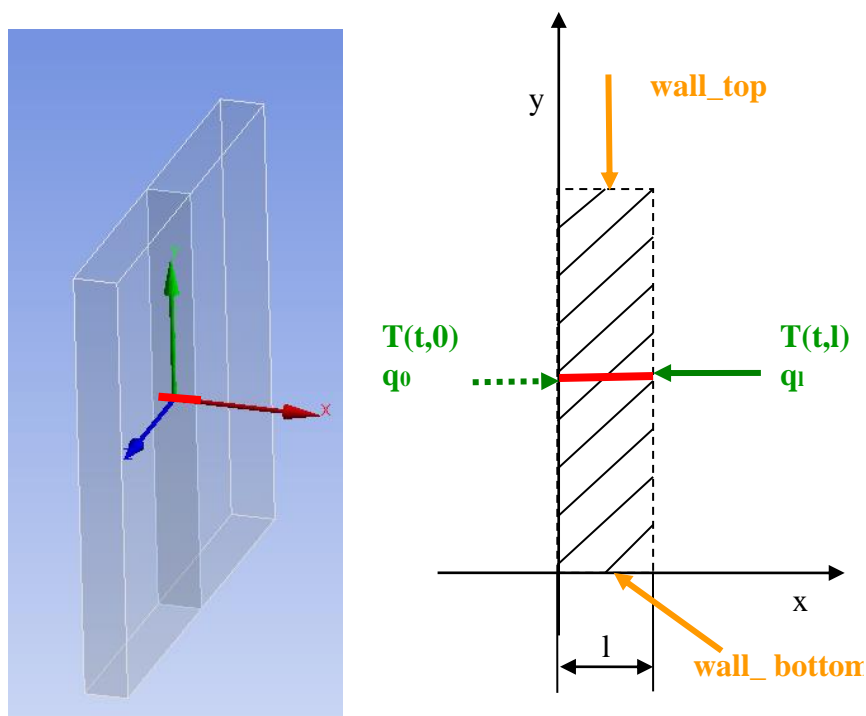


Fig. 3.4 Scheme of large plate of a given thickness in the coordinate systems and solution area – red line (1D)

The equation for this problem is

$$0 = \frac{\partial}{\partial x} \cdot \left(\lambda \frac{\partial T}{\partial x} \right) \quad (3.4.1)$$

This homogeneous equation has nonzero solutions for non-zero boundary and initial conditions, as shown in Fig. 3.4. Thus, when a constant heat conductivity λ is supposed, the solution can be deduced

$$\frac{\partial}{\partial x} \left(\lambda \frac{\partial T}{\partial x} \right) = 0 \Rightarrow \frac{\partial T}{\partial x} = C_1 \Rightarrow q = -\frac{C_1}{\lambda} \quad (3.4.2)$$

$$T = C_1 x + C_2$$

If they are given boundary conditions, e.g. $T(0) = T_1$ and $T(l) = T_2$, then

$$T_1 = C_1 \cdot 0 + C_2 \Rightarrow C_2 = T_1$$

$$T_2 = C_1 l + C_2 = C_1 l + T_1 \Rightarrow C_1 = \frac{T_2 - T_1}{l}$$

The solution is then in the form

$$T = \frac{T_2 - T_1}{l} x + T_1 \quad (3.4.3)$$

This result is also confirmed by the numerical solution in Fluent.

If the heat source inside the area given by heat flux q' is assumed, then the differential equation has the form:

$$\frac{\partial}{\partial x} \left(\lambda \frac{\partial T}{\partial x} \right) + q' = 0 \quad (3.4.4)$$

The general solution has the form:

$$T = -\frac{q'}{2\lambda} x^2 + C_1 x + C_2 \quad (3.4.5)$$

The constants are determined from the boundary conditions as well. The solution is a parabola, in case of identical conditions in both border areas it is symmetric parabola.

3.4.2. Numerical solution

This chapter illustrates on how to set and solve the temperature distribution in the plate of a given thickness in Fluent. Subsequently, this solution will be compared with the analytical solution. The task is:

- define the physical model, the physical properties of the material
- define a mathematical model, boundary conditions
- create geometry and grid
- specify boundary and initial conditions in Fluent, calculation
- evaluate the computed values
- compare the solutions with the analytical solution
- apply the same technique to different variants of boundary conditions and heat sources

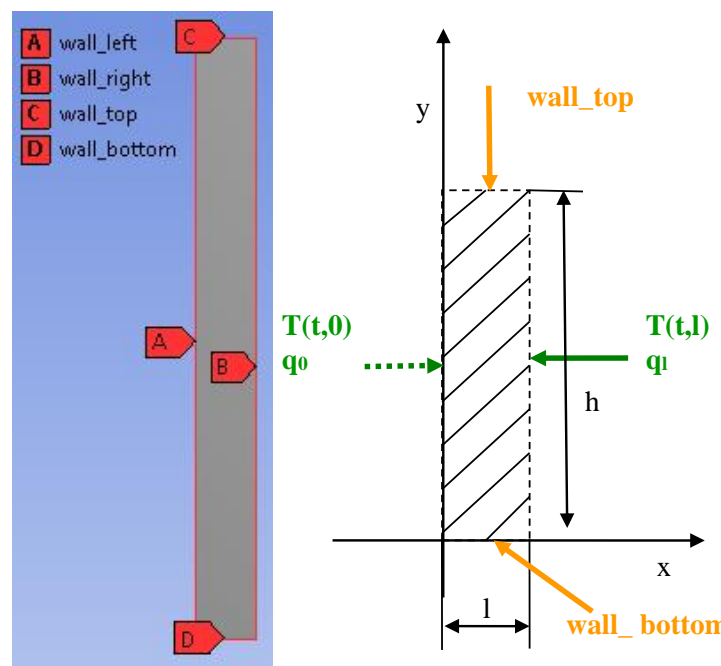


Fig. 3.5 Scheme of endless plate (2D)

Solve the temperature distribution in an infinitely large steel plate of a given thickness. The physical model is given by the shape of the area whose scheme is shown in 2D in Fig. 3.5 and the dimensions and physical properties in the table (1D area can not be resolved, it does not correspond to reality).

The dimensions of the area and physical properties of various materials for calculating variants are specified in Tab. 3.1 and Tab. 3.2.

Tab. 3.1 Geometry of area

thickness of area / [m]	0.01
height of area h [m]	0.1

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

materiál	wood	steel	aluminium	cooper
density ρ [$\text{kg}\cdot\text{m}^{-3}$]	700	8030	2719	8978
specific heat capacity c_p [$\text{J}\cdot\text{kg}^{-1}\cdot\text{K}^{-1}$]	2310	502.48	871	381
thermal conductivity λ [$\text{W}\cdot\text{m}^{-1}\cdot\text{K}^{-1}$]	0.173	16.27	202.4	387.6

Boundary conditions are defined on the left wall by temperature T_0 and on the right wall by temperature T_l or by heat flux q_l or by the ambient temperature T_∞ and the external heat transfer coefficient α . For solution it is prepared five variants of boundary conditions (A to E in **Tab. 3.3**) to be tested, because their task and the calculation is using identical geometry very easy. Due to the large plate dimensionality above and below the conditions of symmetry are defined.

Tab. 3.3 Boundary conditions

variant	left wall	right wall	right wall	right wall
	$T(0) = T_1$	$T(l) = T_2$	$-\lambda \frac{\partial T(l)}{\partial x} = q_l$	$-\lambda \frac{\partial T(l)}{\partial x} = \alpha(T_\infty - T(l))$
				$\alpha(l)$ T_∞
A	50	-10		
B	-20	100		
C	50		162700	
D	50		0	
E	50			40 -10

Mathematical model

In this task there is no flow, it is fictionally solved with zero velocity of flow, therefore, as laminar. The temperature distribution is controlled by the above-mentioned differential equation.

Geometry and grid generation

In an environment DesignModeller there is creating the exact geometry using a technique similar to the environment CAD programs. In addition, we use the options of this program to form grid, see Fig. 3.6.

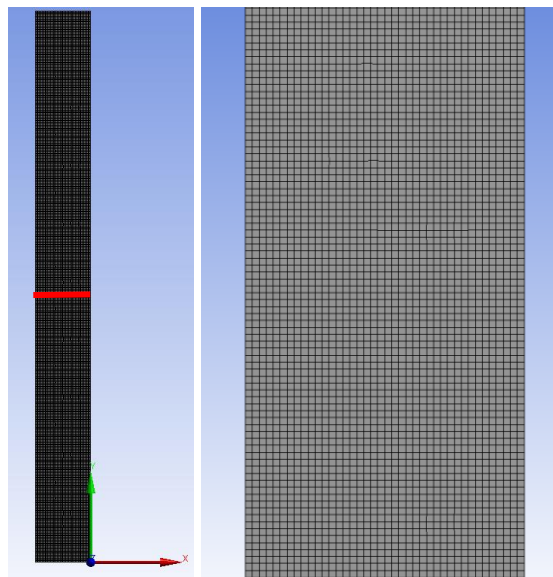


Fig. 3.6 Computational grid with red line marked for detailed evaluation of temperature and detail of mesh.

The results of the calculation of variant A

For clarity the evaluation options are presented, i.e. filled isolines of temperature, and other variables are meaningless, even if they are offered, such as pressure, velocity, etc.

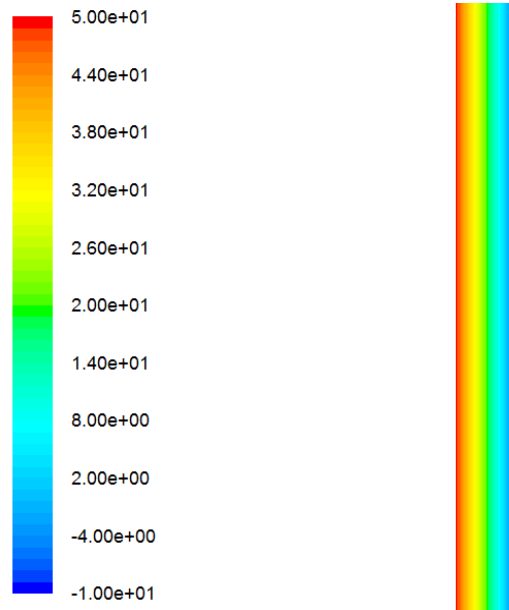


Fig. 3.7 The temperature distribution in the whole area [°C]

The temperature distribution in a cross section in the middle of area is in **Fig. 3.8**, which shows a linear temperature decrease from 50 ° C to -10 ° C. This is consistent with the analytical solution (line connecting boundary temperature values) in the previous chapter. This figure can be edited in Excel transmission data in text format.

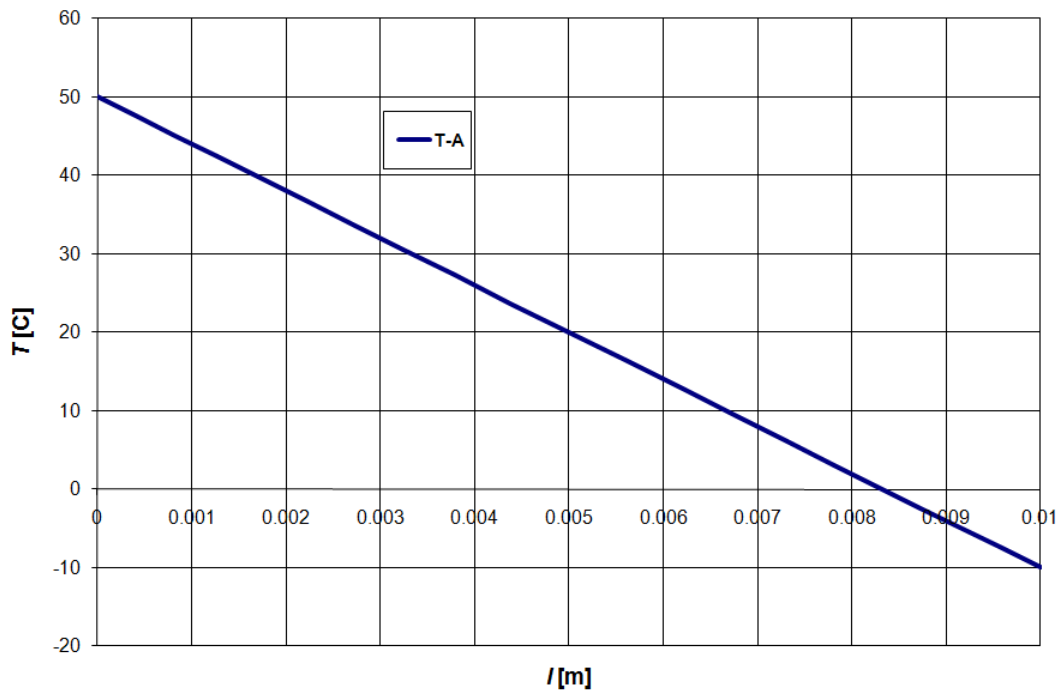


Fig. 3.8 The temperature distribution in the cross-sectional area

Very interesting is the evaluation of the heat through the left resp. right wall:

Tab. 3.4

heat through the wall Q [W]	steel
wall left	9761.44
wall right	-9762.65

Heat transfer through the wall elements in unit [$W \cdot m^{-2}$] can also be evaluate in detail in every location of the wall. In this simple case it is constant, because the temperature distribution in the x direction is linear, thus there is one slope (the derivative of temperature is the flow rate) but in general this result is not realistic.

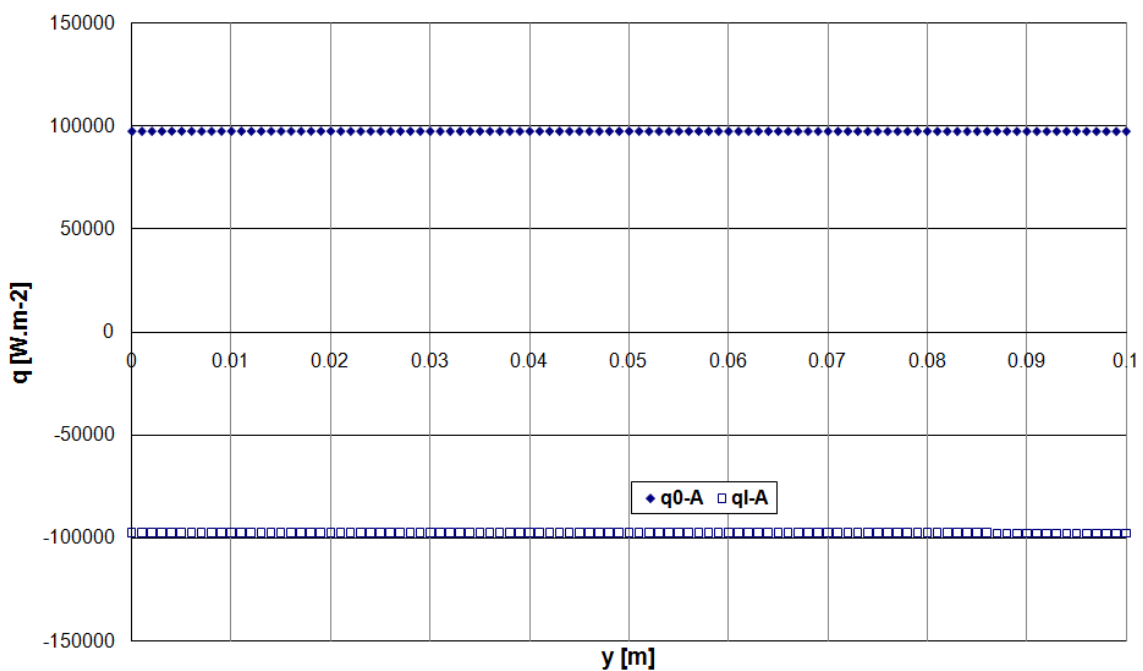


Fig. 3.9 The distribution of heat flux through the left and right wall

The calculation results of other boundary condition variants for steel

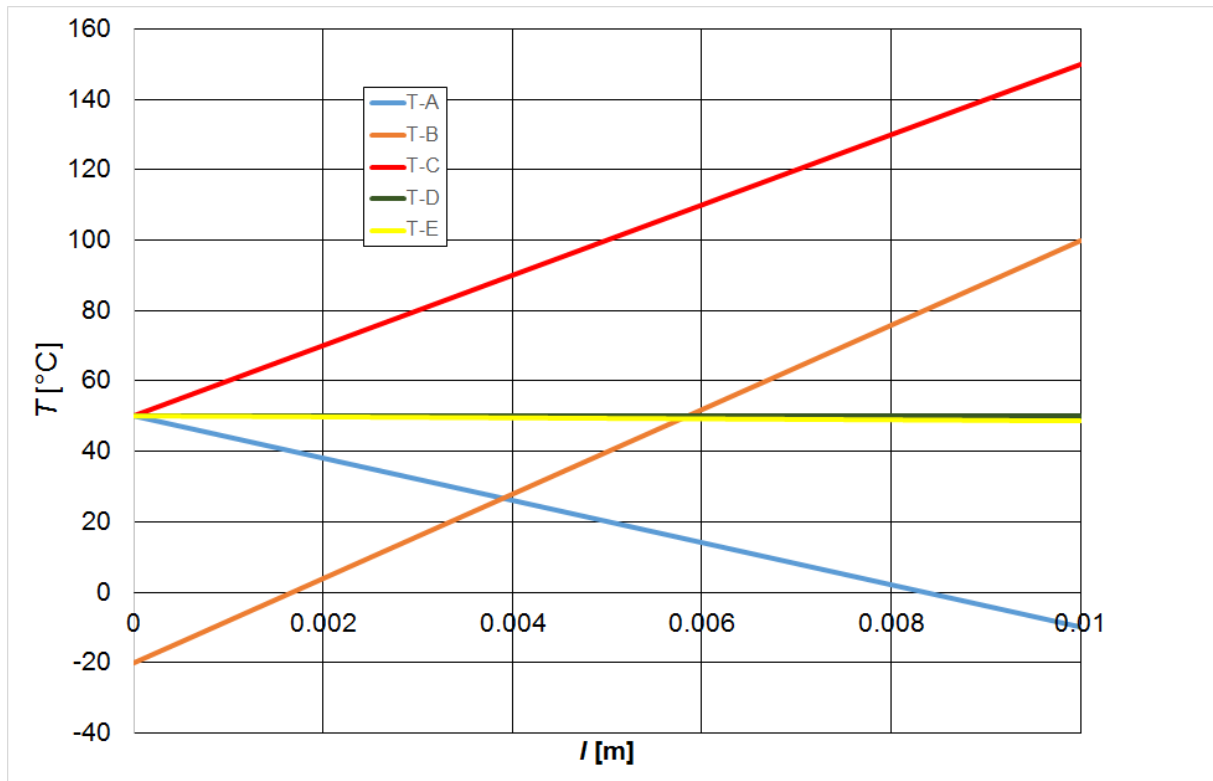


Fig. 3.10 The temperature distribution in the cross-sectional areas for variants A-E

Tab. 3.5

heat through the wall Q [W]	A	B	C	D	E
wall left	9760	-19522	-1623	-4.365	22.5
wall right	-9764	19526	16270	0	-23.4

Heat flux through the wall in $[W \cdot m^{-2}]$ can be also evaluated in detail in each grid cell. Because it is constant, it is evaluated only the average value for each variant:

Tab. 3.6

average heat flux through the wall q $[W \cdot m^{-2}]$	A	B	C	D	E
wall left	97518	-195216	-162536	-126	225
wall right	-97720	195265	162700	0	-234

Values are approximately ten times larger, because the flow area is 0.1 m^2 .

3.5. Temperature distribution at the transient transfer

A mathematical model solved by FVM is still the same, only in the equations there will be considered a member generally called accumulation and containing time derivatives. Thus, the solution will be defined with additional time step, which is estimated from a real setting and the number of time steps. The total time is thus the product of the time step and the number of steps.

Solve the problem of coating of infinite large aluminum sheet of given thickness by epoxy, which must be applied at least 5 min at 150°C. The process thus takes place in two stages. In the first stage the aluminum heats by the air in a furnace at a temperature of 175°C. In the second stage it is cooled in the space by air at temperature of 25°C.

The physical model is given by the shape of the area whose scheme is shown in 2D in **Fig. 3.11**, only sizes will be updated. Physical properties and boundary conditions are defined below in tables.

Tab. 3.7 Geometry of area

thickness area l [m]	0.003
height area h [m]	0.01

Tab. 3.8 Physical properties of material

(aluminium) at 300 K

material	aluminium
density ρ [kg·m ⁻³]	2719
specific heat c_p [J·kg ⁻¹ ·K ⁻¹]	871
heat conduction λ [W·m ⁻¹ ·K ⁻¹]	202

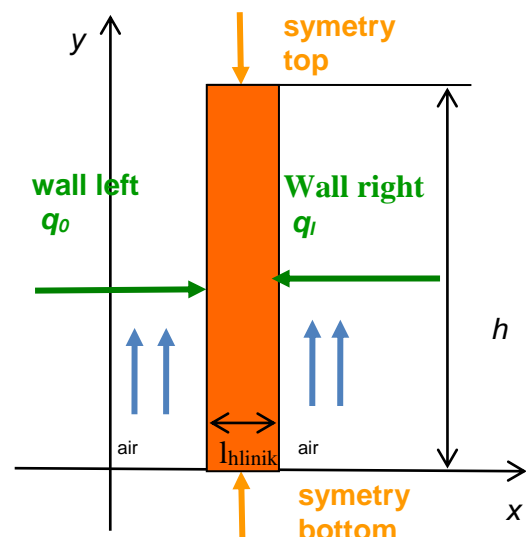


Fig. 3.11 Scheme of solved task

Boundary conditions

The boundary conditions are defined on the left and right wall by the same ambient temperature and the same heat flux q_l , influenced by the air flow, while the cooling or heating variant is distinguished. Due to the large size of plate, symmetry conditions are defined above and below the region. The temperature for this boundary condition must be defined in Kelvin.

Tab. 3.9

Variant	$\alpha(t)$ [$\text{W}\cdot\text{m}^{-2}$]	T_{∞} [$^{\circ}\text{C}$]	T_{∞} [K]
A heating	40	175	448
B cooling	10	25	298

Mathematical model

There is no flow in this task, so it is fictitious flow with a zero-velocity, as a laminar. The temperature distribution is controlled by the above differential equation. The first stage is solved, when the **heating takes place for a time estimated in advance**,

$$10 \text{ min} = 600 \text{ s.}$$

From the graph of the mean aluminum temperature vs time (**Fig. 3.12**), it is possible to see at what time the desired temperature (150 °C) is reached. After 5 min, it is possible to change the boundary conditions given in the second cooling stage and to continue the calculation. It is again clear from the graph when the required temperature of the aluminum plate is reached. Therefore, the calculation could be shortened by the period corresponding to the crossed-out part of the curve, that is, by the time

$$T = 600 - 463 = 137 \text{ s}$$

Thus, the calculation of the heating would be set not for 600 seconds but for 463 seconds, and then the boundary conditions would change to the cooling conditions.

Notes:

Of course Fluent allows you to automatically change the boundary conditions when the required time and temperature is reached using UDF functions (User Defined Function). The temperature distribution throughout the area is constant, so the cross-sectional temperature will not be plotted.

Results

The result is a temperature vs time plot showing the change in boundary conditions in Excel.

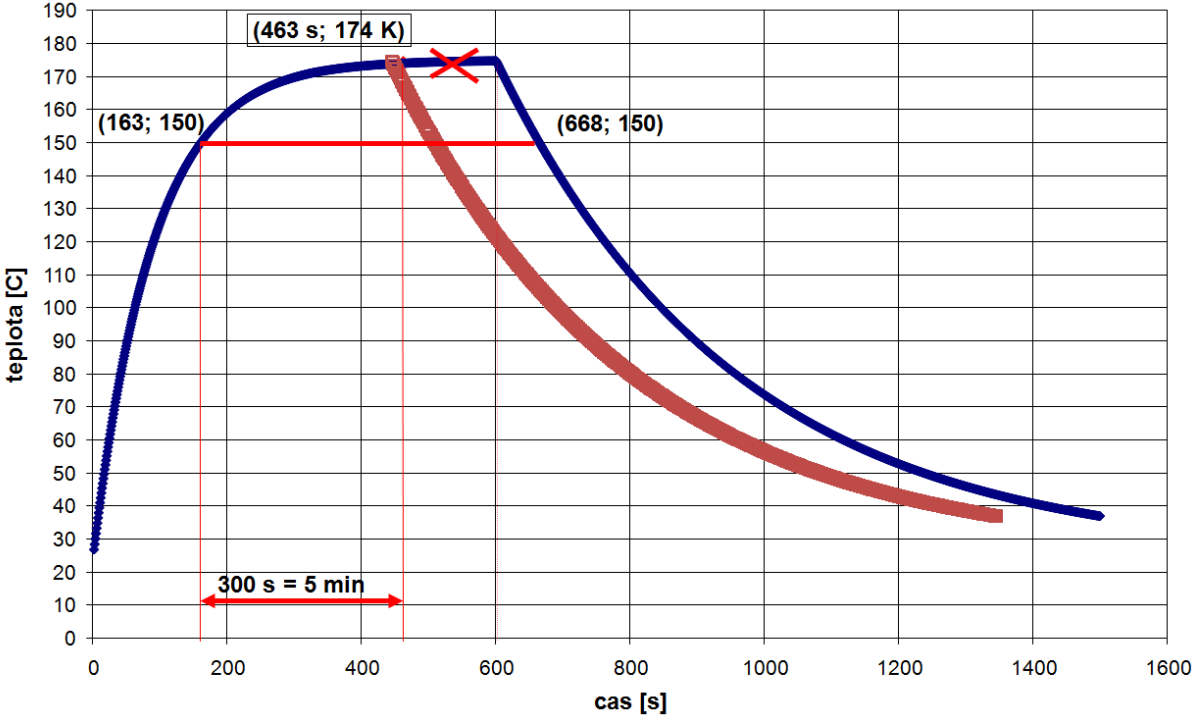


Fig. 3.12 Temperature vs. time graph

4. Mass and momentum transfer

4.1. Continuity equation

Continuity equation is identical for ideal and real fluid, ie according to law of conservation of mass (resp. mass flow rate). The sum of the temporal and convective flow change is equal to zero eventually to source member (e.g. flue gas from the stack in the solved area):

$$\iiint_V \frac{\partial \rho}{\partial t} dV + \iint_S \rho \vec{u} \cdot \vec{n} dS = S_z \quad (4.1.1)$$

You can also write the continuity equation in differential vector form:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{u}) = S_z \quad (4.1.2)$$

or in differential form:

$$\frac{\partial(\rho)}{\partial t} + \frac{\partial(\rho u_x)}{\partial x} + \frac{\partial(\rho u_y)}{\partial y} + \frac{\partial(\rho u_z)}{\partial z} = S_z. \quad (4.1.3)$$

This equation is the general equation of continuity for spatial unsteady flow of compressible fluid.

For **steady flow** of an incompressible fluid ($\rho = \text{const}$) the continuity equation is expressed by the relationship in vector form (source equals zero):

$$\nabla \cdot \vec{u} = 0 \quad (4.1.4)$$

In case of pipe flow and compressible respectively incompressible fluid a simplified well known relationship is applied and **mass** respectively **volumetric flow rate Q_V is constant**

$$Q_m = \rho u S = \text{konst.} \text{ resp. } Q_V = u S = \text{konst.} \quad (4.1.5)$$

4.2. Navier-Stokes (momentum) equation

Balance of forces in the flow of a real fluid is given by the Navier-Stokes equations expressing the relationship, when **inertia force is equal to the sum of the mass and surface (pressure and friction) forces.**

$$\vec{F}_s = \vec{F}_o + \vec{F}_p \quad (4.2.1)$$

In the real fluid flow we choose elementary volume dV . To that fluid volume the external volume force $d\vec{F}_o$ is applied (e.g. gravity, centrifugal force or general force vector

defined by acceleration \vec{a}). Differential of mass force and consequently the total mass force is given by

$$d\vec{F}_o = \vec{a} dm = \rho \vec{a} dV \Rightarrow \vec{F}_o = \iiint_V \vec{a} \rho dV \quad (4.2.2)$$

Similarly, the inertial force is determined by the fluid acceleration (substantial derivative)

$$\frac{D\vec{u}}{Dt},$$

$$d\vec{F}_s = \frac{D\vec{u}}{Dt} dm = \frac{D\vec{u}}{Dt} \rho dV \Rightarrow \vec{F}_s = \iiint_V \frac{D\vec{u}}{Dt} \rho dV \quad (4.2.3)$$

Surface force that includes both pressure and friction force can be written using molecular stress tensor $\bar{\Pi}$, ie. both shear and normal stress [4] [5] [10] :

$$\bar{\Pi} = -p \bar{\delta} + \bar{\tau}$$

where p is the normal component of the stress (static pressure), which specifies the pressure force in the direction of inner normal in hydromechanics, and so that a pressure force is defined with a minus sign, $\bar{\tau}$ is the shear stress tensor, $\bar{\delta}$ is a unit tensor with components which may take the values 1 if $i = j$ and 0 if $i \neq j$. To illustrate the mathematical presentation of the friction forces the simplified Newton relationship applied in the coordinate system is used (see Fig. 4.1):

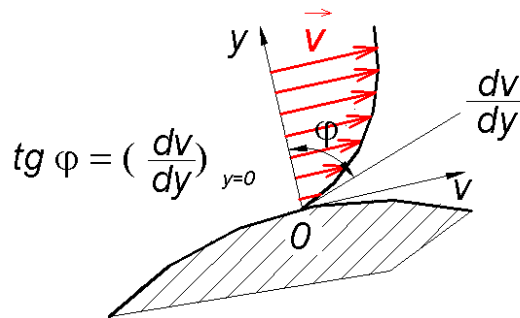


Fig. 4.1 Velocity profile depending on coordinate y [11]

$$\tau = \eta \frac{dv}{dy} \quad (4.2.4)$$

This already known form expresses the relation between viscous stress and velocity derivative according to one direction perpendicular to the movement.

Vector-tensorial formulation of shear stress in space is [8] :

$$\bar{\tau} = \eta \left[\left(\nabla \vec{v} + \left(\nabla \vec{v} \right)^T \right) - \frac{2}{3} \left(\text{div} \left(\vec{v} \right) \right) \bar{\delta} \right] \quad (4.2.5)$$

where $\nabla \vec{v}$ is tensor of velocity gradients with components $\left(\frac{\partial}{\partial x_i} \right) v_j$, $\left(\nabla \vec{v} \right)^T$ is transposed

tensor of velocity gradients with components $\left(\frac{\partial}{\partial x_j} \right) v_i$ and $\text{div} \left(\vec{v} \right)$ is divergence of velocity

vector. For incompressible fluid the $\text{div} \left(\vec{v} \right) = 0$. Then the differential of surface forces is

$$d\vec{F}_P = \bar{\Pi} \cdot \vec{n} dS = \left(-\rho \bar{\delta} + \bar{\tau} \right) \cdot \vec{n} dS = \left(-\rho \bar{\delta} + \eta \left[\nabla \vec{v} + \left(\nabla \vec{v} \right)^T \right] \right) \cdot \vec{n} dS \quad (4.2.6)$$

where \vec{n} is outer normal vector to element of closed surface dS .

The balance of all forces using vector notation for general compressible fluid in a rectangular coordinate system has the form

$$\iiint_V \frac{D\rho \vec{u}}{Dt} dV = \iiint_V \frac{\partial(\rho \vec{u})}{\partial t} dV + \iint_S \rho \vec{u} (\vec{u} \cdot \vec{n}) dS = -\iint_S \rho \vec{n} dS + \iint_S \bar{\tau} dS + \iiint_V \vec{f} dV$$

resp. (4.2.7)

$$\frac{\partial(\rho \vec{u})}{\partial t} + \nabla \cdot (\rho \vec{u} \vec{u}) = -\nabla \rho + \rho \vec{a} + \nabla \cdot (\bar{\tau}) + S_m$$

where $\vec{u} \vec{u}$ is so called dyadic product of vectors, see chap. 13.1. Equation is called **Navier - Stokes equation**. This equation for illustration can be formally written into three coordinate directions x, y, z in case of incompressible flow:

$$\begin{aligned} \frac{\partial u_x}{\partial t} + u_x \frac{\partial u_x}{\partial x} + u_y \frac{\partial u_x}{\partial y} + u_z \frac{\partial u_x}{\partial z} &= a_x - \frac{1}{\rho} \frac{\partial p}{\partial x} + \nu \left(\frac{\partial^2 u_x}{\partial x^2} + \frac{\partial^2 u_x}{\partial y^2} + \frac{\partial^2 u_x}{\partial z^2} \right) \\ \frac{\partial u_y}{\partial t} + u_x \frac{\partial u_y}{\partial x} + u_y \frac{\partial u_y}{\partial y} + u_z \frac{\partial u_y}{\partial z} &= a_y - \frac{1}{\rho} \frac{\partial p}{\partial y} + \nu \left(\frac{\partial^2 u_y}{\partial x^2} + \frac{\partial^2 u_y}{\partial y^2} + \frac{\partial^2 u_y}{\partial z^2} \right) \\ \frac{\partial u_z}{\partial t} + u_x \frac{\partial u_z}{\partial x} + u_y \frac{\partial u_z}{\partial y} + u_z \frac{\partial u_z}{\partial z} &= a_z - \frac{1}{\rho} \frac{\partial p}{\partial z} + \nu \left(\frac{\partial^2 u_z}{\partial x^2} + \frac{\partial^2 u_z}{\partial y^2} + \frac{\partial^2 u_z}{\partial z^2} \right) \end{aligned} \quad (4.2.8)$$

For one-dimensional flow, this equation is reduced like the continuity equation into a very simple form

$$\frac{\partial u_x}{\partial t} + u_x \frac{\partial u_x}{\partial x} = a_x - \frac{1}{\rho} \frac{\partial p}{\partial x} + \nu \left(\frac{\partial^2 u_x}{\partial x^2} \right) \quad (4.2.9)$$

From equation above you can easily deduce known Bernoulli equation. By solution of flow field the velocity and pressure distributions is calculated. In addition to the momentum equation the continuity equation applies too.

All physical properties can be constant or dependent on the temperature or on the pressure. The most important variable is the density.

In the system of differential Navier - Stokes equations and continuity equation there are four unknowns, ie. velocity components u_x, u_y, u_z and pressure p . To solve these equations the external acceleration \vec{a} , fluid density ρ and boundary conditions must be defined. Navier - Stokes equations are non-linear partial differential equations and are not generally solvable. The analytical solution is available for simpler cases of laminar flow. Nowadays even complex cases of laminar and turbulent flow are solved using numerical methods, eg. finite volume method and finite element method.

4.3. Conditions on flow boundaries and walls

For two flow boundaries only following basic combinations of boundary conditions may occur, (a combination of input velocity and output velocity can not occur because the velocity on the second input is calculated from the continuity equation).

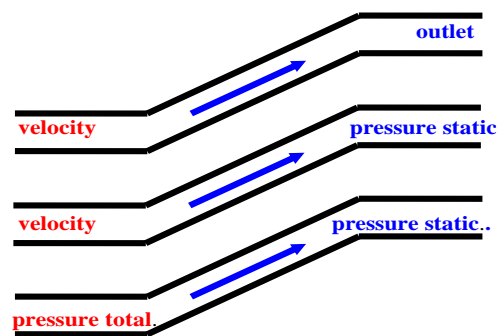


Fig. 4.2 Combination of input and output boundary conditions

Tab. 4.1 Boundary conditions on flow boundaries and walls

	Type (Fluent)	Variable	Unit
INLET	VELOCITY INLET	velocity u	[ms ⁻¹]
	MASS FLOW RATE	mass flow rate Q_m	[kgs ⁻¹]
	PRESSURE INLET	total pressure	[Pa]

		$\rho_{tot} = \rho_{stat} + \rho_{dyn} = \rho_{stat} + \frac{1}{2} \rho u^2$	
OUTLET	OUTFLOW	condition of stable flow $\frac{\partial u}{\partial n} = 0, \frac{\partial p}{\partial n} = 0$	
	PRESSURE OUTLET	static pressure ρ_{stat} (equals zero at flow into atmosphere)	[Pa]
	MASS FLOW RATE	mass flow rate Q_m	[kgs ⁻¹]
WALL	WALL	$u = 0$ is default (moving wall – insert velocity - eg. rotating or sliding wall),	
	WALL	shear stress – fluid adheres to the wall (no slip)	

Note:

For compressible flow

$$\rho_{tot} = \rho_{stat} \left[1 + \frac{\kappa - 1}{2} \text{Ma}^2 \right]^{\frac{\kappa}{\kappa - 1}} \quad (4.3.1)$$

where ρ_{tot} total pressure

ρ_{stat} static pressure

Ma Mach number $\text{Ma} = \frac{\bar{u}}{c} = \frac{\bar{u}}{(\kappa r T_s)^{0.5}}$

r specific gas constant $r = \frac{R}{M}$, M is the molecular weight

κ rate of specific heats $\kappa = \frac{c_p}{c_v}$

c sound speed in fluid

When entering a pressure condition, it is necessary to define the direction of flow with the components of the velocity or flowing in the normal direction to the boundary.

Static inlet pressure must be specified for supersonic flow.

If the flow is influenced by buoyancy forces, the pressure field and the total and static pressure input values are automatically increased by the hydrostatic pressure in the numerical calculation:

$$p' = \rho_{ref} g x_i + p \quad (4.3.2)$$

Thus the deviation from hydrostatic pressure is entered into the boundary condition, it is calculated and also evaluated. However, it is necessary to enter a reasonable reference density value at the reference temperature.

5. Conduction and convection at laminar flow

5.1. Energy equation

Energy equation is derived from the Navier-Stokes equation by scalar multiplication of velocity vector and a coefficient of 0.5. Then the equation will complement by other members expressing the internal energy

$$\begin{aligned} \iiint_V \frac{D(\rho E)}{Dt} dV &= \iiint_V \frac{\partial(\rho E)}{\partial t} dV + \iint_S \rho E (\vec{u} \cdot \vec{n}) dS = \\ &= \iint_S \lambda \nabla T dS - \iint_S \vec{\tau} dS + \iiint_V S dV \end{aligned} \quad (5.1.1)$$

$$\frac{\partial}{\partial t} (\rho E) + \nabla \cdot (\vec{u} (\rho E + p)) = \nabla \cdot (\lambda \nabla T) - \nabla \cdot (\vec{\tau} \cdot \vec{u}) + S_h$$

where the total energy $E = U + \frac{1}{2} \vec{u} \cdot \vec{u}$ is the sum of the internal and kinetic energy, λ is the coefficient of molecular thermal conductivity, the second term on the right side represents the heat generated due to friction, S_h involves chemical reactions and other heat sources.

Introduce the concept of **enthalpy** change h , which is equal to the heat that the system performs at constant pressure (if no volume work take place) and is defined by the relation

$$h = U + \frac{p}{\rho}$$

then $E = h - \frac{p}{\rho} + \frac{1}{2} \vec{u} \cdot \vec{u}$. Change of entalpy is defined for ideal gas as

$$h = \int_{T_{ref}}^T c_p dT \quad [\text{J} \cdot \text{kg}^{-1}] \quad (5.1.2)$$

and for incrompressible fluid (incrompressible gases and liquids) as

$$h = \int_{T_{ref}}^T c_p dT + \frac{p}{\rho} \quad [\text{J} \cdot \text{kg}^{-1}] \quad (5.1.3)$$

In the above enthalpy equations, the calculation is defined for the reference temperature (eg. $T_{ref} = 298.15 \text{ K}$), which can be changed according to the situation.

State function S is called entropy and is defined by formula

$$S = \int_{T_{ref}}^T \frac{dQ}{T} \quad S = \int_{T_{ref}}^T \frac{c_V}{T} dT \quad (5.1.4)$$

5.2. Mathematical model of laminar flow with conduction and convection

In the vast majority of cases, heat transfer is carried out simultaneously by conduction and convection. The mathematical model consists from following equations:

- **continuity equation** $\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{u}) = S_z$

- **momentum equation** - Navier - Stokes equations at laminar flow

$$\frac{\partial (\rho \vec{u})}{\partial t} + \nabla \cdot (\rho \vec{u} \vec{u}) = -\nabla p + \rho \vec{a} + \nabla \cdot (\tau) + S_m$$

- **energy equation** $\frac{\partial}{\partial t} (\rho E) + \nabla \cdot (\vec{u} (\rho E + p)) = \nabla \cdot (\lambda \nabla T) - \nabla \cdot (\vec{\tau} \cdot \vec{u}) + S_h$

The solution is complemented by **boundary conditions on the flow boundaries and on the wall** (see chapter above). In solution of conduction and convection we will model more areas, some of which will be defined as the flowing medium (conduction and convection), some areas will be only solid wall given by thickness where the detailed temperature distribution (conduction only) will be resolved. Then between areas there exist a special interface with special boundary conditions. On the following more or less simple examples of typical energy applications the advantage of numerical finite volume method will be demonstrated.

5.3. Boundary conditions at laminar flow with conduction and convection

In isothermal flow the boundary is meant thin wall surrounding fluid with the boundary conditions given only by fluid variables. In the case of non-isothermal flow the boundary conditions depend on each specific case, ie. if it is necessary to use the full model, partly simplified model or simplified model of heat transfer:

- full model - it solves the temperature distribution in the flowing medium, in the tube wall (conductive zone) and in the environment (air), the boundary conditions are defined by external thermal environmental conditions
- partially simplified model - it solves the temperature distribution in the flowing medium and in the tube wall (conductive zone), it is necessary to define the temperature or heat flux density on the outer wall of the tube
- simplified model - it solves the temperature distribution in the flowing medium with a defined wall boundary of zero thickness (can also respect the thermal resistance of the

specified wall thickness) with defined non-isothermal properties and boundary conditions. Different approaches are shown in Fig. 5.1

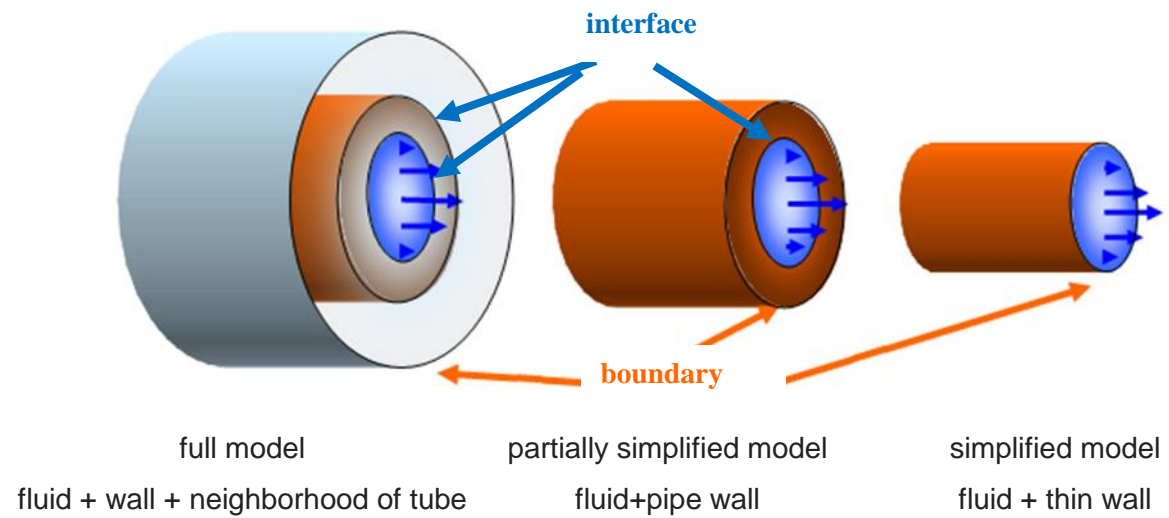


Fig. 5.1 Fluid flow in the tube ■ fluid, ■ tube wall, ■ neighborhood of tube

The next chapters will clarify the definition of special boundary conditions at the boundary of the area and at the interfaces between different material areas.

The boundary is the most distant surface of zero thickness with defined boundary conditions. On the boundary the flow conditions (chap. 4.3) and conditions of heat transfer must be specified.

	Type (Fluent)	Variable	Unit
INLET	VELOCITY INLET	velocity u temperature T	$[\text{ms}^{-1}]$ $[\text{K}]$
	MASS FLOW RATE	mass flow rate Q_m temperature T	$[\text{kgs}^{-1}]$ $[\text{K}]$
	PRESSURE INLET	total pressure $p_{tot} = p_{stat} + p_{dyn} = p_{stat} + \frac{1}{2} \rho u^2$ temperature T	$[\text{Pa}]$ $[\text{K}]$
OUTLET	OUTFLOW	condition of steady flow $\frac{\partial u}{\partial n} = 0, \frac{\partial p}{\partial n} = 0, \frac{\partial T}{\partial n} = 0$	
	PRESSURE OUTLET	static pressure p_{stat}	$[\text{Pa}]$

		(equal zero at flow into the atmosphere) temperature T	[K]
	MASS FLOW RATE	mass flow rate Q_m temperature T	[kgs ⁻¹] [K]
WALL	WALL	$u = 0$ is default (moving wall – insert velocity) temperature T constant heat flux density q zero heat flux density (insulation) $q=0$, convective heat transfer α, T_{ref} radiation	
	WALL	shear stress – fluid adheres to the wall (no slip) (can be specified shear stress and roughness for turbulent flow). temperature T constant heat flux density q zero heat flux density (insulation) $q=0$, convective heat transfer α, T_{ref} radiation	

If the heat transfer is solved, at the border there are setting the temperature conditions, see chap. 3.3 (the latter characterizes the radiation).

The interface can be simplified regarded as single - or bilateral wall zone. If the wall zone is two-sided wall, i.e. the wall forms the interface between two areas, such as the interface fluid/wall or wall/wall for the problem of heat transfer, then it is possible to model the thermal conductivity inside the boundary walls and the inner walls. Furthermore, there exists also the option to choose whether or not conditions for two-sided wall are linked (COUPLED).

Fig. 5.2 can monitor the temperature distribution in the flowing medium and at the wall when on the outer wall surface a constant temperature condition is given. Heat transfer is affected by the material, i.e. wall and water conductivity, and of course by the flow.

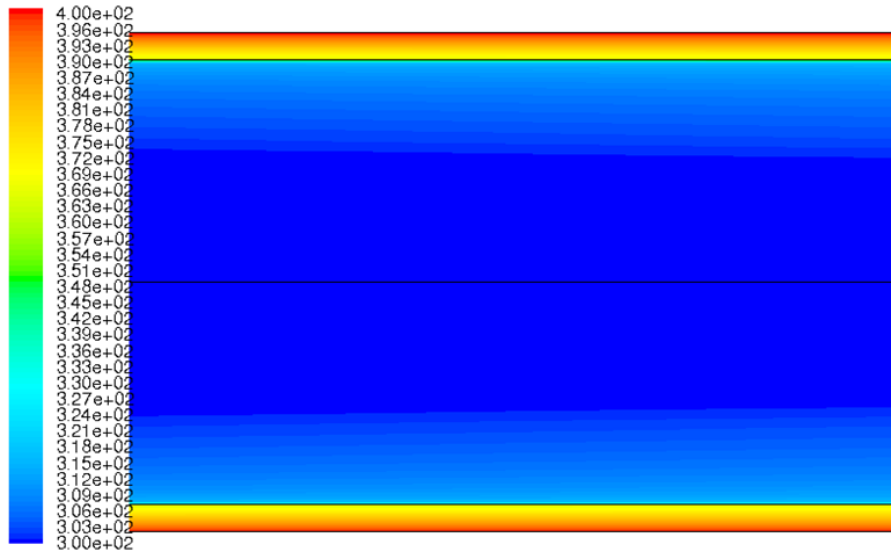


Fig. 5.2 Detail of the end of solved area with a temperature distribution for partially simplified approach.

5.3.1. Boundary conditions on thin wall

As expected, the wall has zero thickness. If the wall thickness is of the non-zero, then the parameters for calculating of the thermal resistance of the thin wall can be set and you can model a thin layer of material between two zones, eg. piece of sheet between the two zones of fluid coating solids, or the contact resistance between two solid regions. FLUENT solves the 1D conduction equation to calculate the thermal resistance defined by wall and heat generation in the wall.

In order to include these effects in the calculation of heat transfer, it is necessary to specify the type of material, wall thickness and heat generation in the wall. Thus, the material is selected and the wall thickness is specified. Thermal resistance of wall is $\frac{l}{\lambda}$, wherein λ is thermal conductivity of the wall material and l is wall thickness. Temperature condition respectively condition of heat flux density will be

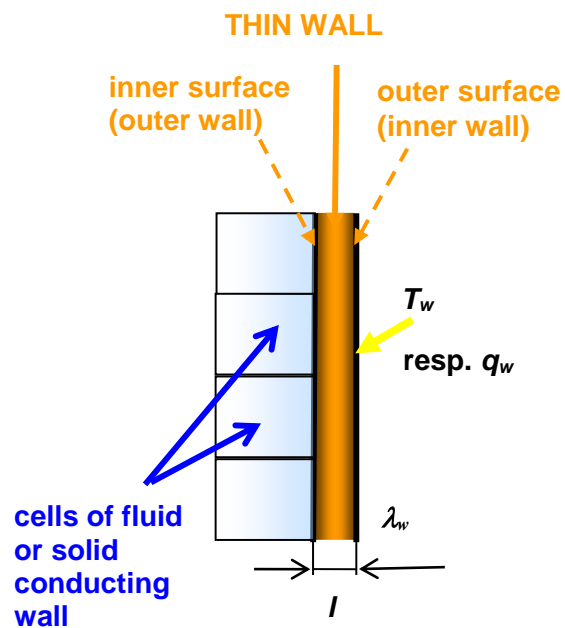


Fig. 5.3 The boundary condition on thin wall

specified on outside of the wall, as seen in Fig. 5.3. According to the convention used in Fluent the wall will be called the inner wall. T_w is constant temperature on the wall. It should be noted that for the thin wall a constant thermal conductivity is only possible to define. If required to use a non-constant thermal conductivity for a non-zero thickness, it is necessary to define a particular geometry of the wall and to mesh it.

5.3.2. Boundary condition on two – sided thin wall

If the wall has on each side a liquid or a solid wall, this wall is called two-sided wall, and is illustrated schematically in Fig. 5.4.

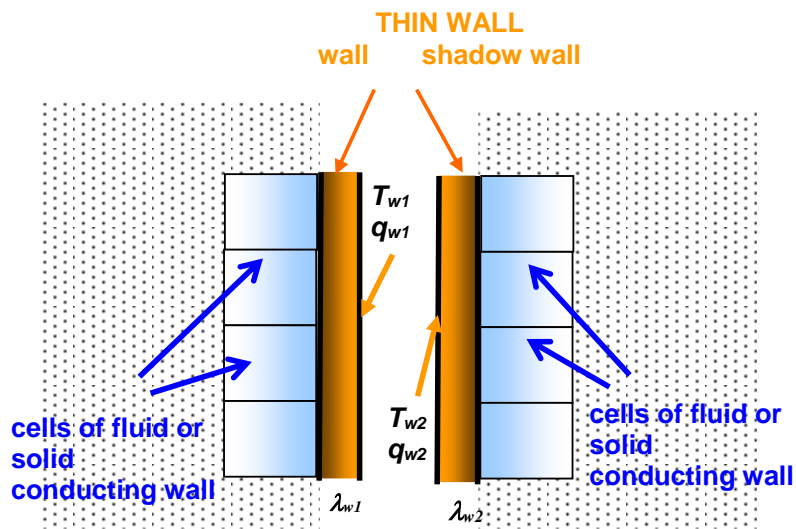


Fig. 5.4 The boundary condition on the wall with two surfaces [1]

When the mesh is loaded with this type of wall in Fluent, we automatically create a "shadow" zone so that each side of the wall is the wall zone. The panel WALL emerges as Shadow Face Zone. Then you can define different thermal conditions in each zone named WALL and SHADOW WALL, or link (coupled) both zones.:

- At linked wall zones the Coupled option in Thermal Conditions (this parameter appears in the Wall panel when the wall is two-sided) must be selected. No additional thermal boundary conditions are required, because the heat transfer is solved from the equations for neighboring cells. But one can define the type of material, thickness and heat generation for the calculation of the thermal resistance, as mentioned above. Heat resistance parameters are automatically set to "shadow" wall zone

- At different (unlinked) wall zones the different thermal conditions on each of them can be defined. You need to select Temperature or Heat Flux (Convection and Radiation are not possible for two - sided wall). Both unlinked walls may have a different roughness, and are mutually insulated. If it is necessary to specify a non-zero thickness of the walls for the unconnected zone, the thermal conditions are defined on the outer surface of non-zero walls, as shown in **Fig. 5.4**, where T_{w1} and T_{w2} are temperatures (q_{w1} and q_{w2} are the heat flux densities) defined on either side, λ_{w1} and λ_{w2} are the thermal conductivities of on non-zero unconnected walls. The gap between the walls is not part of the model and it is for illustrative purposes only included in figure.

5.4. Heat transfer coefficient

The **heat transfer coefficient** on the wall is the quantity defined by the equation

$$\alpha = \frac{q}{T_{wall} - T_{ref}} \quad [W \cdot m^{-2} \cdot K^{-1}] \quad (5.4.1)$$

where q is the convective heat flow rate, T_{wall} is the wall temperature and T_{ref} is the reference temperature that should be representative for the problem. The reference temperature can be defined as:

- constant value (Surface Heat Transfer Coefficient) is appropriately defined on the basis of the user's experience and is commonly used in engineering applications
- temperature near the wall (Wall Heat Transfer Coefficient) resulting from numerical calculation. It depends on the boundary layer and in case of modeling also on cell size. Additionally, in the case of an irregular mesh, the size of the cells may change this value significantly.

Therefore, the heat transfer coefficient will be evaluated as **Surface Heat Transfer Coefficient**. The second variant is suitable for detailed investigation of heat transfer with very fine grid.

5.5. Heat transfer at flow around plate

Following an experience of modeling of heat transfer in different materials presented in chap. 3 the same geometry with more layers of material will be used. The middle layer will be again SOLID material (steel), but around the left and right side the air in a laminar regime will flow.

Solve the temperature distribution due to conduction and convection in the layer of steel, the airflow will be on both sides. The physical model is given by the shape of the area

whose 2D scheme is shown in Fig. 5.5 and the dimensions and physical properties are in v Tab. 5.1 a Tab. 5.2.

Tab. 5.1 **Geometry of region**

thickness of region $l_{steel} = l_{air}$ [m]	0.01
height of region h [m]	0.1

Tab. 5.2 **Physical properties of material** (steel, air) at 300 K

material	steel	air
density ρ [$\text{kg}\cdot\text{m}^{-3}$]	8030	1.225
specific heat capacity c_p [$\text{J}\cdot\text{kg}^{-1}\cdot\text{K}^{-1}$]	502.48	1006.43
thermal conductivity λ [$\text{W}\cdot\text{m}^{-1}\cdot\text{K}^{-1}$]	16.27	1006.43
viscosity η [$\text{kg}\cdot\text{m}^{-1}\cdot\text{s}^{-1}$]		$1.7894\cdot 10^{-5}$

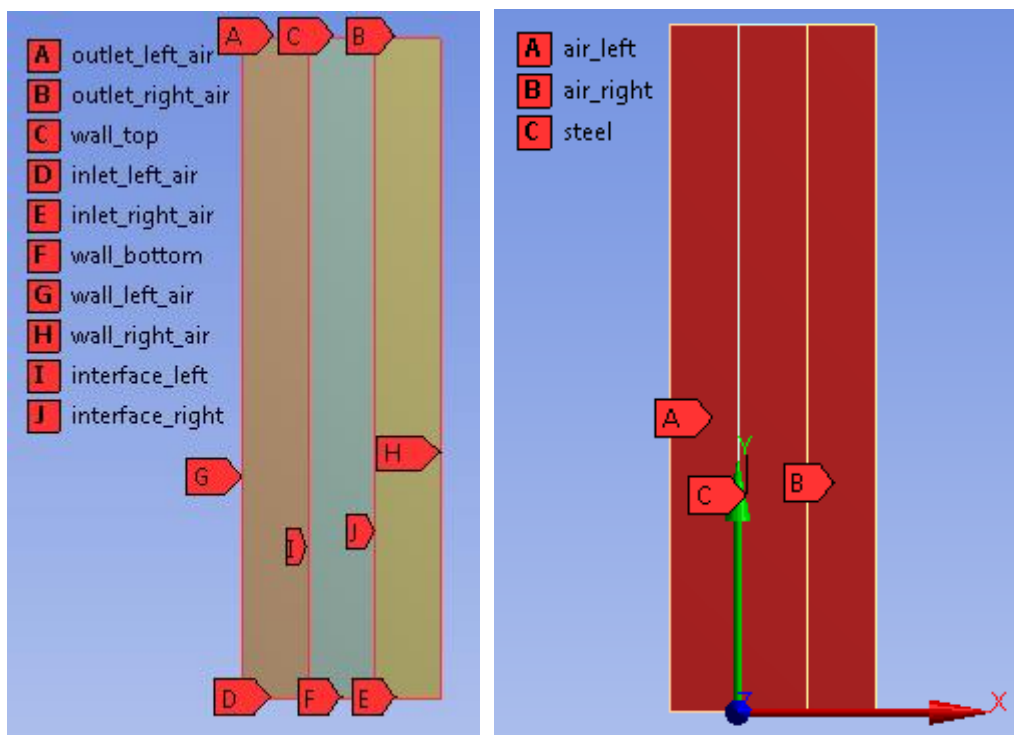


Fig. 5.5 Scheme of the plate in coordinate system and boundary conditions

Boundary conditions

Boundary conditions are defined on the left and right wall by temperature, eg. variant A from last exercise. At air inlet the velocity is defined and at outlet the static pressure is

specified. Due to the large dimensionality of the plate, the conditions of symmetry above and below are defined.

Between the flowing gas and the plate there is the interface where the heat transfer is solved. In Fluent, this interface is defined by two zero-thickness surfaces. This method is used in the flow around thin plates (compared to the solved region), radiators, etc. These objects may not be caught by the grid, thus we save the amount of cell during meshing. This duplicate surface will always appear when transferring Meshing data or another geometry creation program to Fluent. The wall name remains and, in addition, creates its shadow.

interface vs. interface shadow

On these walls the boundary conditions do not have to be entered, the heat penetrates them, they are called "coupled". This case suits the task assignment. Another option is specification of the detailed boundary conditions on both sides.

For all types of walls, the wall thickness is entered. If the wall is a boundary, its thickness is zero, if it is a real wall, a thickness (3 mm) can be entered.

Tab. 5.3 Boundary conditions

name	type (Fluent)	temperature [K]	velocity [m·s⁻¹]	pressure [Pa]
wall left	wall	323		
wall right	wall	263		
inlet left air	velocity inlet	323	0.1	
inlet right air	velocity inlet	263	0.1	
outlet left air	pressure outlet	323		0
outlet right air	pressure outlet	263		0
wall top	symmetry			
wall bottom	symmetry			
interface left	coupled			
interface left shadow	coupled			
interface right	coupled			
interface right shadow	coupled			

Estimation of dimensionless flow parameters

For calculation, formulas for flow around the plate were used, see chap. **Chyba! Nenalezen zdroj odkazů.** The calculations are made for left flow around the plate, with the right flow around the plate, the estimates are slightly different because other physical

properties of the air are used due to temperature changes.

Resultes

Reynolds number	Re=	629
Prandtl number	Pr=	0.773856
Nusselt number	Nu=	15.29314
Heat transfer coefficient tepla	$\alpha =$	40.22 W.m ⁻² .K ⁻¹

Mathematical model

In this task, there is a laminar flow, a laminar mathematical model is thus used. Velocity, pressure and temperature distribution is controlled by above differential equations. Laminarity criterion is the Reynolds number:

$$\text{Re} = \frac{u \cdot d}{\nu} = \frac{0.1 \cdot 0.1}{1.7894 \cdot 10^{-5}} = 629$$

Creating geometry and mesh

In environment Workbench the exact geometry is creating and covering by mesh, see Fig. 3.6.

Calculation results

To illustrating, the possibilities of evaluation shall be indicated, i.e. filled isolines of temperature. It can be seen that heat permeates through the steel very well but there is no linear temperature distribution across all three area. In the vicinity of the material (steel plate) there is observed typical temperature distribution when air flows around the material. The temperature is not constat in limits, because the thickness of two outer layers should be greater, plus the velocity profile was not properly evolved from rectangular to steady parabolic profile, see **Fig. 5.6**.

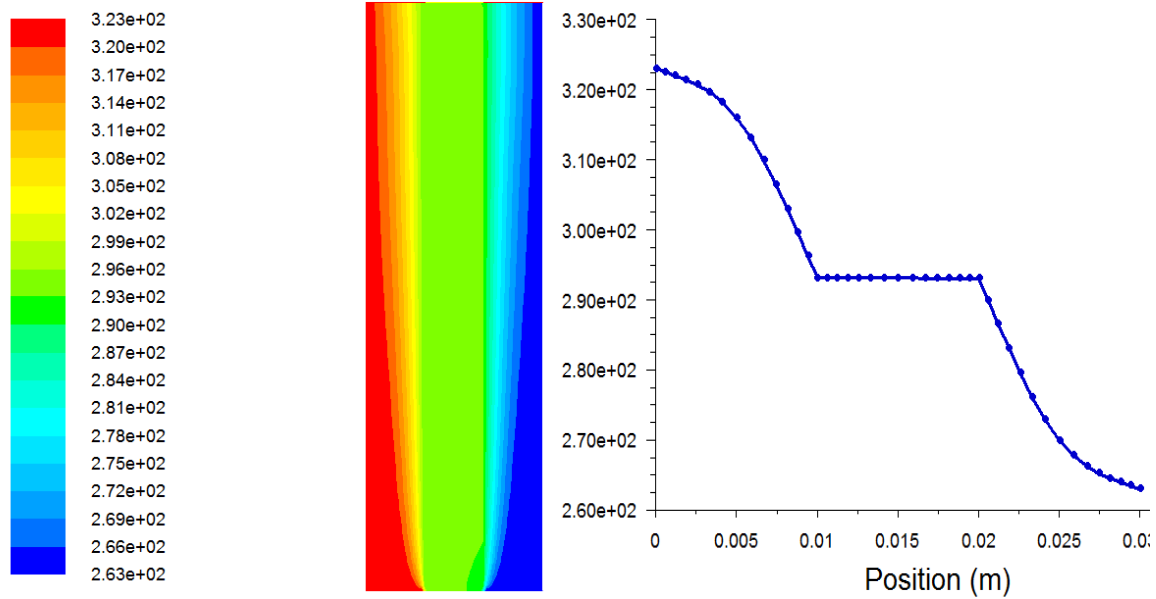


Fig. 5.6 Distribution of temperature [K] in the steel plate, in surrounding air and at the evaluation plane in the middle of the region

To calculate Nusselt number it is necessary to use the reference values. Reference temperature for the wall is determined by the left wall temperature and reference dimension is given by the length of the steel plate. First, determine an surface heat transfer coefficient according to equation:

$$\alpha = \frac{q}{T_s - T_{ref}} \tag{5.5.1}$$

which is proportional to the heat flux and subsequently Nusselt number:

$$Nu = \frac{\alpha d_{ref}}{\lambda} \tag{5.5.2}$$

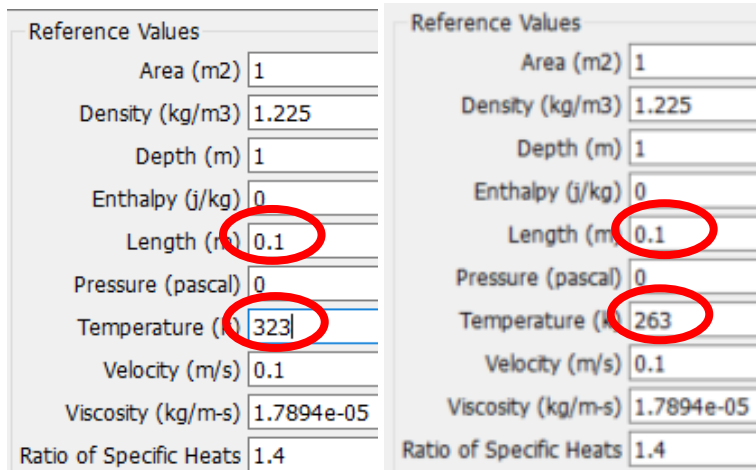
For the right wall the procedure is analogous, only the reference temperature values are different.

Reference values

left area

right area

- area of the plate wall
- air density
- length of the plate
- enthalpy
- size of the air area (Re)
- pressure
- temperature of the left wall
- air velocity at the inlet



viscosity of air
specific heat

Fig. 5.7 Definition of the reference values for calculating the parameters of heat transfer

Fig. 5.8 shows the distribution of heat flow, heat transfer coefficient and Nusselt numbers at the air - steel and steel - air interfaces, where the opposite orientation of the heat flow at the interface. It is also necessary to evaluate these values in cells on the wall, not in the middle of the cells.

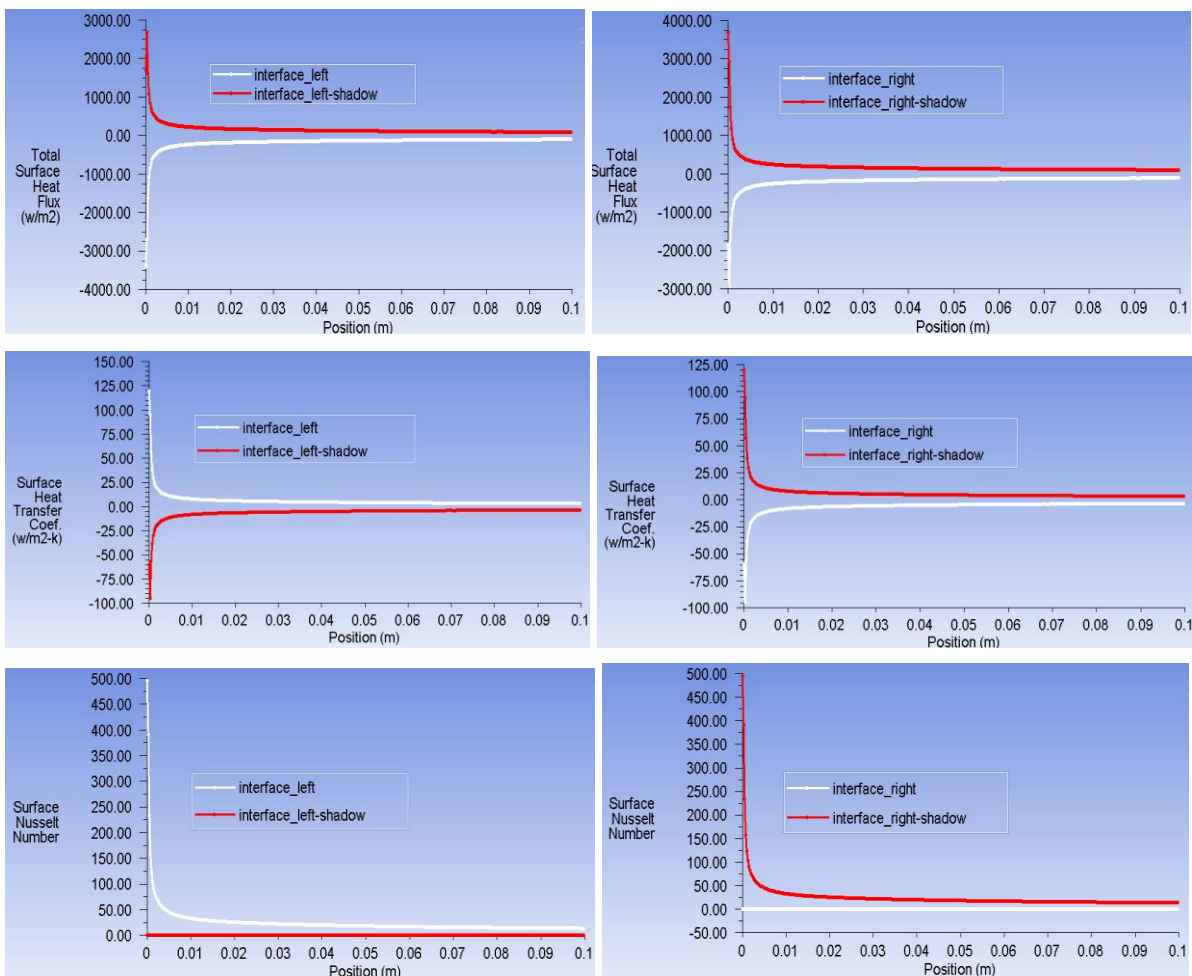


Fig. 5.8 Distribution of heat flux density, surface heat transfer coefficient and surface Nusselt number along the interfaces calculated by Fluent for $T_{ref}=323$ K and 263 K

Heat transfer coefficient and Nusselt number depend of reference values, which is practically very difficult to define because of dependency on measurement. In Fluent these values are the result and for further calculations they are not required. Nusselt number on the left interface takes two values, while on the left interface shadow the Nusselt number is equal to zero. So this wall does not make sense physically, but it's always necessary to verify if the Nusselt's number on the wall or wall_shadow is equal zero at the "coupled" condition.

Fluent software evaluates the above mentioned variables at every condition and the user must consider which values have the physical sense. CFX is treated in this sense better.

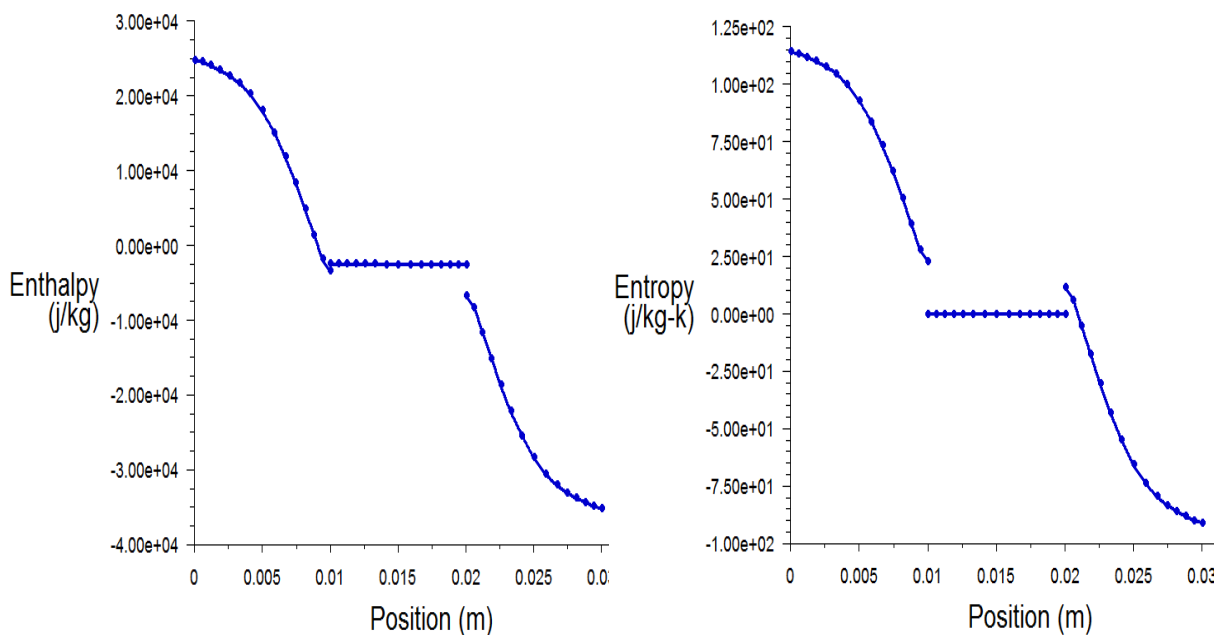


Fig. 5.9 The distribution of enthalpy and entropy along the evaluating central plane

For practical application, it is possible to obtain an average value of all the above parameters, but you need to consider accuracy of the weighted average. The weight is the area of mesh elements. Heat power per surface is determined uniquely, see Fig. 5.10.

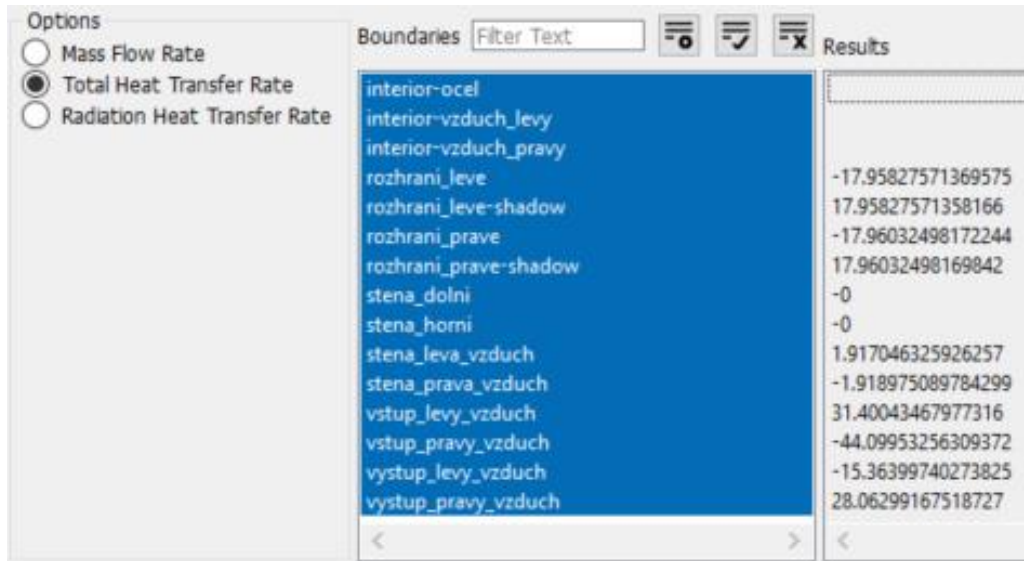


Fig. 5.10 The heat power per individual walls.

Other parameters (Total Surface Heat Flux q , Surface Heat Transfer Coefficient and Nusselt number Nu) are determined by weighted average.

Tab. 5.4 Averages (Area-Weighted Average) of heat flux, surface heat transfer coefficient and Nusselt number

	Surface Heat Transfer Coef. [W·m ⁻² ·K ⁻¹] estimation	Nu estimation	Surface Heat Transfer Coef. [W·m ⁻² ·K ⁻¹] Fluent	Nu Fluent
Wall left air			0	0
Interface left	5.14	19.5	6.00	24.8
Interface left shadow			-6.00	0
Wall right air			0	0
Interface right shadow	5.5	20.9	6.00	24.8
Interface right			-6.00	0

Differences between the estimation of the Nusselt number and the heat transfer coefficient against the Fluent calculation are due to the influence of the current acceleration and the short flow direction.

If the averaged heat transfer coefficient for the left wall is determined, it is possible to simplify the area so, that it will not be dealt with the left part of the air flow. The boundary condition on the left side of the steel will be given by heat transfer coefficient and the

reference temperature, i.e. 323 K. The same boundary condition can be applied to the right wall. The result is a temperature distribution of practically constant value, i.e. 293 K, which is the same temperature value in the steel, as on **Fig. 5.6**.

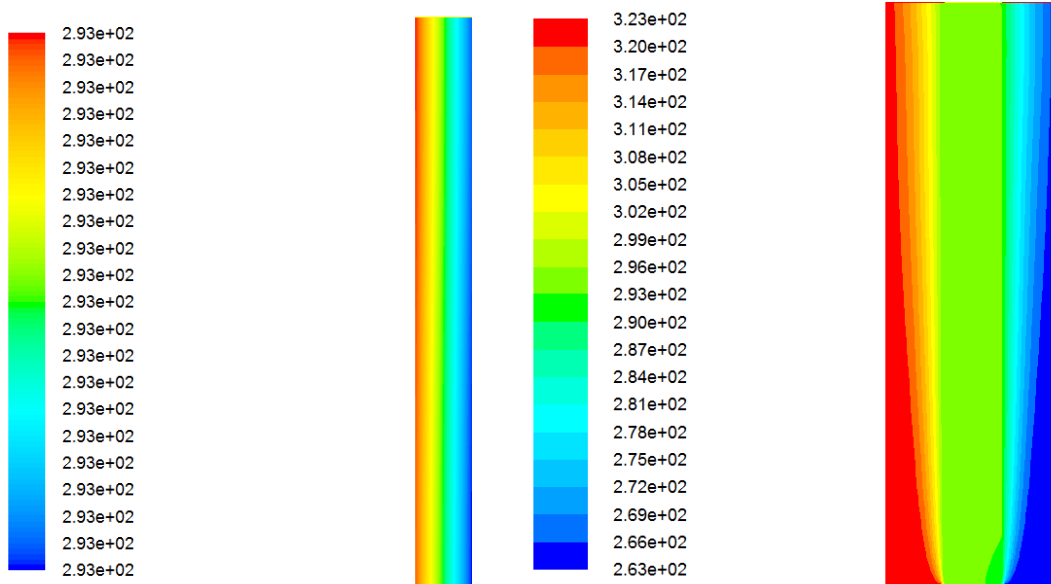


Fig. 5.11 Comparison of temperature [K] in a simplified geometry by boundary condition of heat transfer coefficient and in the geometry with flowing parts

6. Turbulence

The flow of real fluids can be classified as laminar or turbulent flow. The flow is generally called turbulent if the variables exhibit chaotic fluctuations both in space and in time, see **Fig. 6.1**. Despite turbulence randomness the detailed studies indicate that turbulent flow comprises a spatial structures which are commonly called "Eddies" (turbulent eddies).

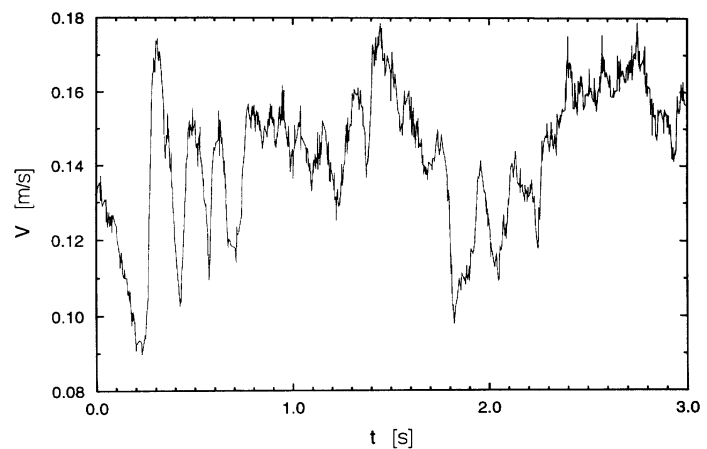


Fig. 6.1 Fully developed turbulent flow - velocity as a function of time [13]

Because in mechanical applications the turbulent structures can be characterized with dimensions of order of magnitude smaller than the characteristic dimensions of the region and changing in time of the order of ten-thousandths of a second, the detailed modeling of turbulent structures due to current hardware capabilities is very limited. Therefore, it is effort to find so called turbulent models that could reasonably filter the small turbulent fluctuations and engineering applications could deal only with great vortices. The above described equations describe both laminar and turbulent flow and for their solutions so-called „Direct solution methods“ can be used. For turbulence other approaches will be discussed, especially a time averaging method.

Modeling of turbulence is still in state of research and development, which is constantly changing with advances in the mathematical, physical and technical fields. In numerical simulation of turbulent flow, there are three theoretically different approaches that result from simplifying modifications of the initial equations describing the flow [13]

- **DNS-Direct Numerical Simulation**
- **LES-Large Eddy Simulation**
- **RANS-Reynolds Averaged Navier-Stokes equations**

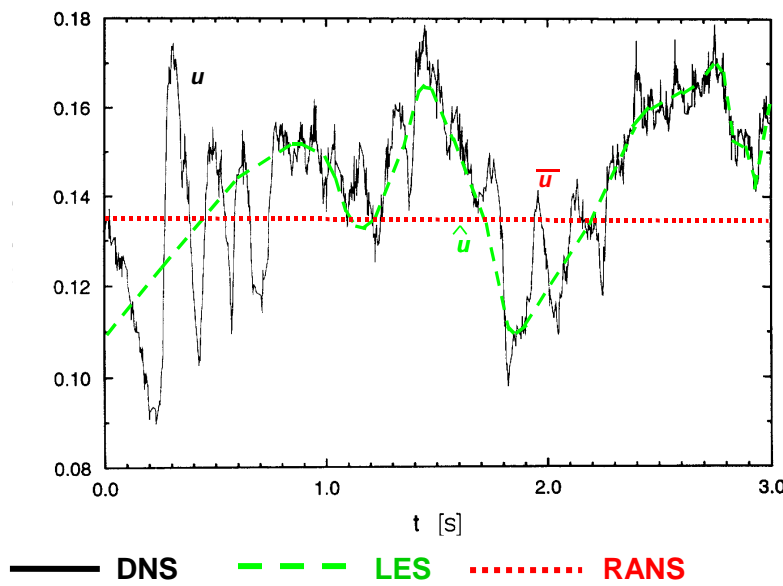
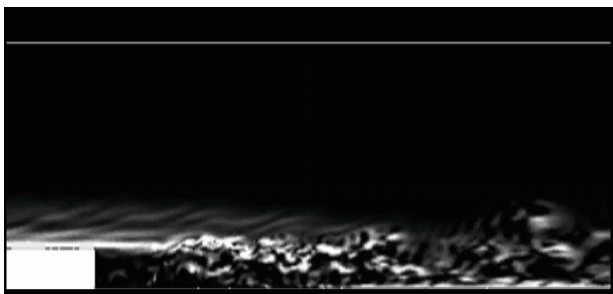

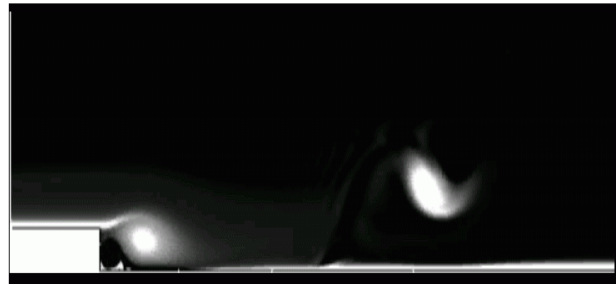


Fig. 6.2 Methods of turbulence modeling [13]

For most of the engineering tasks of turbulent flow, the most frequently used tool remains statistical turbulence models that are based on the Reynolds averaged method of turbulent flow variables and on the following procedure of time averaged balance equations (RANS).

<p>DNS is problematic because of the large capacity requirements of the computer for very fine grid. The number of grid cells can be estimated $N_p \approx Re^9 / 4$.</p>	
<p>LES is based on the modeling of large vortices as spatially time-dependent objects that can be captured by the grid. Turbulent small-scale vortices are parameterized by so-called subgrid models.</p>	

RANS are statistical models of turbulence, based on the time averaged (Reynolds) method of turbulent variables and on the following time-averaging procedure of equilibrium equations.



For example, in engineering applications it is possible to characterize turbulent structures with dimensions much smaller than the dimensions of the region and changing over time in the order of ten thousandths of a second, the modeling of all details of turbulent structures with respect to the current hardware possibilities is very limited. Therefore we will only deal with large vortices and, above all, with the method of time averaging.

6.1. Reynolds time averaging

According to O. Reynolds (1895) instantaneous values of parameters describing turbulent flow can thus be decomposed into a time averaging part $\bar{\zeta}$ and fluctuation component ζ' (see Fig. 6.3), whichever is [6]

$$\zeta = \bar{\zeta} + \zeta' \quad (6.1.1)$$

where $\bar{\zeta} = \frac{1}{T} \int_0^T \zeta d\tau$ $\zeta' = 0$ resp. $\bar{\zeta} = \frac{1}{N} \sum_i \zeta_i$. For calculation with the time-averaged quantities there are certain rules, see **Fig. 6.3**, [6].

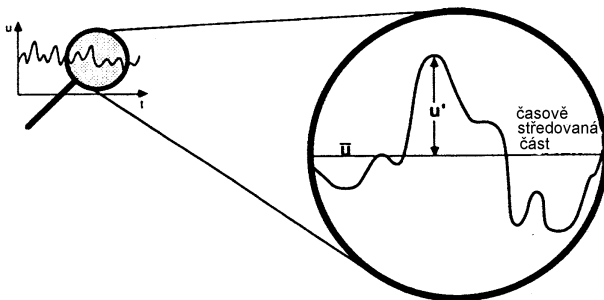


Fig. 6.3 Fluctuation and time averaging part

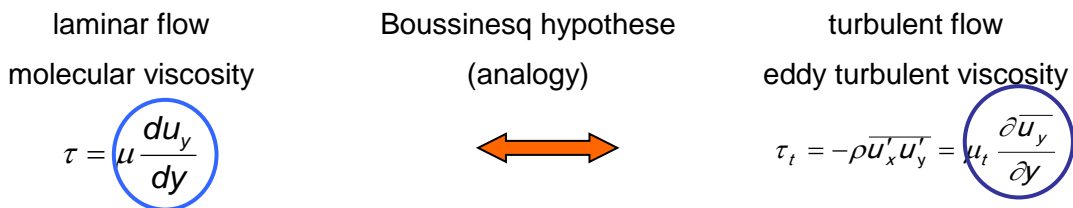


Osborn Reynolds

Applying time averaging to the basic continuity equation and Navier-Stokes equations so called Reynolds equations are obtained. They are characterized in that they are formally similar to default equations, but solved variables are time-averaging. Due to nonlinearity of the Navier Stokes equations the additional member appears in them and corresponds to the stress and is defined as

$$\tau_{t-ij} = -\rho \overline{u'_i u'_j} \tag{6.1.2}$$

The number of members is nine for various indexes and they are called Reynolds (turbulent) stresses that exist only in turbulent flow. They manifest themselves as a viscous deformation by stress effects on elementary fluid volume. Turbulent stresses are new, unknown variables in the equation system, therefore it is necessary to define them. The most commonly used method is a method of Boussinesq hypotheses about “eddy (vortex)” turbulent viscosity. This hypothesis assumes that (as in simplified two-dimensional laminar flow the shear stress Newtonian relationship is used) turbulent stresses and turbulent fluxes are proportional to the gradient of time averaging velocity, temperature, concentrations, etc.



6.2. k-ε two-equation model of turbulence

In the heat exchangers tasks the most general equations expressing the conservation laws must be considered. Additionally non-constant density, which may be dependent on temperature and pressure, both for gases and liquids, can be supposed. You can specify an external volume forces. These equations can be expressed as in integral form, so in differential form. Differential form is common form in the theory of fluid mechanics, so that the equations will be presented in this form. But according to the previous chapter the equations will be solved by finite volume method based on integral form. As already noted, we assume turbulent flow, thus equations will be defined for time averaging quantities (pressure, velocity). As already mentioned, turbulent flow is assumed, ie the equations will be defined for the mean values (pressure, velocity) and will formally be analogous to the laminar equations:

- **continuity equation valid for time averaging variables**

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{u}) = S_z \tag{6.2.1}$$

- **three Reynolds equations** for the transfer of momentum for time averaging variables (Navier - Stokes equations modified by time averaging)

$$\frac{\partial(\rho \vec{u})}{\partial t} + \nabla \cdot (\rho \vec{u} \vec{u}) = -\nabla \bar{p} + \rho \vec{a} + \nabla \cdot (\vec{\tau}_t) + \mathcal{S}_m \quad (6.2.2)$$

- **equation for turbulent kinetic energy** $k = \frac{1}{2}(\overline{u_1'^2} + \overline{u_2'^2} + \overline{u_3'^2})$, while it is possible to take into account the production of turbulent kinetic energy due to the tension and lift forces

$$\frac{\partial}{\partial t}(\rho k) + \nabla \cdot (\vec{u} k) = \nabla \cdot \left(\frac{v_t}{\sigma_k} \nabla k \right) + \rho P - \rho \varepsilon \quad (6.2.3)$$

- **equation for turbulent dissipation rate** $\varepsilon = \nu \frac{\partial u_i' \partial u_i'}{\partial x_j^2} = C_D \frac{k^{3/2}}{l}$, where l is the turbulent length scale

$$\frac{\partial}{\partial t}(\rho \varepsilon) + \nabla \cdot (\vec{u} \varepsilon) = \nabla \cdot \left(\frac{v_t}{\sigma_\varepsilon} \nabla \varepsilon \right) + C_{1s} \rho P - C_{2s} \rho \frac{\varepsilon^2}{k} \quad (6.2.4)$$

- in the model the number of constants determined empirically is also used

To complement the Reynolds stresses $\overline{u_i' u_j'}$ are defined according Boussinesq hypotheses by relationship

$$-\rho \overline{u_i' u_j'} = \mu_t \frac{\partial \overline{u_i}}{\partial x_j} \quad (6.2.5)$$

where turbulent (eddy) viscosity μ_t is expected as a function of length and velocity scale according to the Kolmogorov-Prandtl hypothesis:

$$\mu_t \approx l \bar{u} = \rho C_\mu \frac{k^2}{\varepsilon} \quad (6.2.6)$$

In some applications, the basic model can be extended to other equations focused on mass fractions of chemicals and compounds, etc.

The standard $k-\varepsilon$ model is suitable for high turbulent flow. For low Reynolds number so called RNG $k-\varepsilon$ model is preferably used. In Fluent there are many other turbulent models, each of which is recommended for a different type of flow. Nevertheless, it is necessary to have a physical experiment to verify at least some flow parameters. All turbulent models near the walls utilize the wall functions to approximate the turbulent velocity profile near the wall.

6.3. Boundary conditions for k-ε turbulent model

6.3.1. Turbulent variables on boundary

Great importance in connection with initial boundary condition is the setting of turbulent parameters in the form of values of **turbulent kinetic energy** and **dissipation rate**. A more accurate statement of these values is, of course, the profile obtained from empirical data or from empirical formulas. If the profile is not precisely known, you can enter a constant value estimated on the basis of experience. These turbulent quantities can be determined by means of quantities more easily identifiable such as turbulent intensity, ratio of turbulent and molecular viscosity, hydraulic diameter and turbulent length scale. The size of turbulent fluctuations usually is described by intensity of turbulence. Assuming isotropic turbulence ($\sqrt{u_1'^2} = \sqrt{u_2'^2} = \sqrt{u_3'^2}$) the **relative turbulent intensity** is explained as the ratio of **rms fluctuation velocity component to the mean velocity** in the same point of the flow usually expressed as a percentage. Usually only one directional component is measured:

$$I = \frac{\sqrt{u_1'^2}}{u_1} \quad (6.3.1)$$

Ordinary turbulent flow is an anisotropic flow (heterogeneous in coordinate directions), but the anisotropy is small. The large differences are between the longitudinal and transverse components of motion. In general the turbulent intensity is specified by formula:

$$I = \sqrt{\frac{u_j' u_j'}{3\bar{u}u}} \quad (6.3.2)$$

The difference between the velocity fluctuations in transverse direction u_2' and u_3' is usually very small. The value of turbulent intensity in several cases is approximately given in

Tab. 6.1 :

Tab. 6.1

	I [%]
windtunnel	0.05%
turbulent flow generated by grate	1-5%
wake	2-10%
flow in the boundary layer and in the flow in the tube	5-20%

drowned stream	20%
recirculation flow with low velocity \bar{u}	100%

Turbulent scale l is limited by the size of the region, because the turbulent vortices can not be larger than the dimension of region. Approximate value of the turbulent scale is determined from the relation $l = 0.07L$, where L is the characteristic size or hydraulic diameter. Turbulent intensity and hydraulic diameter are available variables that can be specified as boundary conditions, the others are then converted by the following relationships.

Tab. 6.2

turbulent intensity	$I = \sqrt{\frac{u_j' u_j'}{3\bar{u}\bar{u}}}$
turbulent scale	$l = 0.07L$
rate of turbulent viscosity $\frac{\mu_t}{\mu}$	$\bar{v} = \sqrt{\frac{3}{2}\bar{u}.I.I}$
turbulent kinetic energy	$k = \frac{3}{2}(u')^2$ nebo $k = \frac{3}{2}(\bar{u}I)^2$
dissipation rate	$\varepsilon = C_\mu^{\frac{3}{4}} \frac{k^{\frac{3}{2}}}{l} = \rho C_\mu \frac{k^2}{\mu} \left(\frac{\mu_t}{\mu}\right)^{-1}$

Of course, turbulent energy and dissipation rate can also be defined directly. Depending on the complexity of the mathematical model other variables associated with heat transfer or other scalar variables can be defined. The value of turbulent intensity in the case of LES is defined by a random velocity fluctuations at the input.

6.3.2. Variants of turbulent boundary conditions

In any case, two boundary conditions must be entered because of number of equations for turbulent quantities (two). It is possible to select from the following combinations:

- Turbulent kinetic energy + turbulent dissipation
- Turbulent intensity + turbulent length scale
- Turbulent intensity + viscosity rate

- Turbulent intensity + hydraulic diameter

The most commonly used variant is the last variant because the turbulent values can be qualified. If another measurement is available, another variant can be used.

6.3.3. Boundary conditions

On flow boundaries the conditions are the same as the boundary conditions for laminar flow and, in addition, turbulent conditions are added to each variants, with the exception of the reverse flow conditions, otherwise they are counted, see Tab. 6.3.

Tab. 6.3

	Type (Fluent)	Variable	Unit
INLET	VELOCITY INLET	velocity u	[ms ⁻¹]
		turbulent intensity I	[%]
		hydraulic diameter d_h	[m]
	MASS FLOW RATE	mass flow rate Q_m	[kgs ⁻¹]
		turbulent intensity I	[%]
		hydraulic diameter d_h	[m]
	PRESSURE INLET	total pressure	[Pa]
		$p_{tot} = p_{stat} + p_{dyn} = p_{stat} + \frac{1}{2} \rho u^2$	
		turbulent intensity I	[%]
		hydraulic diameter d_h	[m]
OUTLET	OUTFLOW	steady flow condition	
		$\frac{\partial u}{\partial n} = 0, \frac{\partial p}{\partial n} = 0$	
		turbulent intensity I	[%]
		hydraulic diameter d_h	[m]
	PRESSURE OUTLET	static pressure p_{stat}	[Pa]
		(0 at outlet into atmosphere)	
		turbulent intensity I	[%]
		hydraulic diameter d_h	[m]
	MASS FLOW RATE	mass flow rate Q_m	[kgs ⁻¹]
		turbulent intensity I	[%]
		hydraulic diameter d_h	[m]

WALL	WALL	$u = 0$ default (moving wall – define velocity) roughness for turbulent flow	
	WALL	shear stress or fluid sticks on the wall (no slip) roughness for turbulent flow	

6.4. Wall function, possibility of more accurate calculation

Modeling of the flow near the wall affects the accuracy of numerical solutions in the whole area. Near the wall the solved variables are rapidly changing, significantly the transfer of momentum and scalar quantities is here applied. Turbulence is suppressed close to the wall. However, in the outer part of boundary layer there is a significant production of turbulent kinetic energy due to Reynolds stresses and the mean velocity gradient. Numerous experiments have shown that the area near the wall, so called **boundary layer** can be divided into multiple parts. Immediately near the wall the **viscous (laminar) sublayer** is placed, the flow is here nearly laminar and molecular viscosity has a dominant influence on the transfer of momentum, heat and mass. The outer part of the boundary layer becomes **fully turbulent layer** and turbulence plays a dominant role here. Between laminar and fully turbulent layer the **transition layer** occurs, in which the effects of molecular viscosity and turbulence apply equally. Distribution of the boundary layer is shown in **Fig. 6.4**.

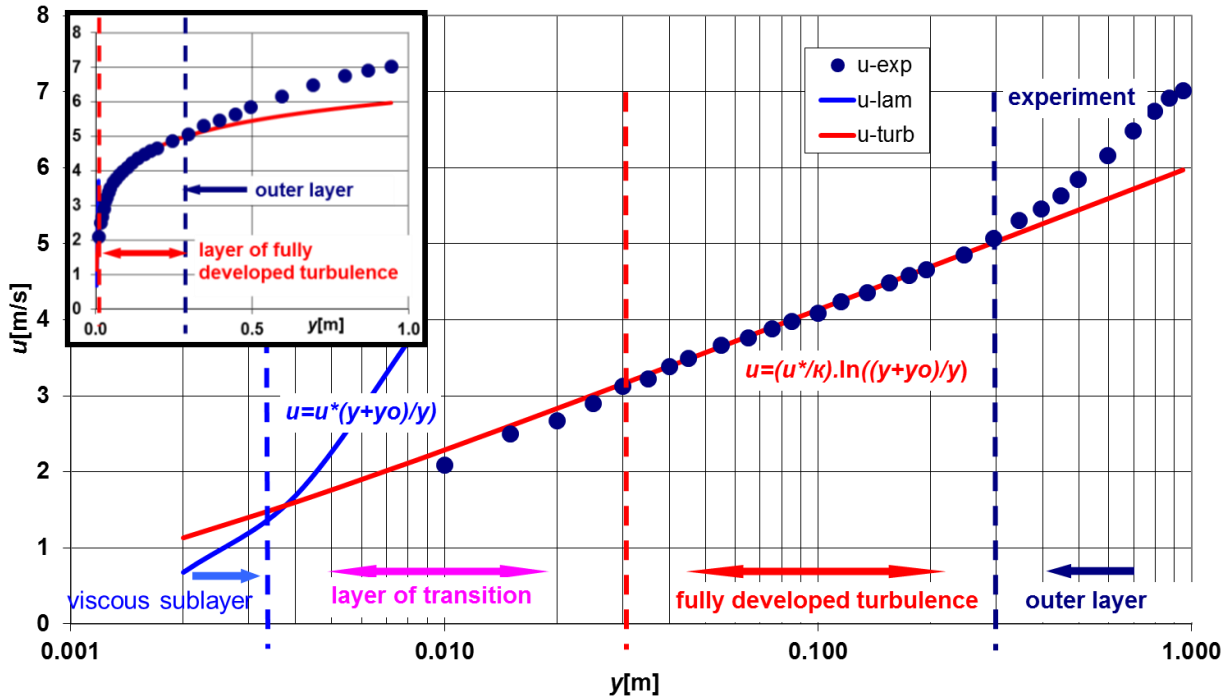
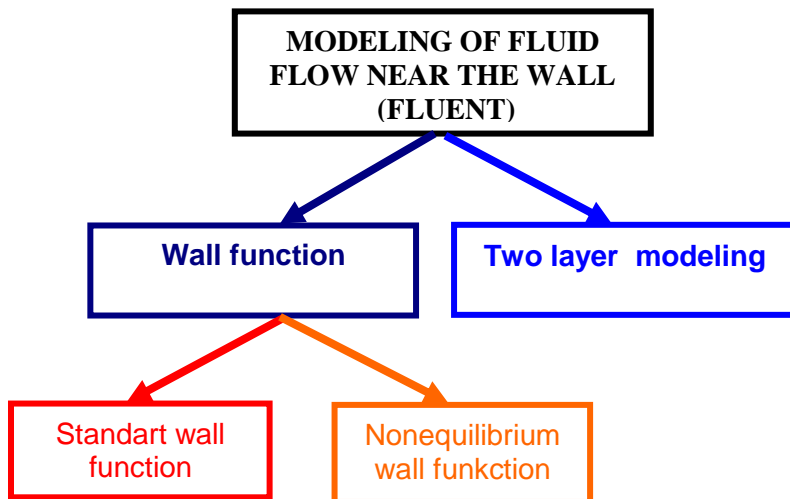


Fig. 6.4 Distribution of layer near the wall - in linear and logarithmic coordinates

Flow near the wall can be modeled in two approaches:



- **using of wall functions (coarsen mesh)** by which the region between laminar sublayer and transition layer is spanning, where the molecular and turbulent viscosity appears, ie the area between the wall and the area of fully developed turbulent flow
- **two layer modeling for finer mesh (Enhanced wall teratment)** includes the viscous sublayer in connection with the finer mesh near the wall.

The essence of both approaches is illustrated on Fig. 6.5.

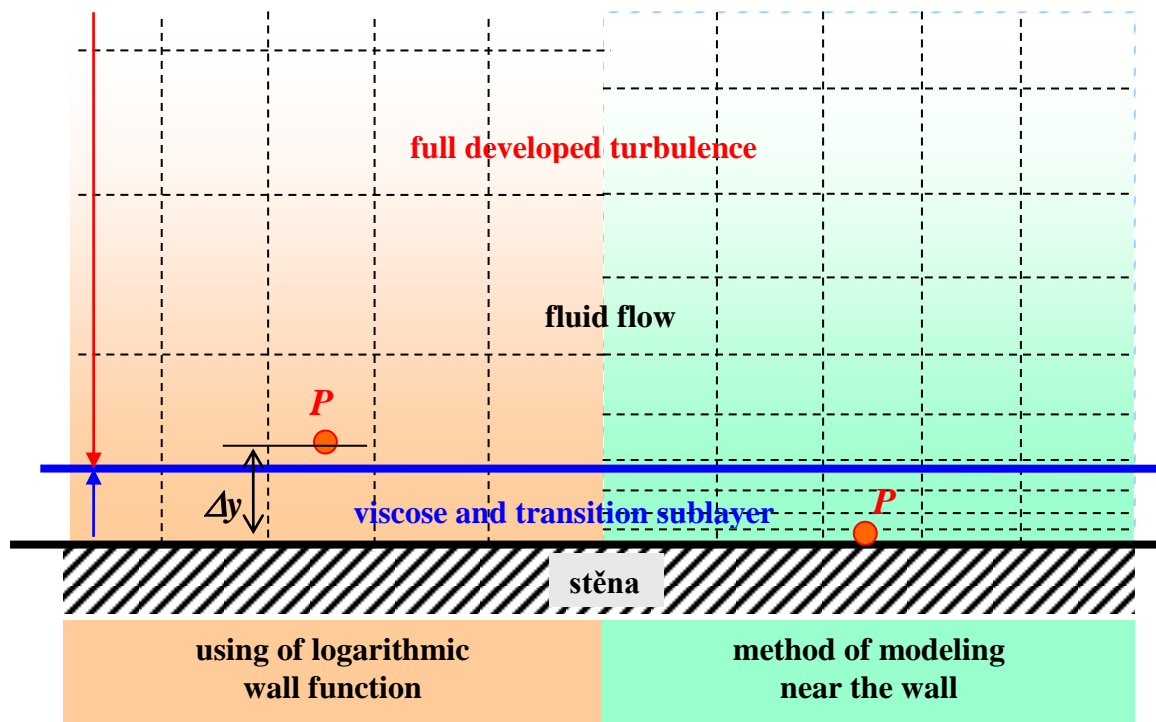


Fig. 6.5 Two approaches to the modeling of flow near the wall in Fluent

Theory of wall functions by Launder a Spalding

Wall functions based on Launder and Spalding theory are widely used mainly in industrial applications. In turbulent flow the boundary layer consists of a viscous sublayer and the logarithmic law for time averaging velocity in the turbulent field in a simplified two-dimensional case:

$$\bar{u}^+ = \frac{1}{\kappa} \ln(E \cdot y^+) \quad (6.4.1)$$

Dimensionless variables in this equation are defined as:

$$\bar{u}^+ = \frac{\bar{u}}{u_\tau} \quad y^+ = \frac{\rho u_\tau y}{\mu} \quad u_\tau = \sqrt{\frac{\tau_w}{\rho}} \quad (6.4.2)$$

where

- κ = von Kármán constant (=0.42)
- E = empirical constant (=9.81)
- \bar{u} = time averaging flow velocity in point P
- u_τ = friction velocity
- y = difference of point P from the wall in normal direction
- μ = dynamic viscosity of fluid

Friction velocity u_τ is determined by shear stress defined as Reynolds stress.

Wall functions are a set of empirical relationships and functions that allow you to "link" variable solved in a cell near the wall with the corresponding value on the wall. Wall functions include logarithmic law for mean velocity and temperature and equations for turbulent quantities near the wall.

6.5. Influence of mesh quality on the choice of wall functions for various models of turbulence

The dimensionless distance of the center of cells adjacent to the wall from these walls is determining if the choice of the logarithmic wall function is correct or a second approach is to be selected.

- logarithmic law is valid for $y^* > 30 \div 60$
- enhanced wall treatment is valid for $y^* < 4 \div 5$, ideally at least 10 cells should be in the laminar sublayer
- Spalart Allmaras model uses wall logarithmic function assuming a very fine grid ($y^* = 1$) or grid for which it is $y^* \geq 30$.
- Large Eddy Simulation model uses a logarithmic wall function for very fine grid (order of $y^* = 1$)

Determination of dimensionless distance y^* is possible up in Fluent, so mesh refinement is carried out to find the command ADAPT. Boundary layer of velocity and temperature profile will then be better detected and will lead to a more accurate calculation of heat transfer between the wall and the fluid. For illustration, in examples using adapted and coarse grid the differences in heat flux and other variables will be also evaluated.

6.6. Choice of turbulent model for more accurate calculation

The basic problem of the turbulent shear flow calculation lies in the presence of unknown Reynolds stresses in the equations describing the motion of fluid medium, so that the system of equations is not closed as in the case of laminar flow. Set of additional equations and empirical relationships, which together with the equations of motion forms a solvable system of equations, is called a model of turbulence. Selection of turbulent model depends on the type of flow:

- level of turbulence, which is determined by the Reynolds number. At high Reynolds number (10^5 in the order) it is developed turbulent flow and the standard $k-\varepsilon$ model is

used. At lower Reynolds numbers it will be appropriate another variant, for example. RNG $k-\varepsilon$ model or $k-\omega$ model.

- simple flow vs. swirl flow in the field. At the existence of a secondary swirl flow it is again advisable to use RNG $k-\varepsilon$ model or $k-\omega$ model.
- calculation of heat transfer. In the tasks of heat transfer $k-\omega$ model is useful.
- computational speed. The fastest and most stable calculation is by using the standard $k-\varepsilon$ model.

Turbulence models can be divided into several groups. For simplicity there are the most commonly used models, whose selection is determined by the Reynolds number and speed of calculation.

Tab. 6.1

$k-\varepsilon$ model	high Re number
RNG $k-\varepsilon$ model	low Re number
$k-\varepsilon$ available model	
$k-\omega$ model	low Re number + heat transfer

6.7. Flow around the tube in transverse direction

6.7.1. Flow around the tube - theory, measurement

Flow around the tube and subsequently flow through tubular bundle is one of the basic fluid flow and that is a typical problem for a range of heat exchangers. In the opening chapters these issue will be discussed on a steady flow around one tube with heat transfer and then on steady flow through tubular bundle.

When solving tasks flow around the tube it can be evaluated in addition to basic physical quantities such as velocity, pressure and their statistical processing also Reynolds number, Strouhal number (frequency of the largest floating vortical structures), resistance coefficients, location of the boundary layer separation, or the length of the wake [17].

Effect of actual flow field of viscous fluid flow around the body depends on the value of **Reynolds number** $Re = \frac{ud_h}{\nu}$. The basic distribution of flow character around the tube at different Reynolds numbers was established experimentally by Roshko [9]. He divided the flow around the tube, depending on the Reynolds number in the following areas:

Tab. 6.4

$40 < Re < 150$	stabil area
$150 < Re < 300$	transient area
$300 < Re < 200\ 000$	instabil area

More detailed separation is yet problematic due to the character of turbulence. Further examination of the parameters is given in the literature.

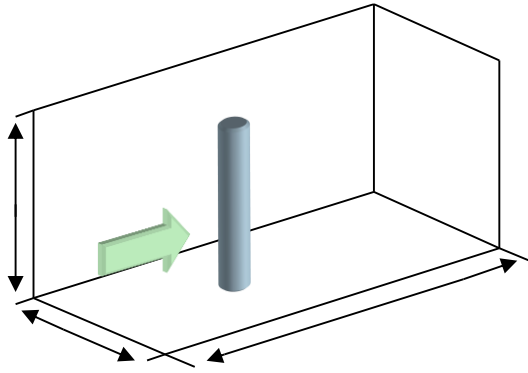
Strouhal number specifies the flow dynamics, ie. the frequency of vortex shedding

$$Sh = \frac{f d_h}{u} \quad (6.7.1)$$

The value $Sh = 0.2$ permits for a given geometry (diameter) and for the physical properties of the flowing medium (viscosity) to determine the frequency of vortex shedding. It follows that the solution is time-dependent, i.e. at each time step of period given by relationship

$T = \frac{1}{f}$ the flow field is different. This is in terms of numerical solution and also in terms global determining of heat transfer disadvantageous. We also can not forget the time dependence, since the stationary task does not converge, but converges at each time step, which is eg. hundred-th of period. But there is a possibility to find a solution statistically averaged, and thereby estimate the basic parameters of heat transfer. This is a complicated process, but the only one possible. Furthermore, this procedure corresponds to experimental measurements.

Measurement of such time-dependent processes can be carried out by series of measuring instruments whose outputs can be recorded as a time series into a computer. They include hot-wire anemometer CTA, Laser - Doppler anemometer LDA, Particle Image Velocimetry PIV etc. In this case to measuring the air flow field behind the tube the device Mini-CTA was used. To determine the parameters of the flow field the measurements were taken at certain points behind the tube (Fig. 6.6), the profile of the mean velocity and turbulence intensity was drawn (Fig. 6.7) and further the vortex shedding frequency from a time series with the FFT method was evaluated (Fig. 6.8). The frequency of vortex shedding is $f = 105$ Hz.



Scheme of measurement [17]

Physical experiment:

tube diameter: 20 mm

air temperature: 22 °C

air density: 1.225 kg·m⁻³

viscosity of the air: 1.7894·10⁻⁵ Pa·s

The flow parameters at the inlet into the measuring part of the tunnel:

air velocity: 10 m·s⁻¹

intensity of turbulence: 1.5 %

Fig. 6.6 Scheme of the measuring section and flow parameters

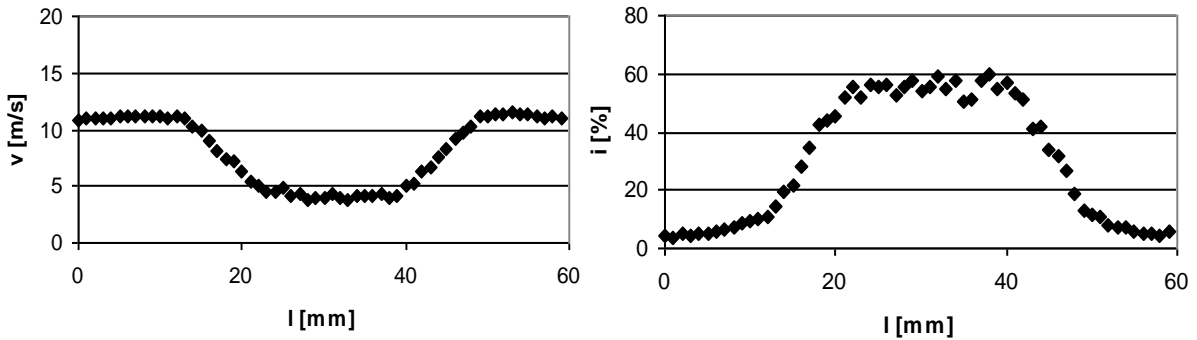


Fig. 6.7 Distribution of the mean velocity and turbulent intensity in cross-section 40 mm behind the tube [17]

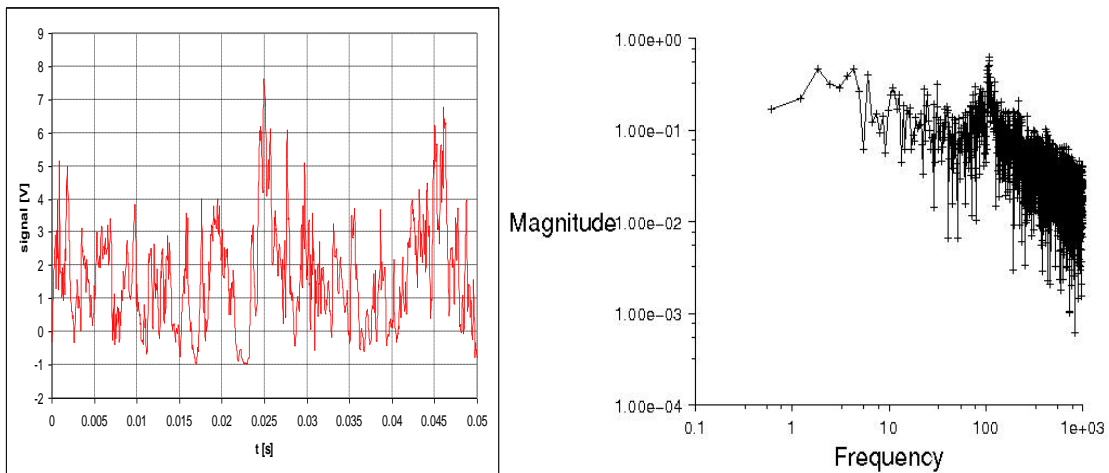


Fig. 6.8 Time velocity record measured at a point 40 mm behind the tube and 10 mm next the tube axis and the power spectral density [17]

6.7.2. Flow around the tube - numerical solution

For an easy testing the task in longitudinal section through the center of the dimension was solved. To obtain satisfactory precision the symmetrical mesh along the axis of symmetry and most significantly compressed to the tube wall has been formed. This is a necessary condition not only for flow around obstacles, but also for heat transfer, see Fig. 6.9.

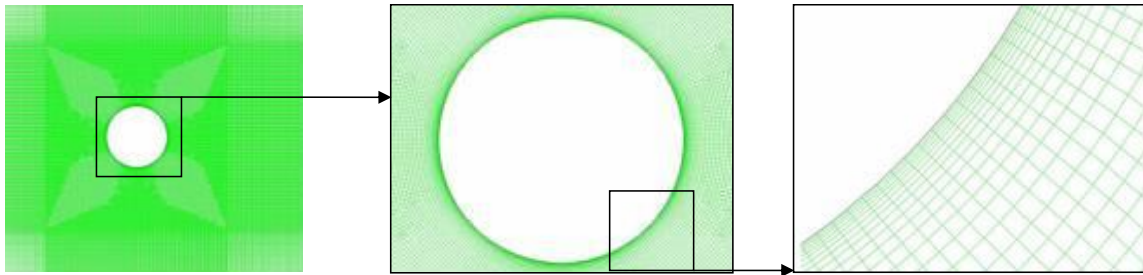


Fig. 6.9 Creating geometry and mesh of area [17]

Stationary boundary conditions were set according to the physical experiment, see Tab. 6.5.

Tab. 6.5

diameter of the inserted tube [mm]	20
air velocity [$\text{m}\cdot\text{s}^{-1}$]	10
ambient air temperature [$^{\circ}\text{C}$]	22
air density [$\text{kg}\cdot\text{m}^{-3}$]	1.225
air viscosity [$\text{Pa}\cdot\text{s}$]	$1.7894\cdot 10^{-5}$
intensity of turbulence at inlet into measuring part of the tunnel [%]	1.5

The series of mathematical models to get high-quality results for a confrontation with experiment was tested. To illustrate here the RNG k - ϵ turbulent model was chosen. The results were evaluated by means of instantaneous and time averaging values, see **Fig. 6.10**. Using better quality of numerical models and modeling in 3D geometry, the numerical calculation is even more complicated as the flow exhibits swirl structures even in the direction of the axis of the pipe, see **Fig. 6.11**. When evaluating the power spectral density, however, the result is much more consistent with the experiment (**Fig. 6.12**).

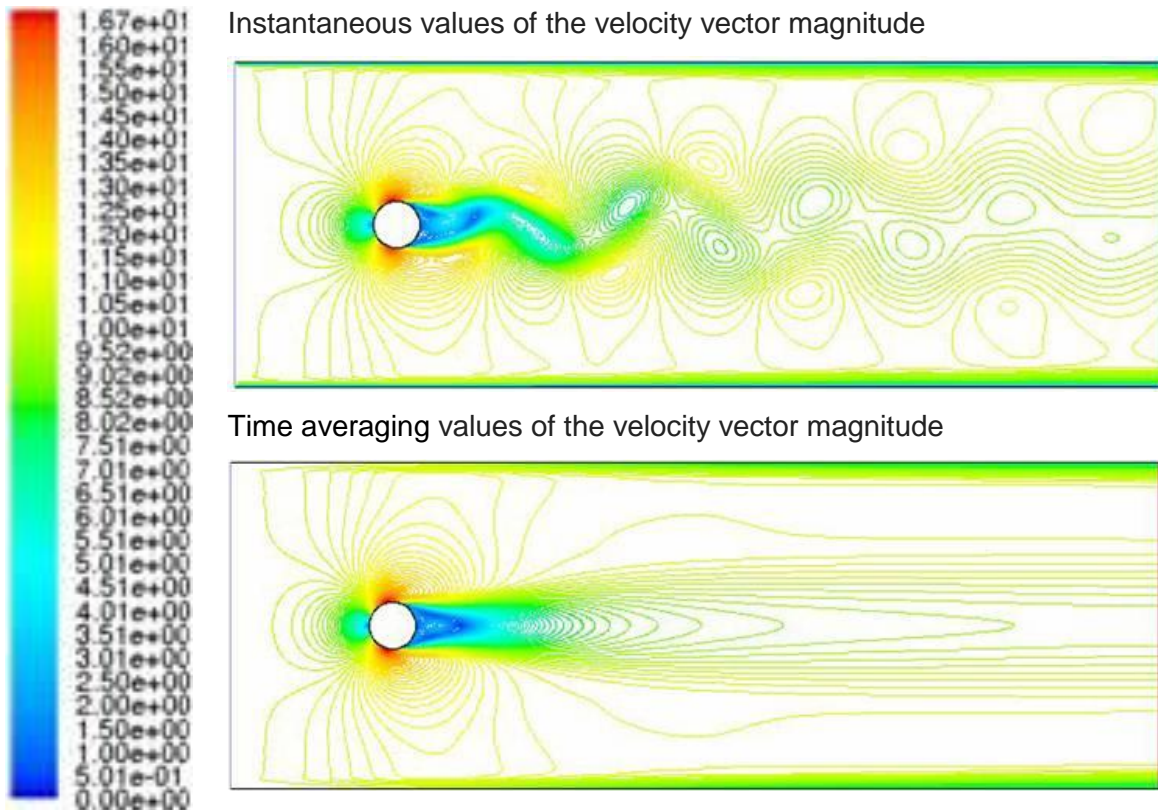


Fig. 6.10 Velocity magnitude [17]

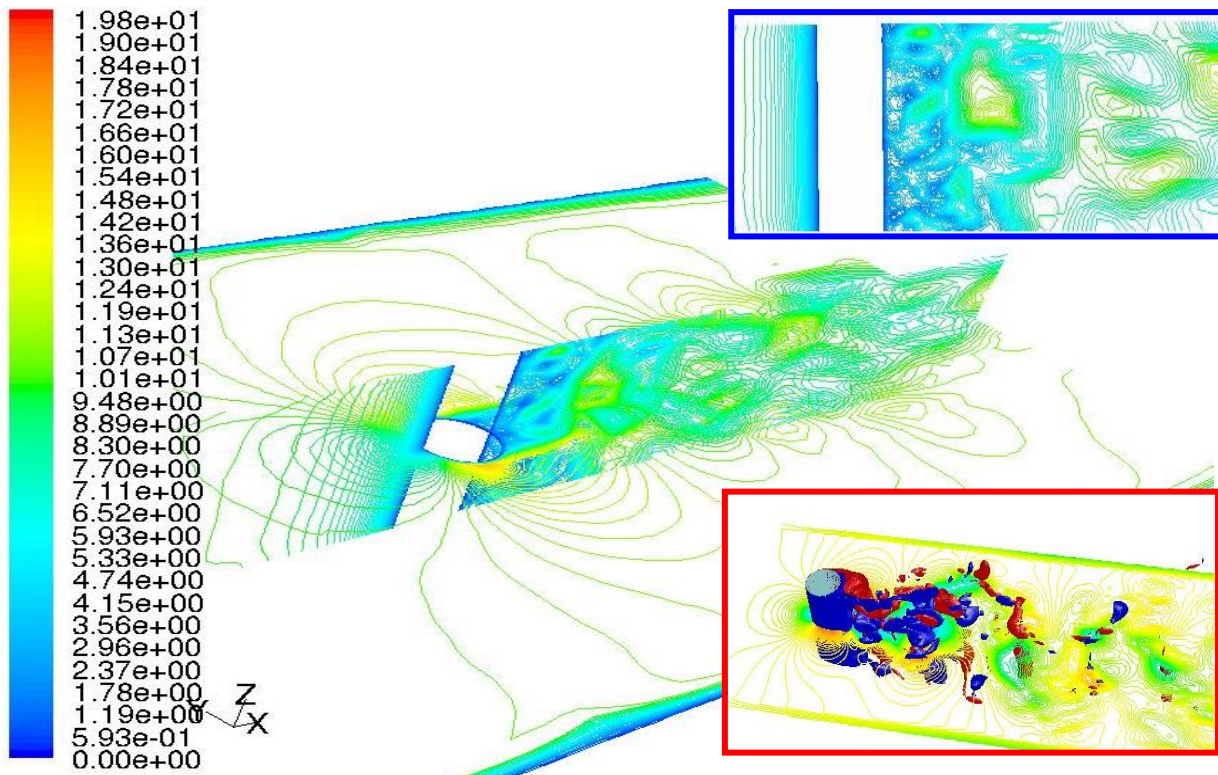


Fig. 6.11 3D model – velocity vector magnitude in the axis, detail and space view [17]

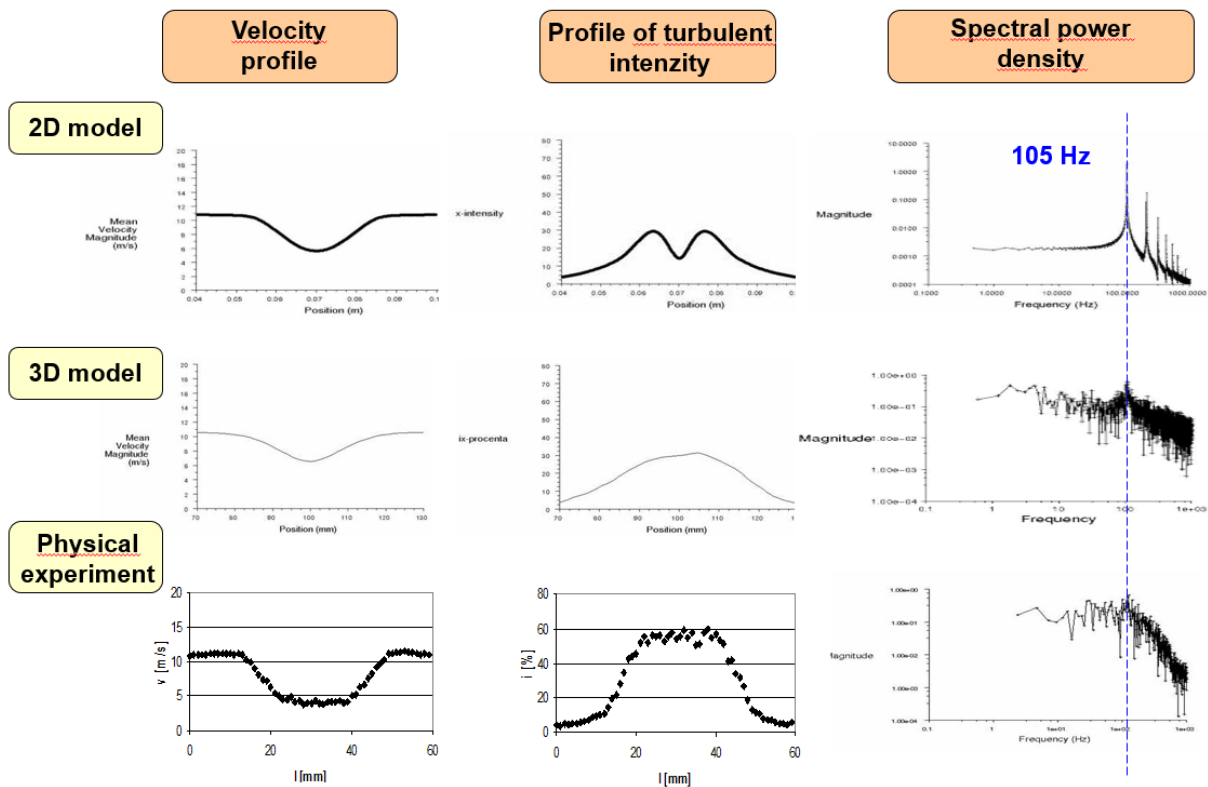


Fig. 6.12 Comparison of experiment and model [17]

6.7.3. Flow around two tubes

At flow around two or more tubes the flow and heat transfer depends on the tube diameter, on the flow velocity and on distance between the tubes. Modeling will evaluate these proposals and new optimal possibilities in the design will arise.

For this task the physical experiment was at first prepared. All preparations and settings were the same as for the task of flow around one tube in the previous chapter. Only one difference to the previous task was the fact that in the measuring section a second cylinder of the same diameter, i.e. $D = 20$ mm, has been inserted. Spacing between the cylinders was set to $2D$ i.e. 40 mm.

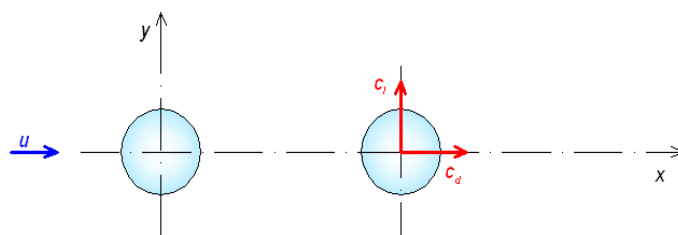


Fig. 6.13 Schematic view of the modeled the task [17]

At a distance of $x/D = 2,5$ the velocity profile and the turbulent intensity under one cylinder and two cylinders was again measured, see Fig. 6.14 and Fig. 6.15, both for the purpose of mutual comparison and for comparison with the mathematical model.

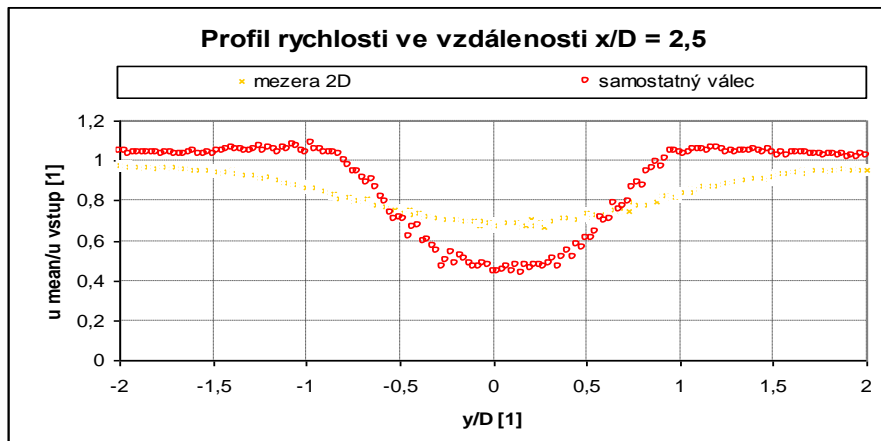


Fig. 6.14 Profile of mean velocity [17]

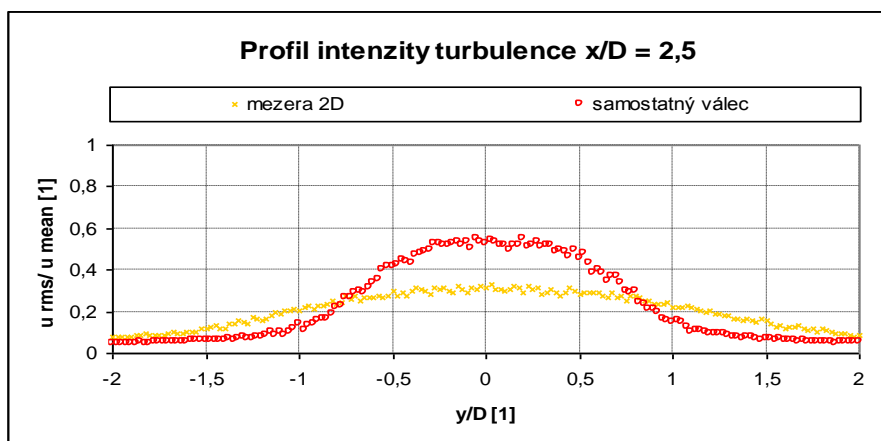


Fig. 6.15 Turbulence intensity profile [17]

For illustrative purposes Fig. 6.16 shows the velocity and the vortex shedding behind second tube in numerical testing.

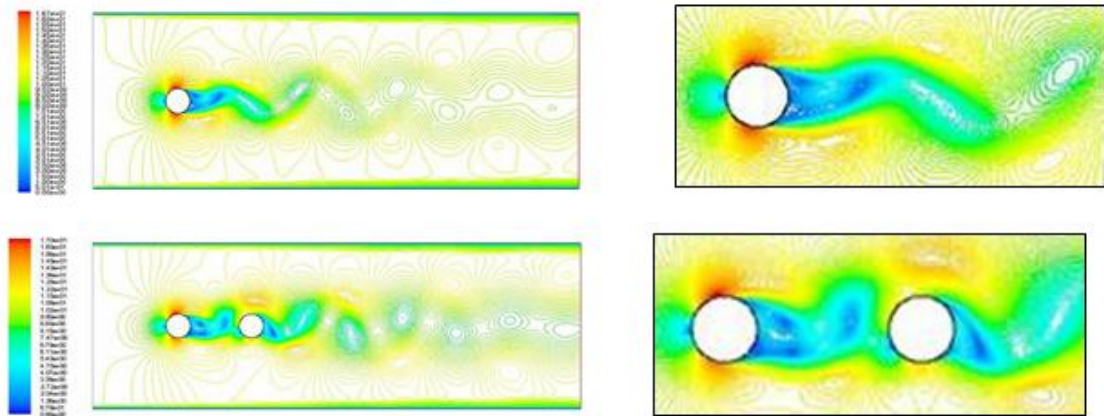


Fig. 6.16 Visual comparison of the size of the instantaneous velocity magnitude in the flow around one and two tubes [17]

The question remains whether the dominant frequency at which a transfer of much energy will be changed. The answer to this question is in Fig. 6.17, where the comparison of power spectra obtained from measurement at a distance of $X = [25; 10]$ behind one cylinder and second cylinder in the pair.

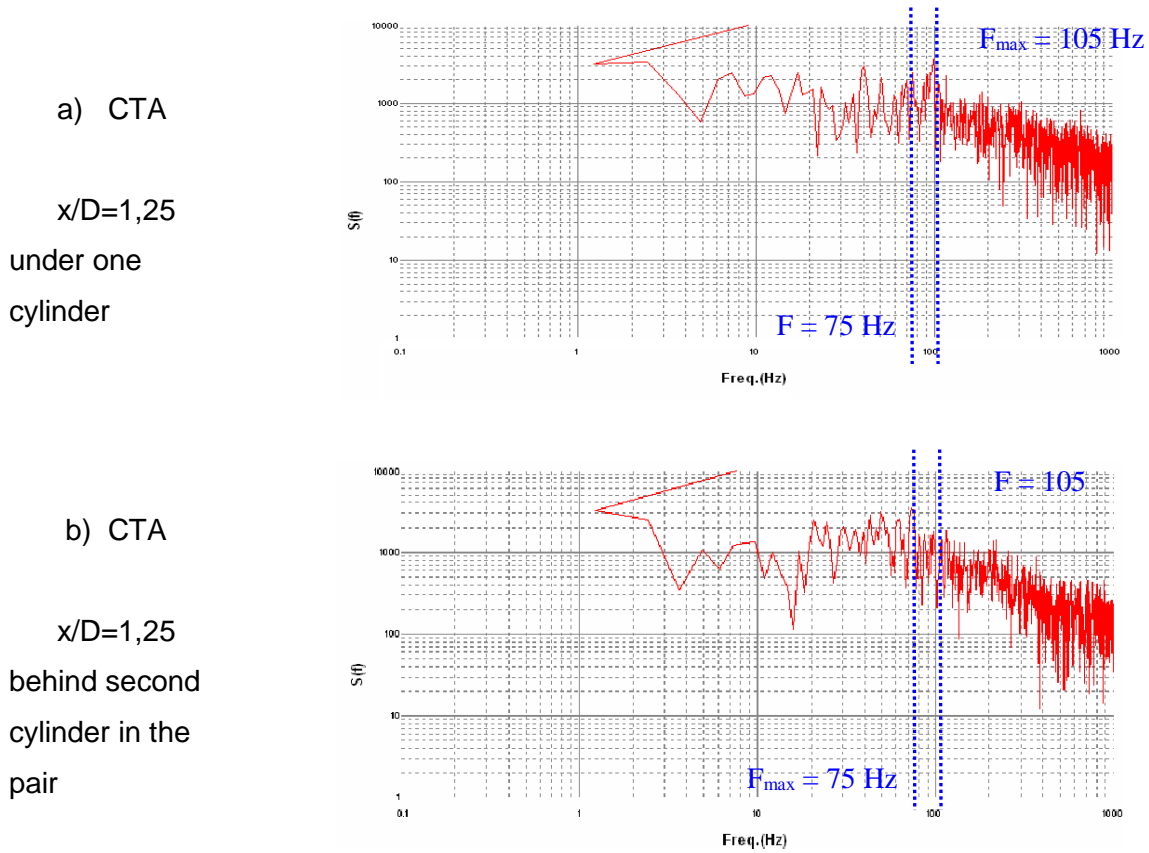


Fig. 6.17 Comparison of the power spectrum measurement by CTA [17]

The comparison shows a marked change in the first frequency at which the most energy transfer occurs. Its value is lower, 75 Hz. In terms of design impact, it is more dangerous than higher own frequencies.

The same information was obtained from numerical experiment.

7. Conduction and convection in turbulent flow

7.1. Energy equation at turbulent flow

The energy equation is analogous to the energy equation for laminar flow, but it is defined for time-averaged energy, temperature, while velocity and stress are also time-averaged:

$$\frac{\partial}{\partial t}(\rho \bar{E}) + \nabla \cdot (\vec{u}(\rho \bar{E})) = \nabla \cdot (\lambda_t \nabla \bar{T}) - \nabla \cdot (\vec{\tau}_t \cdot \vec{u}) + S_h \quad (7.1.1)$$

λ_t turbulent thermal conductivity is proportional to turbulent viscosity.

7.2. Mathematical model of turbulent flow with convection and conduction

When dealing with heat transfer by conduction and convection in turbulent flow, which is the vast majority of engineering applications, the following system of equations will be used:

- **continuity equation for time averaging state values**
- **momentum equations – in turbulent flow Reynolds equations for time averaging values of pressure and velocities**
- **equation for turbulent kinetic energy and turbulent dissipation**
- **energy equation for time averaging state variables**

The solution is complemented by boundary conditions. The accuracy of the solution does not depend on the shape of the area being solved, only on the quality of the grid created. Advantageously, simplicity is used in symmetrical and axially symmetrical regions. The following more or less simple examples of typical energy applications will demonstrate benefits of numerical finite volume method.

7.3. Boundary conditions

The boundary conditions for turbulent flow are supplemented by heat transfer conditions. The following table lists the most commonly used boundary conditions.

Tab. 7.1

	Type (Fluent)	Variable	Unit
INLET	VELOCITY INLET	velocity u	[ms ⁻¹]
		turbulent intensity I	[%]
		hydraulic diameter d_h	[m]
		temperature T	[K]
	MASS FLOW RATE	mass flow rate Q_m	[kgs ⁻¹]
		turbulent intensity I	[%]
		hydraulic diameter d_h	[m]
		temperature T	[K]
	PRESSURE INLET	total pressure	[Pa]
		$\rho_{tot} = \rho_{stat} + \rho_{dyn} = \rho_{stat} + \frac{1}{2}\rho u^2$	
		turbulent intensity I	[%]
		hydraulic diameter d_h	[m]
		temperature T	[K]
OUTLET	OUTFLOW	steady flow condition	
		$\frac{\partial u}{\partial n} = 0, \frac{\partial p}{\partial n} = 0, \frac{\partial T}{\partial n} = 0$	
		turbulent intensity I	[%]
		hydraulic diameter d_h	[m]
		temperature gradient $\frac{\partial T}{\partial n} = 0$	
	PRESSURE OUTLET	static pressure ρ_{stat}	[Pa]
		(0 at outlet into atmosphere)	
		turbulent intensity I	[%]
		hydraulic diameter d_h	[m]
		temperature T	[K]
	MASS FLOW RATE	mass flow rate Q_m	[kgs ⁻¹]
		turbulent intensity I	[%]
		hydraulic diameter d_h	[m]
		temperature T	[K]
WALL	WALL	$u = 0$ default (moving wall – define velocity)	

		<p>roughness for turbulent flow</p> <p>temperature T</p> <p>constant heat flux density q</p> <p>zero heat flux density (insulation) $q=0$,</p> <p>convective heat transfer α, T_{ref}</p> <p>radiation</p>	
	WALL	<p>shear stress or fluid sticks on the wall (no slip)</p> <p>roughness for turbulent flow</p> <p>temperature T</p> <p>constant heat flux density q</p> <p>zero heat flux density (insulation) $q=0$,</p> <p>convective heat transfer α, T_{ref}</p> <p>radiation</p>	

7.4. Heat transfer at turbulent flow around the plate

Following the experience of modeling of heat transfer at laminar flow around the plate the same geometry with the middle layer as SOLID material (steel), but with the left and right layer as FLUID will be used. There will be air and water flow in turbulent regime.

Solve temperature distribution due to conduction and convection in a layer of steel, where on left side the water flows and on right side the air flows. The physical model is given by the shape of the region whose diagram is shown in 2D in **Fig. 7.1** and the dimensions and physical properties in the tables Tab. 7.2, Tab. 7.3.

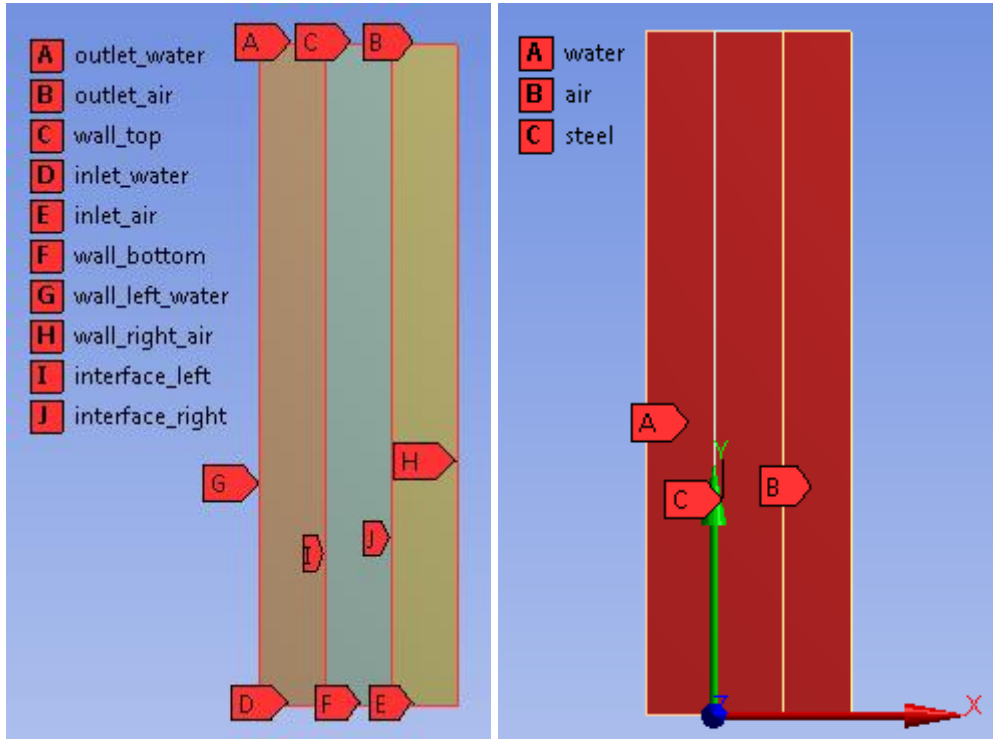


Fig. 7.1 Scheme of dimensional plate in coordinate system and boundary conditions

Tab. 7.2 Geometry of region

thickness of region $l_{steel} = l_{air}$ [m]	0.01
height of region h [m]	1

Tab. 7.3 Physical properties of material (steel, water, air) at 300 K

material	steel	air	water
density ρ [$\text{kg}\cdot\text{m}^{-3}$]	8030	1.225	998.2
specific heat capacity c_p [$\text{J}\cdot\text{kg}^{-1}\cdot\text{K}^{-1}$]	502.48	1006.43	4182
thermal conductivity λ [$\text{W}\cdot\text{m}^{-1}\cdot\text{K}^{-1}$]	16.27	1006.43	0.6
viscosity η [$\text{kg}\cdot\text{m}^{-1}\cdot\text{s}^{-1}$]		$1.7894\cdot 10^{-05}$	0.001003

Boundary conditions

Boundary conditions are defined on the left and right wall by temperature. At air and water inlet the velocity is defined and at the outlet the static pressure and turbulent parameters are specified. Due to the large dimensionality of the plate, the conditions of symmetry above and below of the plate are defined.

Tab. 7.1 Boundary conditions

name	type (Fluent)	T [K]	u [m·s ⁻¹]	p [Pa]	d _h [m]	l[%]
wall left water	wall	323				
wall right air	wall	263				
inlet water	velocity inlet	323	1		1	1
inlet air	velocity inlet	263	10		1	1
outlet left water	pressure outlet	323		0	1	1
outlet right air	pressure outlet	263		0	1	1
wall top	wall	293				
wall bottom	wall	293				
interface left	coupled					
interface left shadow	coupled					
interface right	coupled					
interface right shadow	coupled					

Estimation of dimensionless flow parameters

For the calculation, formulas for turbulent flow around the plate are used, see chap. 1.4. Calculations are made for the water flow around the plate from the left. With the air flow around the plate from the right, the estimates vary due to the change in the physical properties of the flow medium and temperature.

Results – water (left)

Reynolds number Re= 1000000
 Prandtl number Pr= 6.957454
 Nusselt number Nu= 4878
 Heat transfer coefficient α= 2927

Results – air (right)

Reynolds number Re= 629327
 Prandtl number Pr= 0.77429752
 Nusselt number Nu= 1620.05179
 Heat transfer coefficient α= 42.61

Mathematical model

In this task, there is a turbulent flow. Velocity, pressure and temperature distribution is controlled by above differential equations. Turbulent criterion is the Reynolds number:

$$Re_{air} = \frac{ud}{\nu} = \frac{10.1}{1.46073 \cdot 10^{-05}} = 629327$$

$$Re_{water} = \frac{ud}{\nu} = \frac{1.1}{10^{-06}} = 1000000$$

The value of Reynolds number is higher than the limit, flow is turbulent or rather the transition from laminarity to turbulence. Therefore, a turbulent model was chosen, that is more suitable for tasks with a low Reynolds number and for a heat transfer, ie *k-ω* model.

Creating geometry and mesh

In environment Workbench the exact geometry is creating and covering by mesh, see Fig. 3.6.

Results

The evaluation options are the same as in the previous examples, so only some important variables such as the temperature and profiles of the quantities characterizing the transfer of heat through the steel wall will be shown. In addition, turbulent quantities and turbulent physical properties will be evaluated.

Fig. 7.2 shows the temperature distribution in the air, steel and water layers. It can be seen that the air again acts as a good insulator.

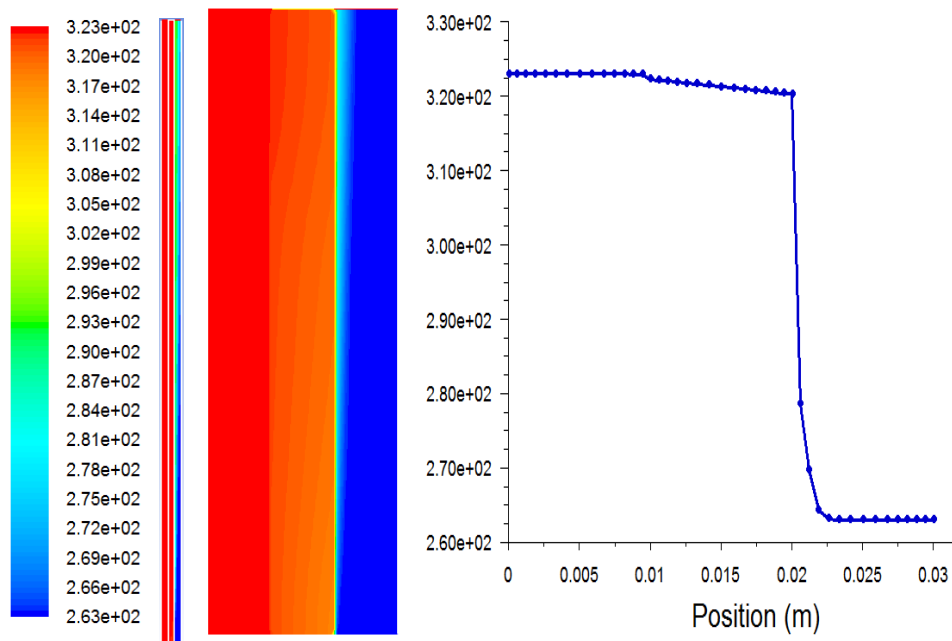


Fig. 7.2 Temperature distribution [K] in the air, steel and water layer and in detail and in the evaluation plane (right part)

Left region – water	Right region – air
---------------------	--------------------

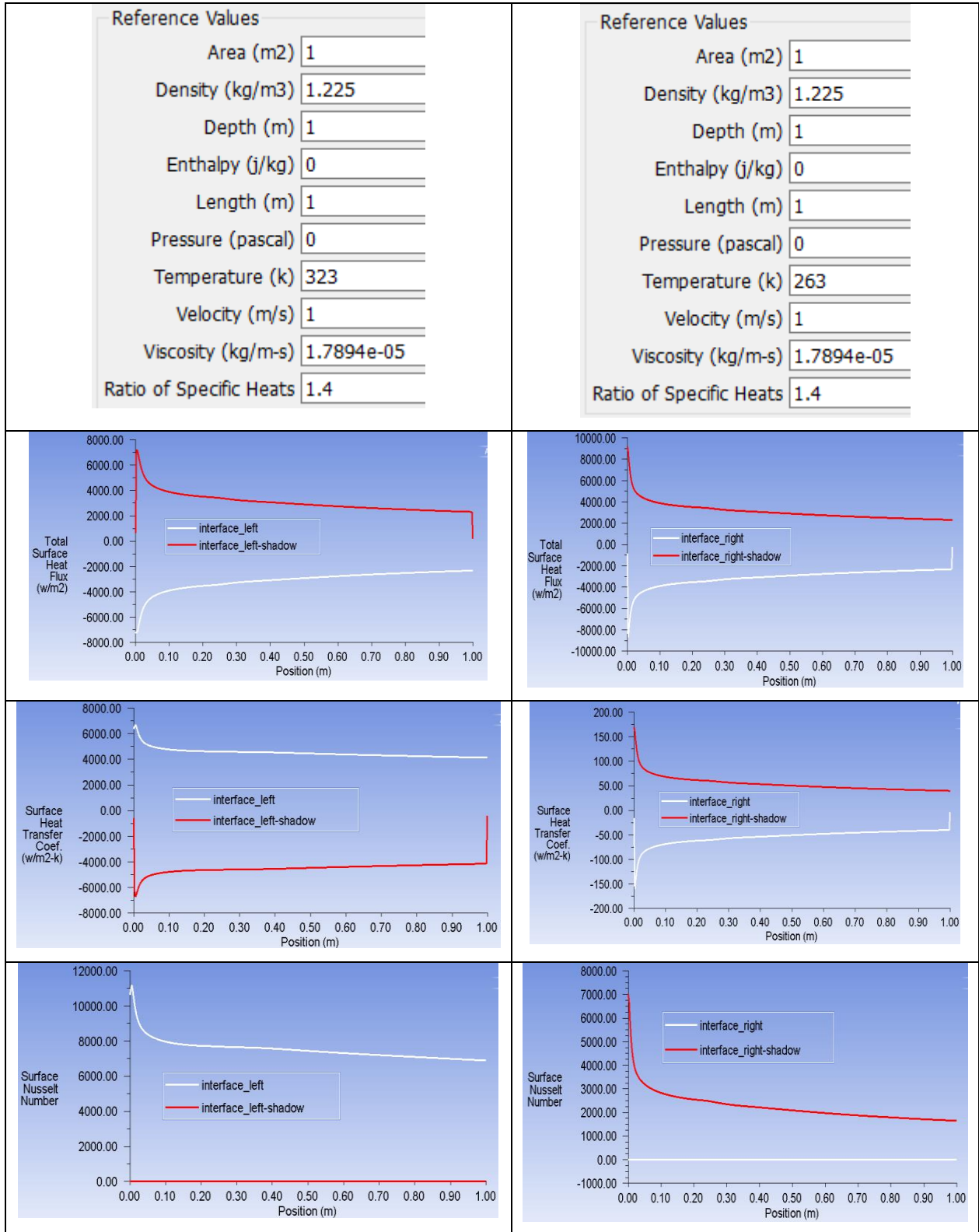


Fig. 7.3 Distribution of total surface heat flux, surface heat transfer coefficient and Nusselt number along the interfaces, $T_{ref}=323$ K and 263 K, $d_h=1$ m

Other parameters (Total Heat Transfer Rate P , Total Surface Heat Flux, Surface Heat

Transfer Coefficient α and Nusselt number Nu) are also determined by the area weighted average.

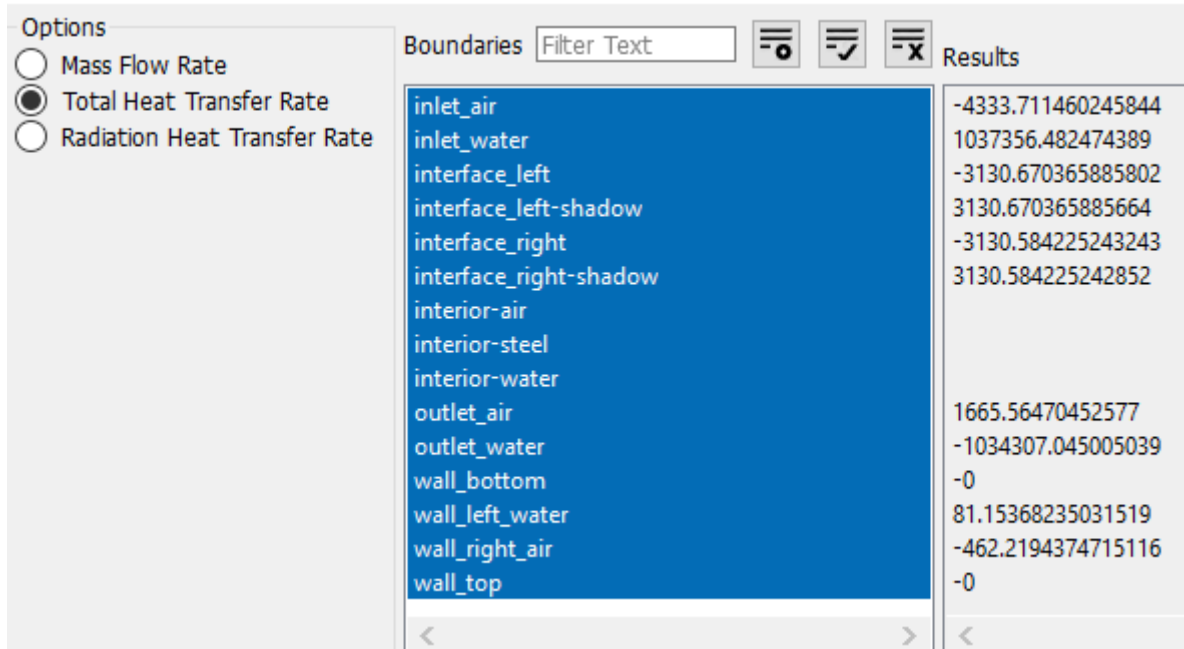


Fig. 7.4 Heat power

For simplified practical applications, it will make sense to evaluate the average values of these values and compare them with estimations, Tab. 7.4.

Tab. 7.4

Average values	water	air	water	air
	left	right	left	right
	estim.	estim.	Fluent	Fluent
Inlet velocity [ms⁻¹]	1	10	1	10
Total Heat Transfer Rate [W]			3130	3130
Surface Heat Transfer Coef. [W·m⁻²·K⁻¹]	2927	42	4507	53
Nusselt number [1]	4878	1620	7512	2228

Different values are due to the constant boundary conditions of the flow and thus due to the acceleration of the flow at the inlet to the area.

7.5. Flow around the tube with heat transfer (without flow inside)

The above defined problem of air flow around the tube will now be complemented by solving of heat transfer. Thus the geometry is identical to a previous task. It is also defined

the air temperature, the wall temperature. The flow inside the tube is not for reasons of simplicity expected, but it is solvable, as will be seen in subsequent chapters.

This element is the cornerstone of a wide range of tubular heat exchangers, therefore partly empirical theories have been developed and which allow to define the relevant flow parameters such as:

Reynolds number is defined as:

$$Re = \frac{u_{ref} d}{\nu} \tag{7.5.1}$$

Prandtl number is the ratio of the viscous and thermal diffusion and is dependent on the material properties of the fluid.

$$Pr = \frac{\rho c_p \nu}{\lambda} \tag{7.5.2}$$

Nusselt number is given by formula :

laminar, transient and turbulent $Nu = C_1 Re^{C_2} Pr^{0,38}$

flow around the tube in cross direction	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

Solve turbulent flow around the heating tube, evaluate the influence of grid and turbulent model on results that compare with empirical estimates. Area is defined by 2D scheme in **Fig. 7.5**

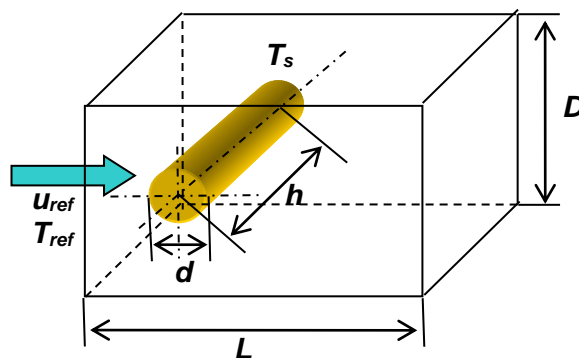


Fig. 7.5 Definition of the region when air flows around tube with heat transfer

Geometrical and physical parameters of the tube and of the neighborhood with the flowing air are given in tables.

Tab. 7.5 Geometrical and physical parameters of the tube

diameter	$d_{ref} =$	0.0127	m	length	$h =$	0.094	m
area of the tube wall	$S =$	0.00375	m ²	wall temperature	$T_s =$	128.4	°C

Tab. 7.6 Geometrical and physical parameters of the neighborhood with a air stream neighborhood

height	$D =$	0.06	m	length	$L =$	0.1	m
Air properties							
density	$\rho =$	1.23	kg·m ⁻³	temperature	$T_{ref} =$	26.2	°C
velocity	$v_{ref} =$	10	m·s ⁻¹	viscosity dyn.	$\eta =$	1.78E-05	Pa·s
viscosity	$\nu =$	1.59E-05	m ² ·s ⁻¹	thermal conductivity	$\lambda =$	0.0242	W·m ⁻¹ ·K ⁻¹
kinematic							
specific heat	$c_p =$	1066.6	J·kg ⁻¹ ·K ⁻¹	thermal conductivity	$a =$	2.25E-05	m ² ·s ⁻¹

From given parameters you can calculate the above parameters of flow and heat transfer.

Estimation of heat transfer coefficient is then determined from Nusselt number $Nu = \frac{\alpha d}{\lambda}$ by

the relationship $\alpha = \frac{Nu \lambda}{d}$.

Tab. 7.7

Reynolds number	$Re =$	7992
Prandtl number	$Pr =$	0.68
Nusselt number	$Nu =$	38.86
Surface heat transfer coefficient	$\alpha =$	80.47 W·m ⁻² ·K ⁻¹

For the numerical solution 2D model is selected and the results will be compared with the calculations for typical tasks such as the dimensionless parameter Reynolds, Prandtl and Nusselt number and heat transfer coefficient.

Mathematical model

In this task we suppose turbulent flow and a mathematical model $k-\omega$ is used. The criterion of turbulence is Reynolds number, which is determined by the velocity of the air, its viscosity and cylinder diameter:

Reynolds number $Re = 7992$ turbulent flow

Reynolds number is higher, but it is a transition from laminarity to turbulence.

Creating a geometry and mesh

In an environment Workbench the exact geometry is created, but with the regard to the further use of modeling heat transfer with the flow in the pipe there is made simpler grid, see Fig. 7.6.

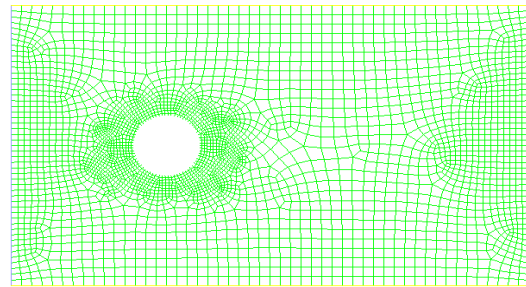


Fig. 7.6 Grid for 2D geometry of the flow around the tube

Results

Evaluated options are the same as in the previous examples, therefore, only some significant variables are shown. To calculate the Nusselt number and the heat transfer coefficient it is necessary to update the reference values, which are basic for calculating the heat transfer coefficient and Nusselt number. Estimation of surface heat transfer coefficient is determined again by equation:

$$\alpha = \frac{q}{T_s - T_{ref}} \quad (7.5.3)$$

and subsequently Nusselt number

$$Nu = \frac{\alpha d_{ref}}{\lambda} \quad (7.5.4)$$

The reference values are defined as follows:

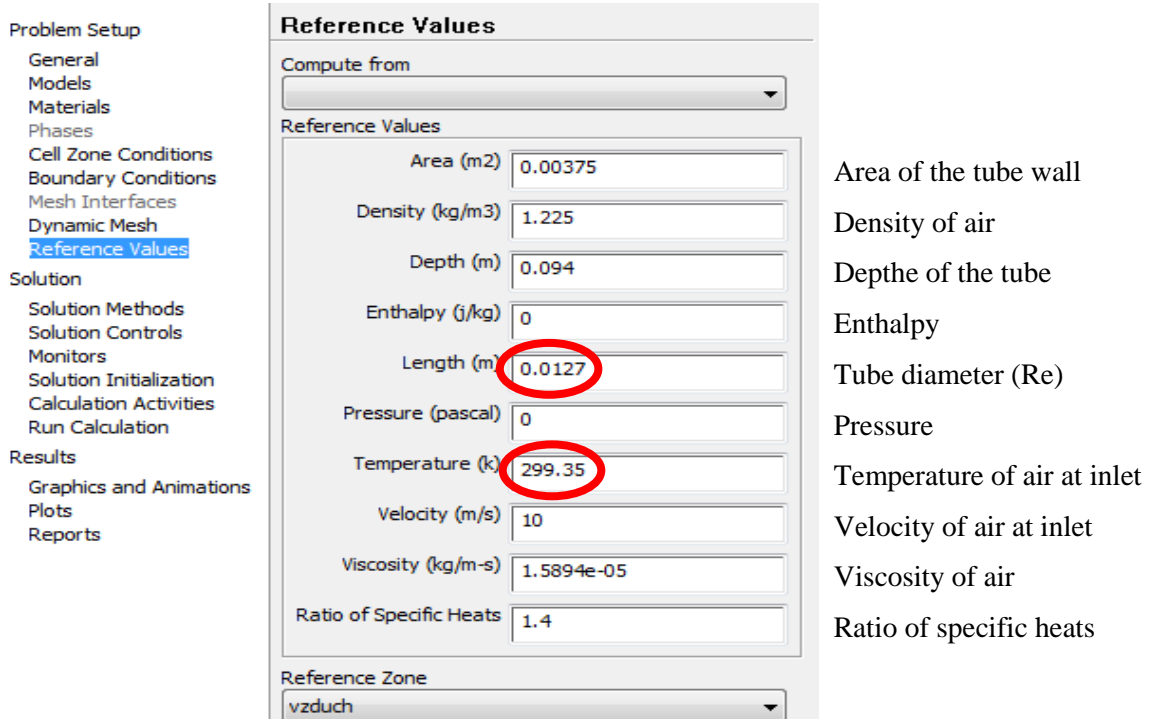


Fig. 7.7 Definition of reference values for calculating the parameters of heat transfer

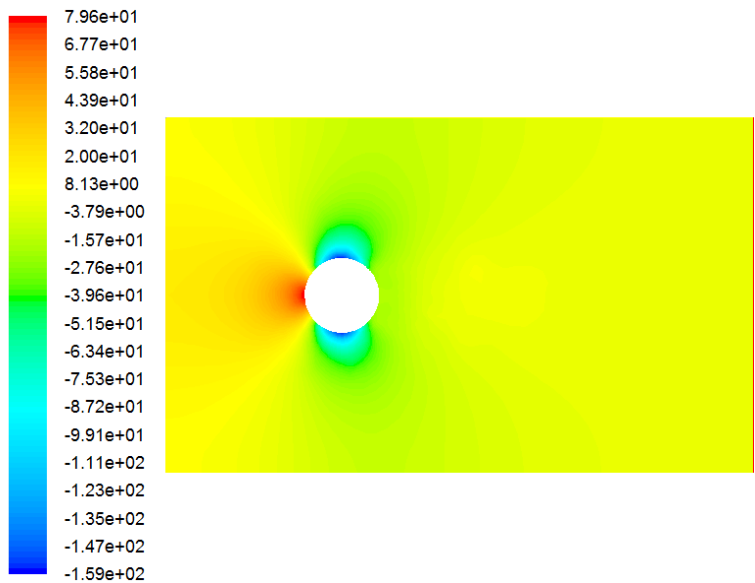


Fig. 7.8 Distribution of static pressure [Pa]

With these parameters it is possible to evaluate the Nusselt number and the heat transfer coefficient on the tube wall, or determine their mean value.

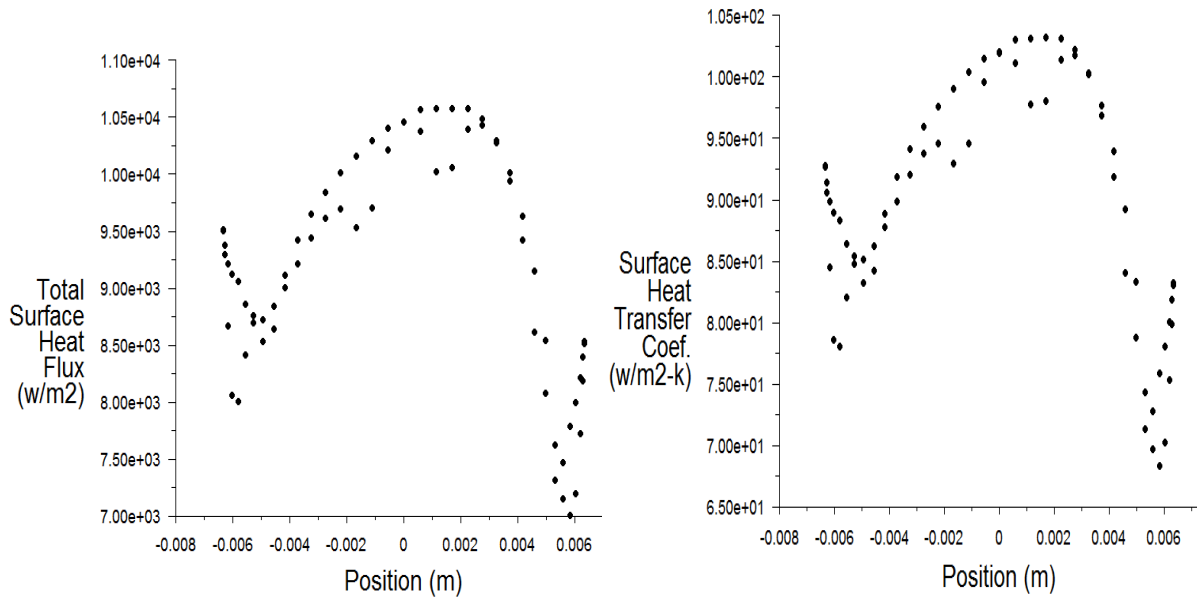


Fig. 7.9 The total surface heat flux and surface heat transfer coefficient along the perimeter of the tube wall, an average value $\alpha=99.33$, $T_{ref}=299.35$ K, $d_h=0.0127$ m

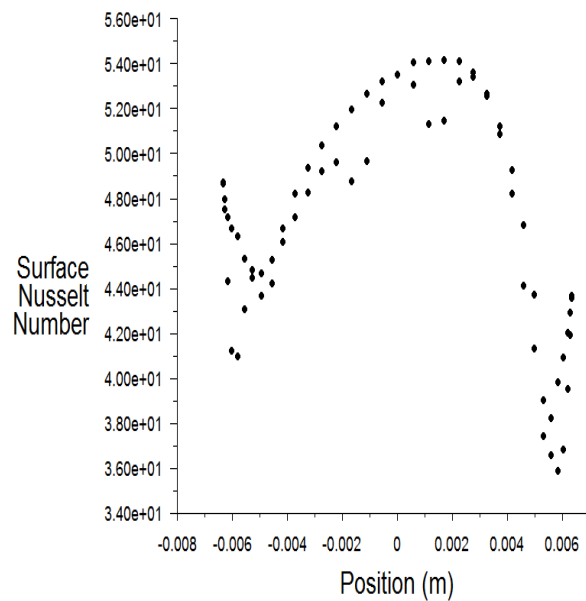


Fig. 7.10 Nusselt number along the perimeter of the tube wall, whose average value is $Nu=44.37$, $T_{ref}=299.35$ K, $d_h=0.0127$ m

The whole numerical calculation was solved earlier by RNG $k-\epsilon$ model and repeated on a much finer mesh around tube. Mesh can also be done in Fluent using command ADAPT. It was then recognized better boundary layer of velocity and temperature profile and it caused substantial heat transfer between the wall and the fluid. Therefore, the different values were found due to coarsen mesh, where the heat from the wall almost did not spread to the

surrounding area. In Fig. 7.11 a Fig. 7.12 you can compare differences in temperature distribution in the area behind the tube. Third variant was solved on coarsen mesh using $k-\omega$ turbulent model.

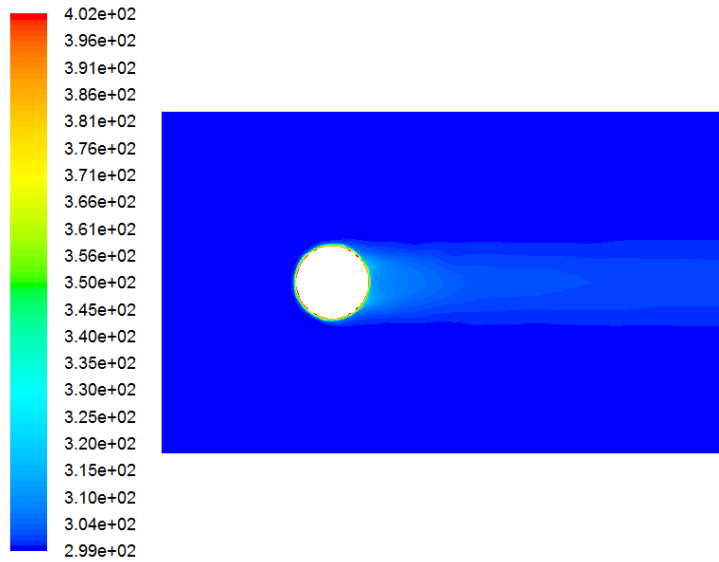


Fig. 7.11 Static temperature distribution on the coarsen mesh

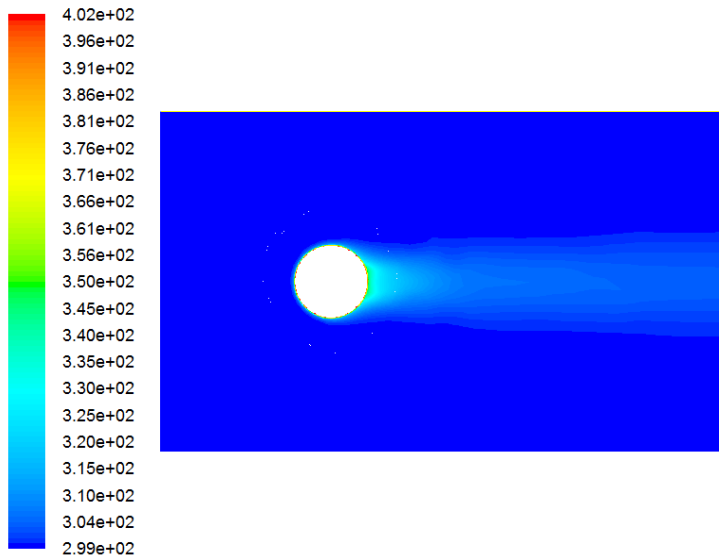


Fig. 7.12 Static temperature distribution on the fine mesh

Another important parameter is the pressure loss in the flow direction using PLOT X-Y evaluation.

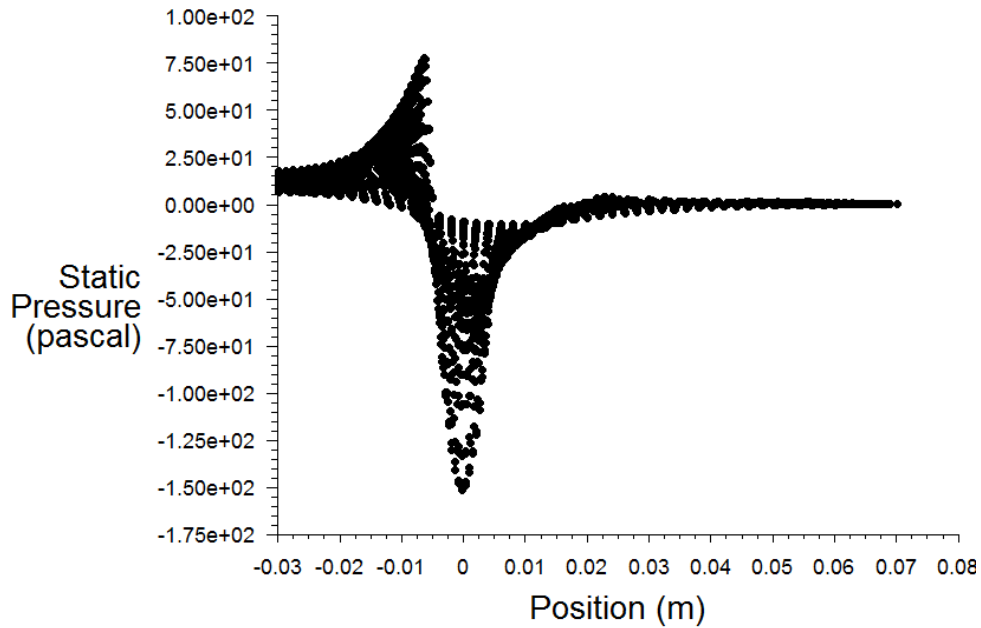


Fig. 7.13 Static pressure distribution in flow direction

The pressure drop from the numerical calculation is determined as difference of the averaged static pressure on the inlet area and the outlet area. From **Fig. 7.13** it is seen that in the area around the tube there are significant changes of pressure, but the pressure loss is given by the above definition. It is obvious that in case of close placement of tubes in a row it will be necessary the pressure changes to model.

$$\Delta p = p_{inlet} - p_{outlet} = 11.7 - 0 = 11.7 \text{ Pa}$$

Comparison between the estimation and the numerical solution obtained by calculating on coarsen and fine mesh is evaluated in the following table.

Tab. 7.8

	estimation	coarsen mesh <i>k-ε</i>	Fine mesh <i>k-ε</i>	Coarsen mesh <i>k-ω</i>
Surface heat transfer coefficient [Wm ⁻² K ⁻¹]	80.47	89.48	142.26	121.1
Nusselt number [1]	38.86	46.96	74.66	63.30
Thermal power [W]	30.84	34.4	54.7	46.11
Pressure loss [Pa]		11.4	7.7	12.8

7.6. Flow around the tube with the heat transfer (with the flow inside)

While simplifying assumption by the previous task the fluid is not flowing through the tube and it is assumed only that the tube wall is heated to a constant temperature, it makes no sense to deal with spatial modeling, as in each section perpendicular to the tube axis the distribution of flowing and temperature field is the same. When fluid passes through a long tube, then there is observable a temperature change along the pipe. In this case, flow and temperature field especially in sections perpendicular to the axis of the tube varies. Therefore, the task was solved with a flowing fluid inside the tube as a 3D spatial problem. It can be seen that for pipe lengths on the order of 1 meter the temperature remained almost is unchanged, the result will not be displayed. In the longer tubes arranged for example in a spiral, the 3D calculation has sense.

7.7. Flow across the tube bundle with heat transfer

The heat transfer at cross flow through the tube bundle has a number of industrial applications, such as steam generation in the boiler or cooling air-conditioning units. The geometrical configuration is in Fig. 7.14.

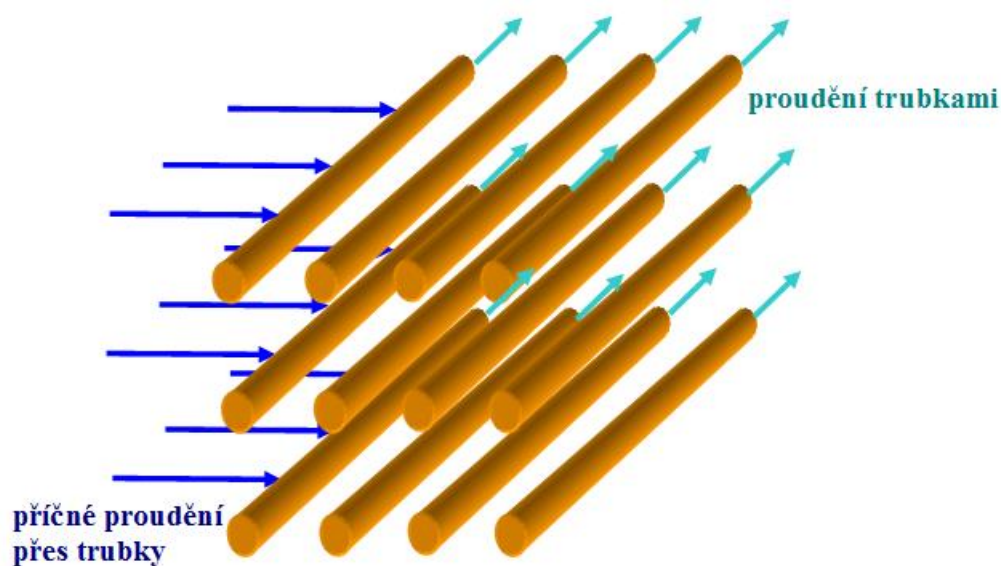


Fig. 7.14 Arrangement of the tubes at cross flow.

The arrangement may be of two kinds, row arrangement and cross arrangement [2] , [3] . When looking in 2D the arrangement is structured as follows:

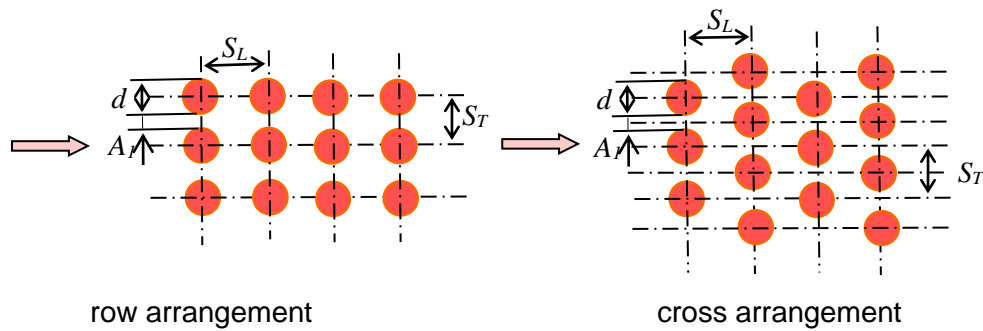


Fig. 7.15 Schematic representation of the arrangement of the tubes in cross flow.

Heat transfer coefficient in this case is associated with the location in the tubular system. Coefficient for the first tube is approximately equal to the coefficient defined for one tube in the cross flow, while coefficients of the tubes inside the tubular system vary and depend on the type of arrangement. In most configurations, the heat transfer conditions have stabilized and small changes appear in the heat transfer coefficient for the tube in the fourth to fifth row. When a larger number of rows (N_L is greater than 10), it is possible to define an average coefficient:

laminar, transient and turbulent flow across the tube bundle, N_L is a number of tube column

$$Nu_D = C_1 Re_{D,max}^m \quad \text{pro } N_L > 10, \quad 2000 \leq Re_{D,max} \leq 40000$$

$Pr = 0.7$, constants C_1 and m are given in table
 S_L – horizontal distance of tubes, S_T – vertical distance of tubes

Constants for determining the Nusselt number when flowing across the tube bundle

tubes in row	$S_T/D= 1.25$		$S_T/D= 1.50$		$S_T/D= 2.00$		$S_T/D= 3.00$	
S_L/D	C_1	m	C_1	m	C_1	m	C_1	m
1.25	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

tubes in 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

(7.7.1)

During the flow through the tubular system there is a significant change in temperature. In this case the wall temperature is reducing and thus the temperature difference is reducing too. The thermal power would be highly overestimated by using the temperature difference $\Delta T = T_s - T_{ref}$. Therefore so called logarithmic temperature difference is used:

$$\Delta T_{lm} = \frac{(T_s - T_l) - (T_s - T_o)}{\ln\left(\frac{(T_s - T_l)}{(T_s - T_o)}\right)} \quad (7.7.2)$$

where T_l, T_o is inlet and outlet temperature of the flowing medium. The outlet temperature which is required for determination ΔT_{lm} may be estimated from follows:

$$\frac{T_s - T_o}{T_s - T_l} = \exp\left(-\frac{\pi d N \bar{\alpha}}{\rho u N_T S_T c_p}\right) \Rightarrow T_o = -\left(\exp\left(-\frac{\pi d N \bar{\alpha}}{\rho u N_T S_T c_p}\right)\right) \cdot (T_s - T_l) - T_s$$

where N is the total number of tubes in the system and N_T is the number of tubes in a vertical direction. Thus ΔT_{lm} is known value and thermal power per unit length of the tube can be calculated from the relationship

$$P = N(\pi d \bar{\alpha} \Delta T_{lm}) \quad (7.7.3)$$

An important parameter is the pressure loss, which is defined by the Bernoulli equation and depends on the loss coefficient corresponding to system of tubes and is determined empirically.

$$\Delta p = N_L \zeta \left(\frac{\rho u_{\max}^2}{2}\right) \text{ resp. } \Delta p = N_L \zeta \left(\frac{8 Q_m^2}{\rho \pi^2 d^4}\right) \quad (7.7.4)$$

Loss coefficient is specific for a different arrangement of tubes. When arranging the tubes in a row the loss coefficient is defined as follows:

$$\zeta = \gamma \left(N_L \frac{S_L}{S_T} A + B \right) \quad (7.7.5)$$

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

At cross tubes arrangement it is similarly defined:

$$\zeta = \gamma \left(0.7 + 0.8 \left(N_L \frac{S_L}{S_T} A + B \right) \right) \tag{7.7.6}$$

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

Coefficient γ depends on the Reynolds number. For values higher than 40 000 it is equal to one and for lower values it is estimated from empirical measurements shown in Fig. 7.16.

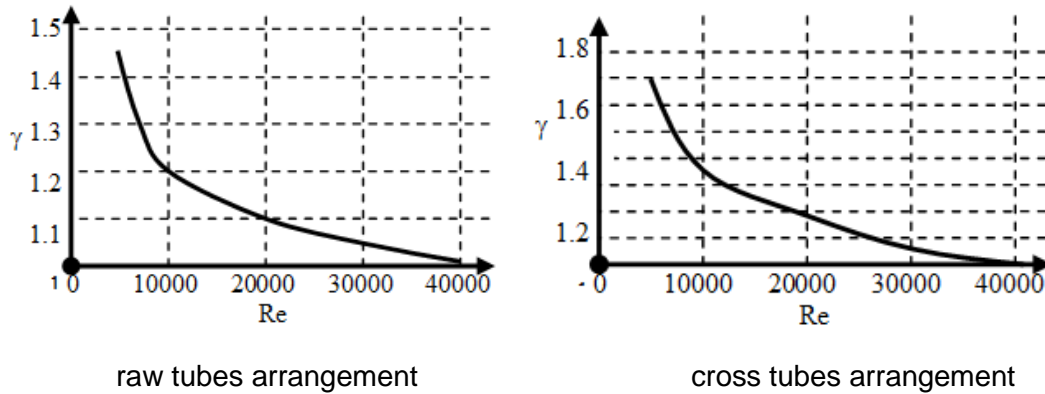


Fig. 7.16 Values of the coefficient γ depending on the number Re [3]

As you can see, the solution of flow across a such tubular system is dependent on number of empirically determined coefficients, the specification is not the goal of this course. In Fluent the pressure gradient is acquired directly. It is also possible to re-determine the loss coefficient which can be the result of calculation. The next chapter outlines the possibility of solving of flow across the tubular system with heat transfer for simplicity in 2D by numerical way.

7.7.1. The arrangement of the tube bundle in a row

According to the above scheme a tube bundle was solved. The number of tubes is $N = N_L N_T = 7 * 8 = 56$ and geometry and mesh was created. Geometrical parameters of the region are as follows:

Tab. 7.9

Tube			
diameter	$d =$	0.0164 m	length $l =$ 1 m
surface of the tube wall	$S =$	0.0515	m^2

horizontal spacing	$S_L=$	0.0343	m	vertical spacing	$S_T=$	0.0313	m
	$S_L/d=$	2.0915			$S_T/d=$	1.9085	
	$S_T/S_L=$	0.9125					
number of tubes horizontally	$N_L=$	7		number of tubes vertically	$N_T=$	8	
number of tubes in system	$N=$	56					
wall temperature	$T_s=$	70	°C	wall temperature	$T_s=$	343.15	K

The tubular system was blown with air in region defined as surroundings while the desired physical properties of air and the flux were given. Data are defined identically for comparison with example published in lit.[2] .

Tab. 7.10
Surroundings

width	$D=$	0.2555	m	length	$L=$	0.2422	m
-------	------	--------	---	--------	------	--------	---

Tab. 7.11
Properties of air

density	$\rho=$	1.2295	kg·m ⁻³	temperature	$T_{ref}=$	15	°C
viscosity	$\nu=$	1.48E-05	m ² ·s ⁻¹	viscosity dyn.	$\eta=$	1.82E-05	Pa·s
thermal conductivity	$\lambda=$	0.0253	W·m ⁻¹ ·K ⁻¹	temperature conductivity	$a=$	2.04E-05	m ² ·s ⁻¹
specific heat	$c_p=$	1007	J·kg ⁻¹ ·K ⁻¹				
velocity	$u=$	6	m·s ⁻¹	velocity max.	$u_{max}=$	7.7	m·s ⁻¹
mass flow	$Q_m=$	0.176	kg·s ⁻¹				

The parameters are estimated according to [2] that will be necessary in a calculation, such as the maximum velocity. Mass flow is defined in 2D task for the depth of region (tube length) equal to 1 m. Physical properties are defined as temperature independent, but they can be used when calculating the relationship of these variables depending on the temperature which offers Fluent (polynomial dependence or using kinetic theory) .

The above parameters of flow and heat transfer (Reynolds number is calculated from velocity maximum) can be computed using given parameters. Estimate of Nusselt number is problematic and it is really approximated only. From this estimation the calculation of heat

transfer coefficient on the wall follows $\alpha = \frac{Nu\lambda}{d}$.

Tab. 7.12

Reynolds number max.	Re=	8521	turbulent flow
Prandtl number	Pr=	0.7253	
correction factor	C_1 =	0.229	
exponent	m =	0.632	
Nusselt number	Nu=	69.81	
heat transfer coefficient	α =	107.69	$\text{W}\cdot\text{m}^{-2}\cdot\text{K}^{-1}$

Next, a calculation of thermal power will have been done, where the estimate of the average logarithmic temperature is used. The results are in the following table.

Tab. 7.13
Calculation of thermal power

temperature difference at the input	$T_s - T_f$ =	55.00	K
estimate of the temperature difference at the output	$T_s - T_0$ =	48.28	K
logarithmic mean temperature	ΔT_{lm} =	51.57	K
thermal power	P =	16023.83	W

Calculation of pressure drop again uses empirical relationships, see Tab. 7.14.

Tab. 7.14
Calculation of pressure drop

coefficient	A =	0.1236
coefficient	B =	1.2115
loss coefficient	ζ =	2.1593
correction coefficient	g =	1.4000
pressure drop	Δp =	334.51 Pa

These calculations will again be compared with the numerical model. It is expected that the task is more complicated in terms of geometry and so the results of numerical solutions are different.

Mathematical model

In this task, there is a turbulent flow according to previous calculations of Reynolds number. Mathematical model RNG k - ε is again used. The Reynolds number is high, therefore it is the turbulent regime, but model k - ω will also be tested.

Creating geometry and mesh

According to the above dimensions the mesh has been formed, see Fig. 7.17.

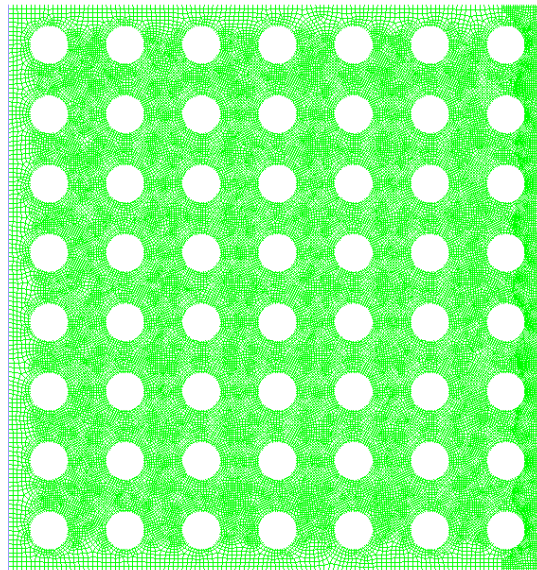


Fig. 7.17 Geometry and mesh

Results

Fig. 7.18 shows a decrease of the static pressure in the flow direction and, moreover, irregular distribution close to tubes due to velocity changes, swirl eventual flow separation behind tubes, which is evident from a further detailed view of the stream function in Fig. 7.19.

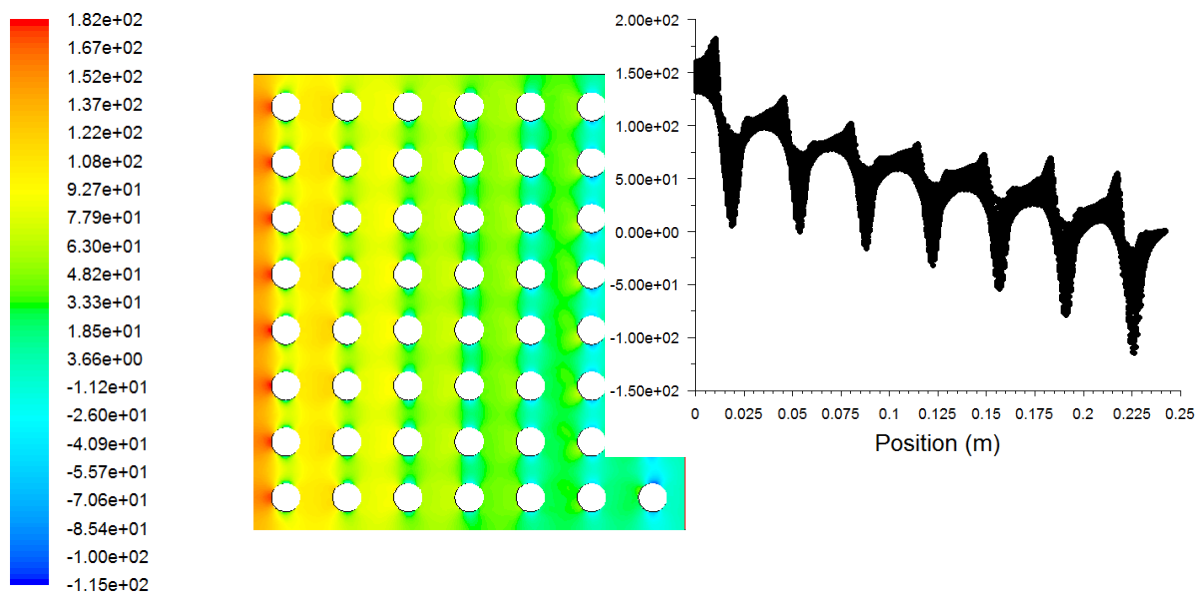


Fig. 7.18 Distribution of static pressure inside the area and a detail of pressure distribution inside the area vs. length area in the graph

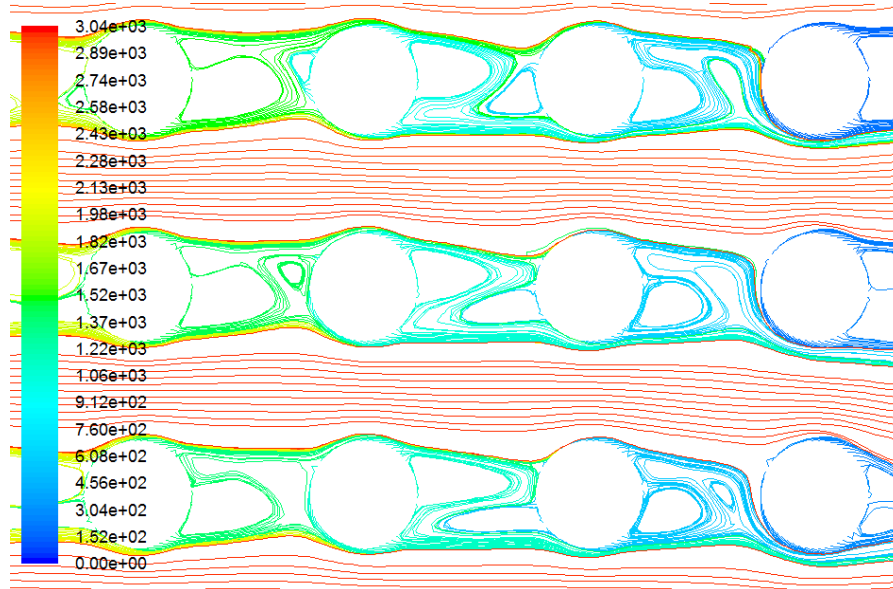


Fig. 7.19 Detail of flow separation behind tubes

It is interesting the temperature distribution. Due to a predefined constant temperature on the tube walls the flow temperature is certainly overestimated, since the tube due to the cool air flow through the tubular system must be cooling. Also, the temperature in the tubes is assumed constant.

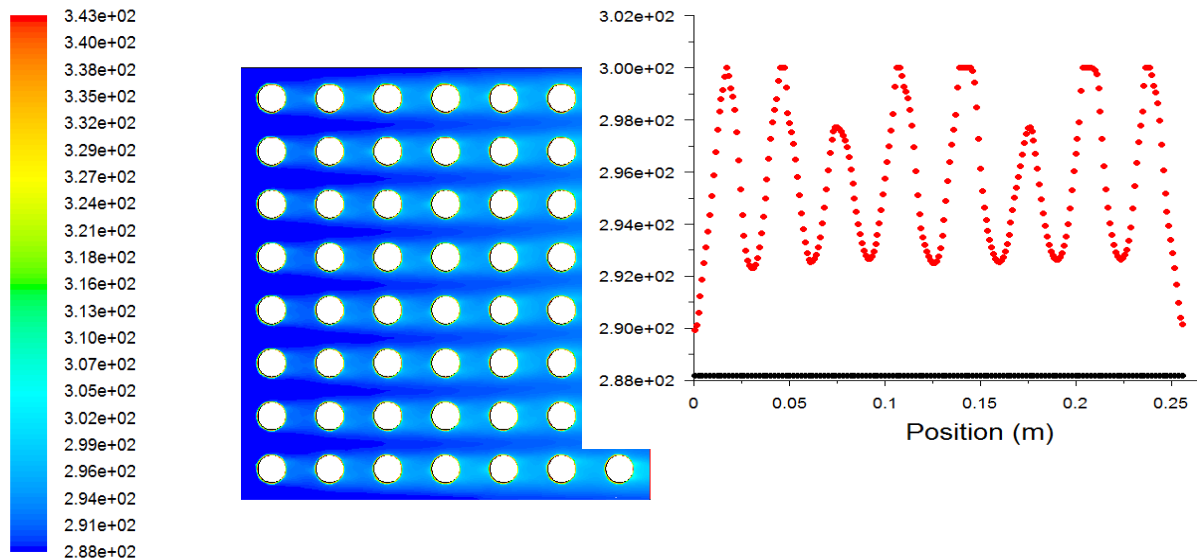


Fig. 7.20 Static temperature in the whole region and at the input (constant) and the output (periodicity is given by bypassing a number of tubes)

Very informative graph dependency is gained using PLOT X-Y. On **Fig. 7.21** and **Fig. 7.22** the curves estimating the surface heat transfer coefficient and Nusselt number are evaluated. It is interesting to observe the periodicity of the values on which it is necessary to determine

the area averaged values to compare it later with empirical estimates. See that the progression of these functions do not change much behind the fourth-fifth series of tubes.

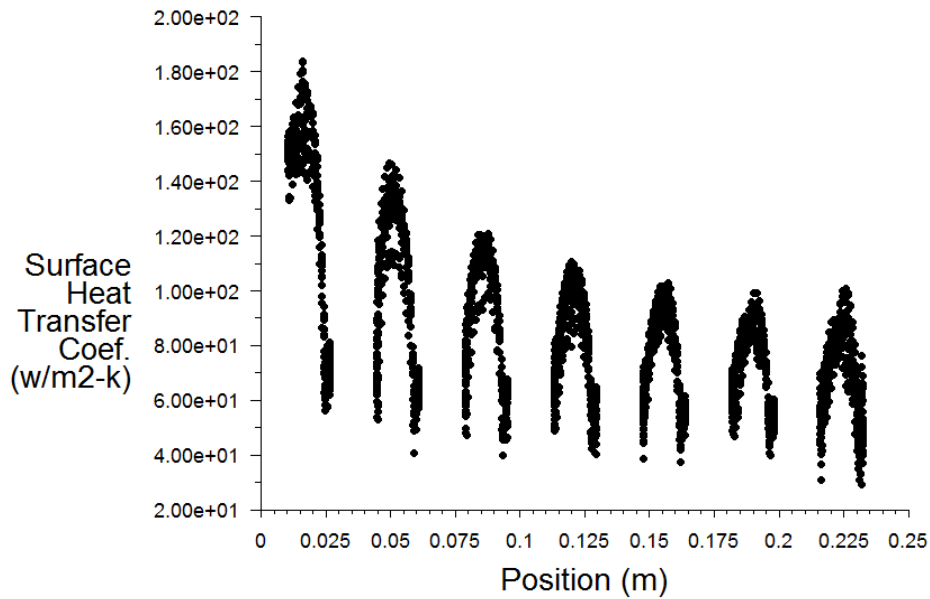


Fig. 7.21 Surface heat transfer coefficient on the walls of the pipe

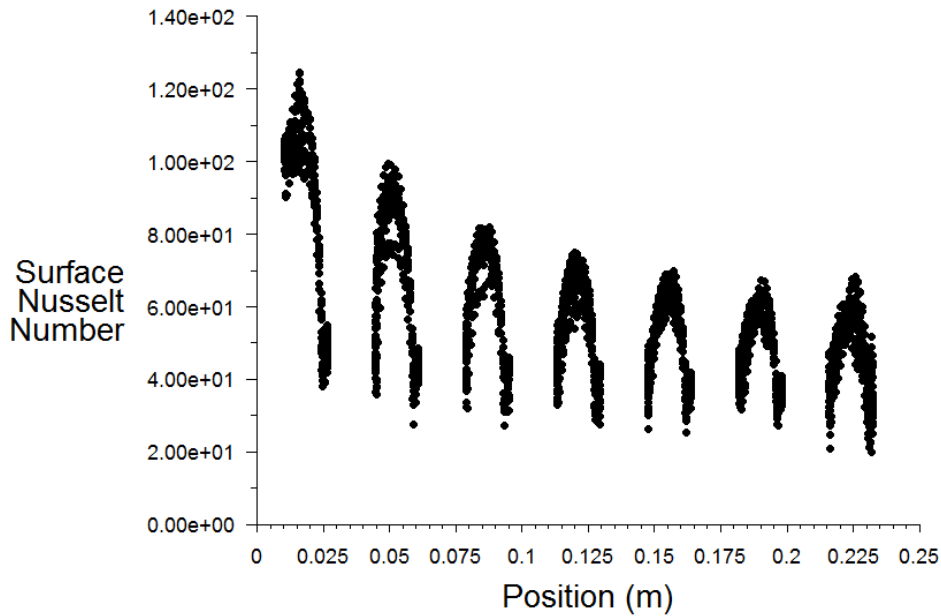


Fig. 7.22 Nusselt number evaluated on the walls of the pipe

The results of theoretical - empirical estimation of significant quantities during the air flow through the tubular system with heat transfer are compared in the following table with the area averaged values obtained from numerical solution. The differences are significant and are caused by bypassing the first three to five tubes, and that the total number of rows is in

real exchangers considerably greater. Average logarithmic temperature in the numerical calculation does not occur because it is not needed to determine other variables.

Tab. 7.15

	estimation	<i>k-ω</i> model adapt
Surface heat transfer coefficient [W.m ⁻² .K ⁻¹]	107.69	85.42
Nusselt number [1]	69.81	57.89
Mean logarithmic temperature	51.57	
thermal power [W]	16023	13591
pressure drop [Pa]	334.51	179.67

7.7.2. The arrangement of the tube bundle to cross

The second commonly used variant of the tube arrangement in heat exchanger is a variant of the arrangement of the tube bundle to cross.

Geometrical parameters of region are very similar, tube dimensions and spacing agree, tubes in each second row are shifted in the vertical direction. Also, velocities, inlet temperature and tube temperature are identical to the previous example. Therefore, only the different parameters are given.

The tubular system was blown with air in an area defined as surroundings, where due to the shift tubes the overall dimensions of region were changed

Tab. 7.16

surroundings

width	$D=$	0.27125	m	length	$L=$	0.2422	m
-------	------	---------	---	--------	------	--------	---

Tab. 7.17

properties of air

density	$\rho=$	1.225	kg·m ⁻³	temperature	$T_{ref}=$	15	°C
viscosity	$\nu=$	1.48E-05	m ² ·s ⁻¹	viscosity dyn.	$\eta=$	1.82E-05	Pa·s
thermal conductivity	$\lambda=$	0.0253	W·m ⁻¹ ·K ⁻¹	temperature conductivity	$a=$	2.04E-05	m ² ·s ⁻¹
specific heat	$c_p=$	1007	J·kg ⁻¹ ·K ⁻¹				
velocity	$u=$	6	m·s ⁻¹	velocity max.	$u_{max}=$	7.38	m·s ⁻¹
mass flow	$Q_m=$	0.176	kg·s ⁻¹				

The above parameters of the flow with heat transfer (Reynolds number is calculated from the velocity maximum) can be calculated by specified parameters:

Tab. 7.18

Reynolds number max.	Re=	8167	turbulent flow
Prandtl number	Pr=	0.7253	
correction factor	C_f =	0.48	
exponent	m =	0.56	
Nusselt number	Nu=	72.13	
heat transfer coefficient	α =	111.28	$W \cdot m^{-2} \cdot K^{-1}$

Estimation of Nusselt number and other flow parameters is performed as in the previous case and is approximated.

Tab. 7.19

Calculation of thermal power

temperature difference at the input	$T_s - T_f$	55	K
estimate of the temperature difference at the output	$T_s - T_o$	48.32	K
logarithmic mean temperature	ΔT_m	51.59	
thermal power	P	15289.96	W

Calculation of pressure drop uses empirical relationships too.

Tab. 7.20

Calculation of pressure drop

coefficient	A	0.1236	
coefficient	B	1.2115	
loss coefficient	ζ	2.4274	
pressure drop	Δp	376.0533	Pa

These calculations will be compared with the numerical model.

Mathematical model

In this task, there is a turbulent flow, but Reynolds number is relatively low. Thus, the results of the numerical model and estimates are accompanied by greater error, because we

are moving in the transition region between laminar and turbulent model, where definition of mathematical model is very difficult. So turbulent model $k-\omega$ will also be used, because it is useful for tasks with heat transfer for lower Reynolds number.

Creating geometry and mesh

According to the above dimensions the mesh has been formed, see **Fig. 7.23**

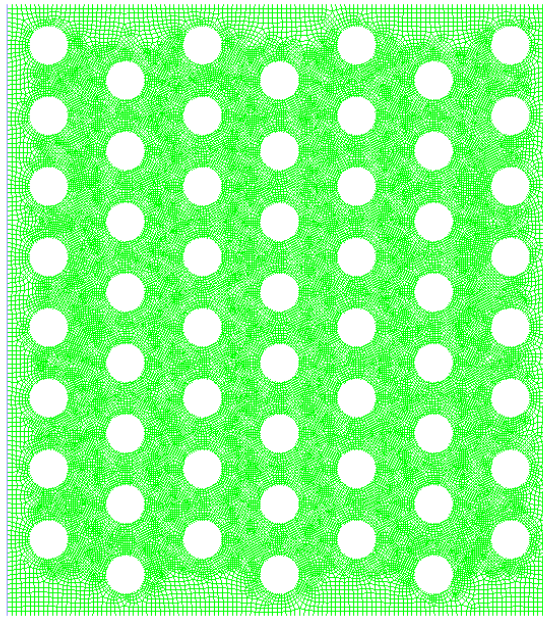


Fig. 7.23 Geometry and mesh

Results

Fig. 7.24 shows a decrease of static pressure in the flow direction and, moreover, irregular distribution close to tubes due to velocity changing, swirl eventual flow separation behind the tubes.

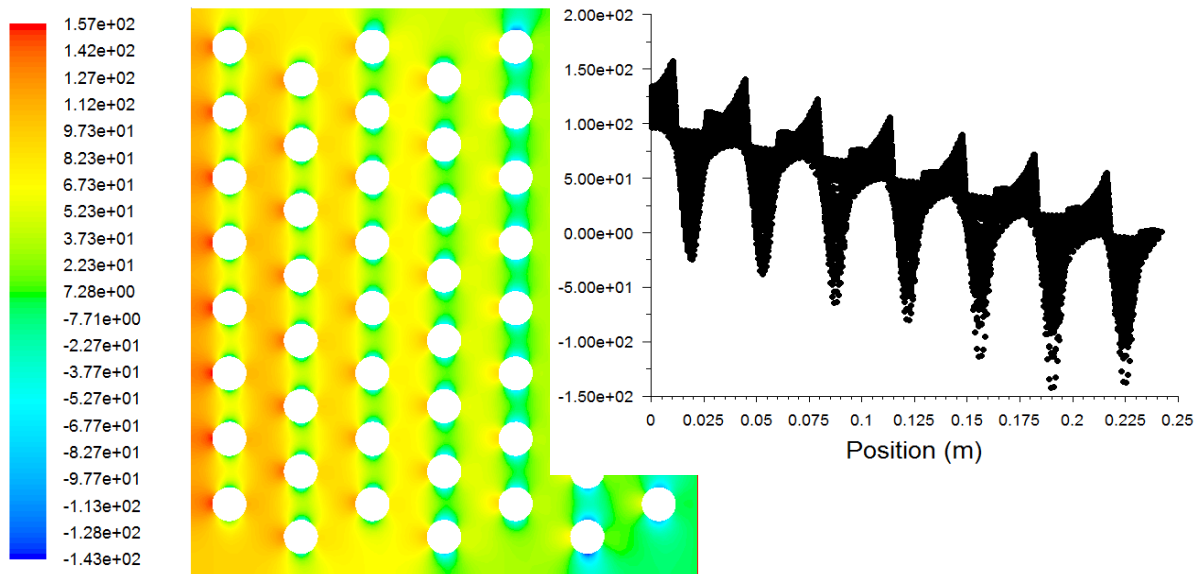


Fig. 7.24 Distribution of static pressure inside the area and a detail of pressure distribution inside the area vs. length area in PLOT X-Y

It is interesting the temperature distribution at the inlet (constant) and at the outlet (periodicity of temperature), see Fig. 7.25.

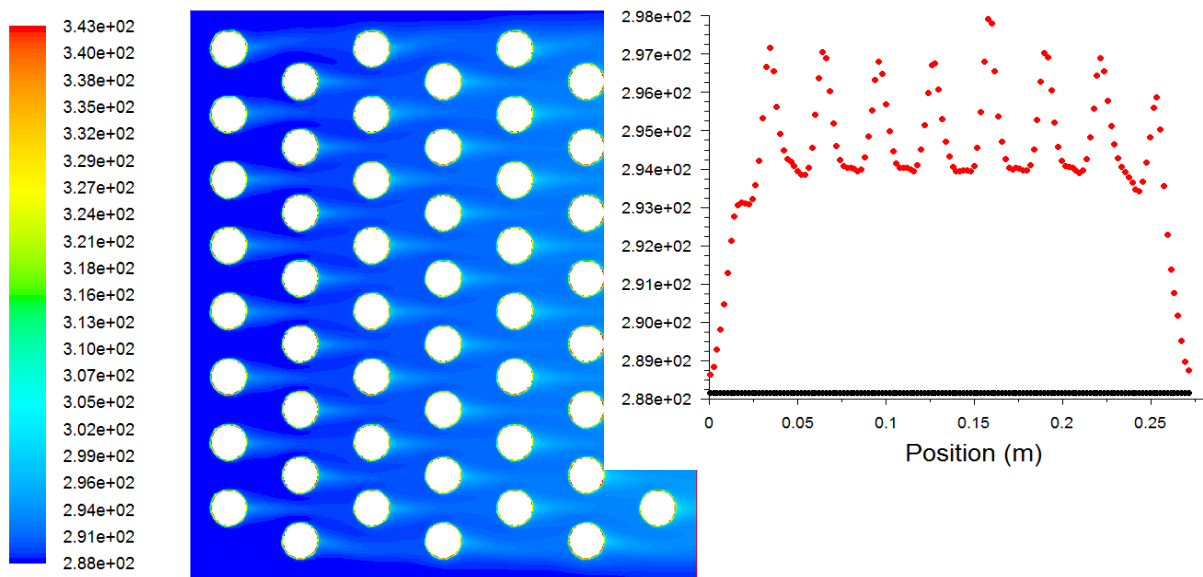


Fig. 7.25 Static temperature in the whole region and at the input (constant) and the output (periodicity is given by bypassing a number of tubes)

Very informative dependency is again evaluated in Fig. 7.26 and Fig. 7.27, i.e. the heat transfer coefficient and Nusselt number. It is interesting to observe the periodicity of the values for which it is necessary to determine the area averaged values and to compare the

results of numerical solution with empirical estimates. See that the progression of these functions do not change much behind the fourth-fifth series of tubes.

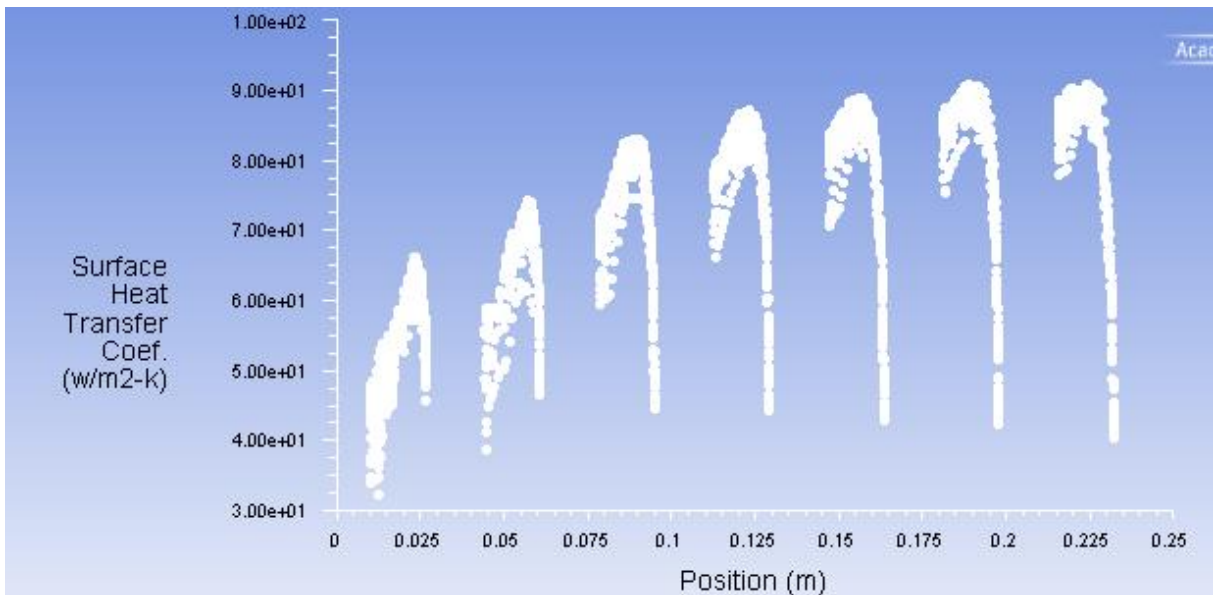


Fig. 7.26 Surface heat transfer coefficient on the walls of the tube

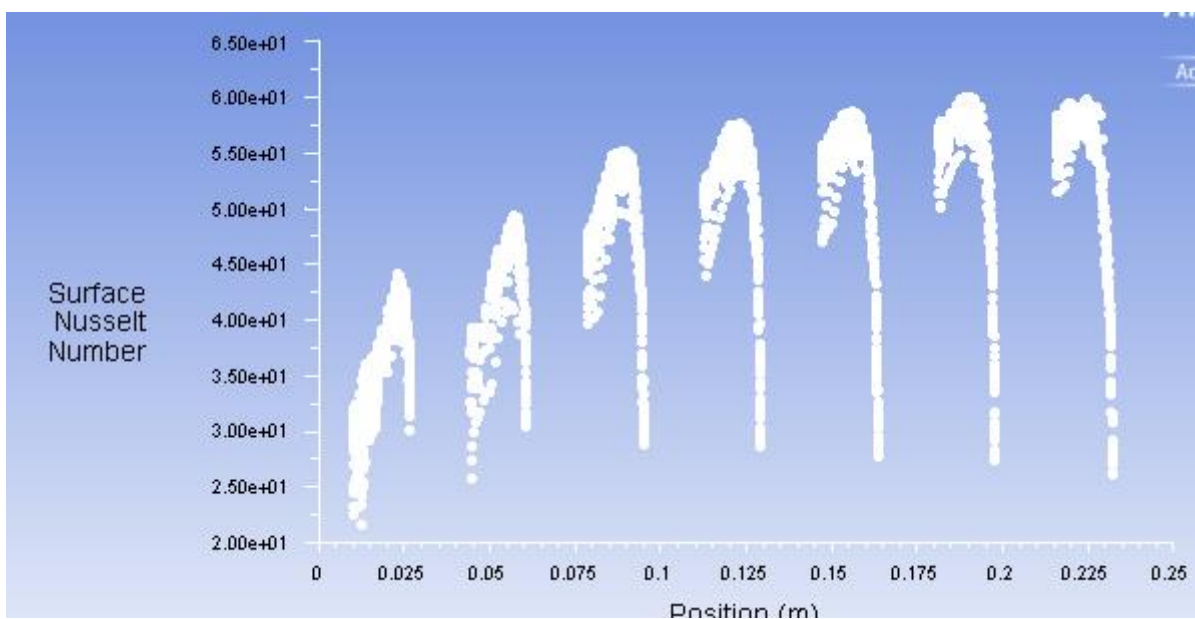


Fig. 7.27 Nusselt number evaluated on the walls of the tube

The results of theoretical - empirical estimation of significant quantities during the air flow through the tubular system with heat transfer are compared in the following table with the averaged values obtained from the numerical calculation. The differences are significant and

are caused by bypassing the first three to five tubes. Average logarithmic temperature in the numerical calculation does not occur.

Tab. 7.21

	estimation	<i>k-ω</i> model adapt
Heat transfer coefficient [W·m⁻²·K⁻¹]	111.28	87.65
Nusselt number [1]	72.13	50.8
Mean logarithmic temperature	51.32	
Thermal power [W]	15289.96	11302.23
Pressure drop [Pa]	376.06	118.91

8. Analysis of the heat exchangers

Heat exchangers are devices that provide internal transfer of thermal energy (enthalpy) between two or more fluids, between a solid surface and the fluid, or between the particles and the fluid, in their interaction delivered without external work and heat. Fluids may be generally one-component or may be a mixture of both single- and binary. Typical applications are two-fluid heaters and coolers of fluids, wherein the two fluids are separated by a solid wall, and evaporators in thermal and nuclear power plants. Heat exchangers can be divided according to the construction, working fluid, the principles of work and many other different criteria.

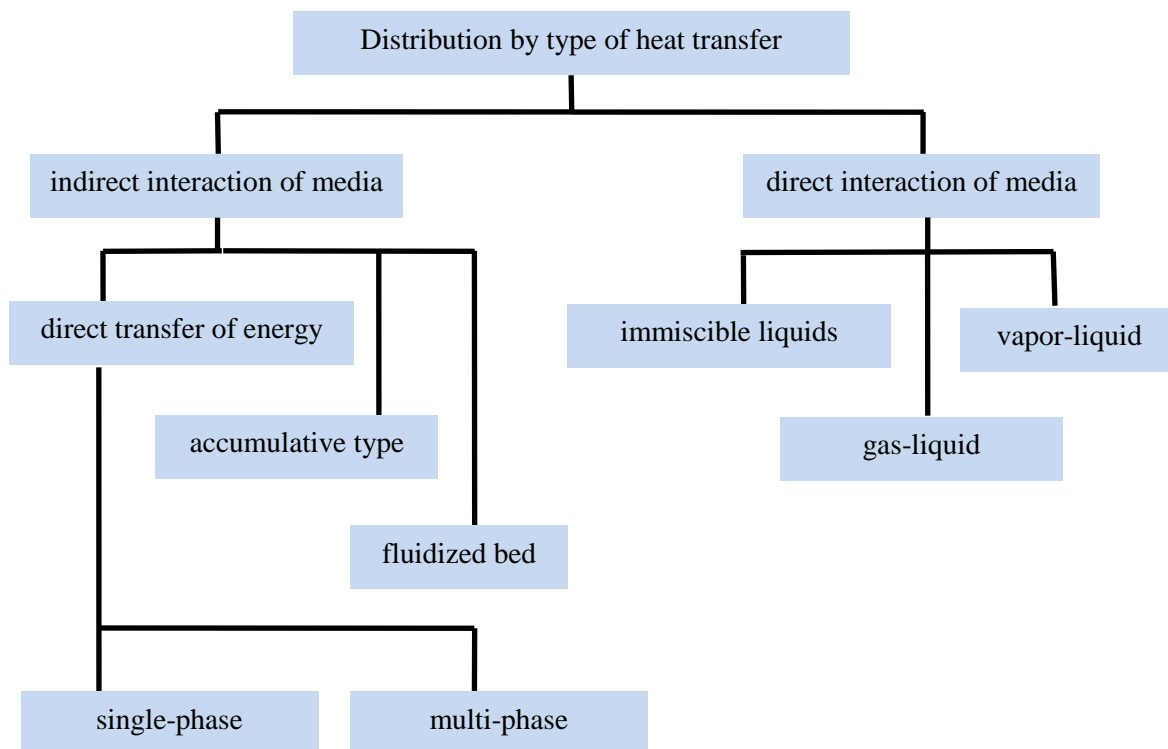


Fig. 8.1 Distribution of heat exchangers by the heat transfer process

Depending on the number of flowing liquids there are two-, three- and N-fluid heat exchangers. Compact heat exchangers have a heat exchange surface density greater than $700 \text{ m}^2 \cdot \text{m}^{-3}$ and noncompact ones less than $700 \text{ m}^2 \cdot \text{m}^{-3}$. According to the construction, there are tubular, plate, with extended area by ribbing etc. and regenerating.

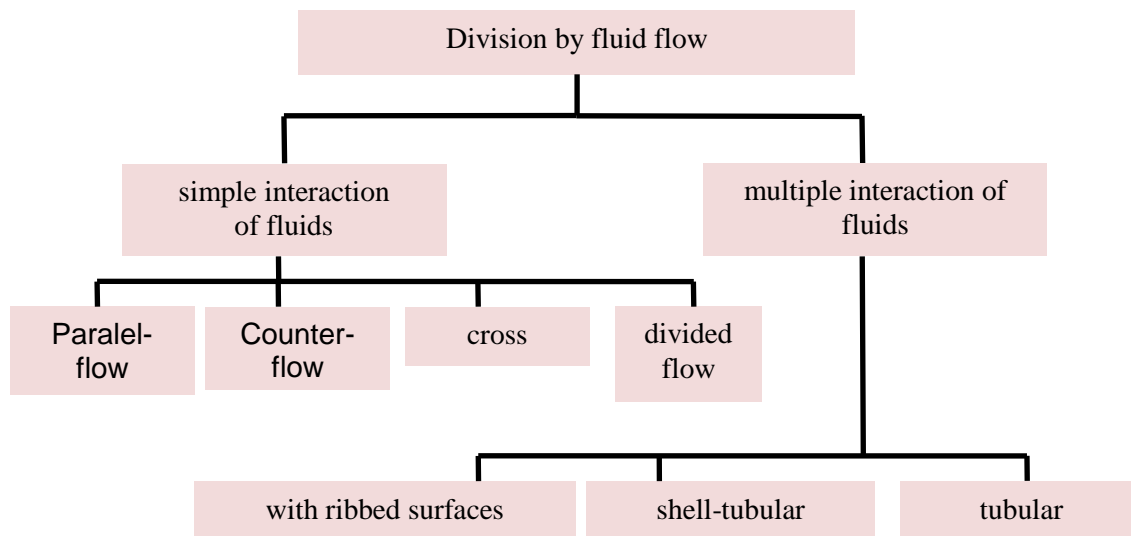


Fig. 8.2 Division of heat exchangers by media flow

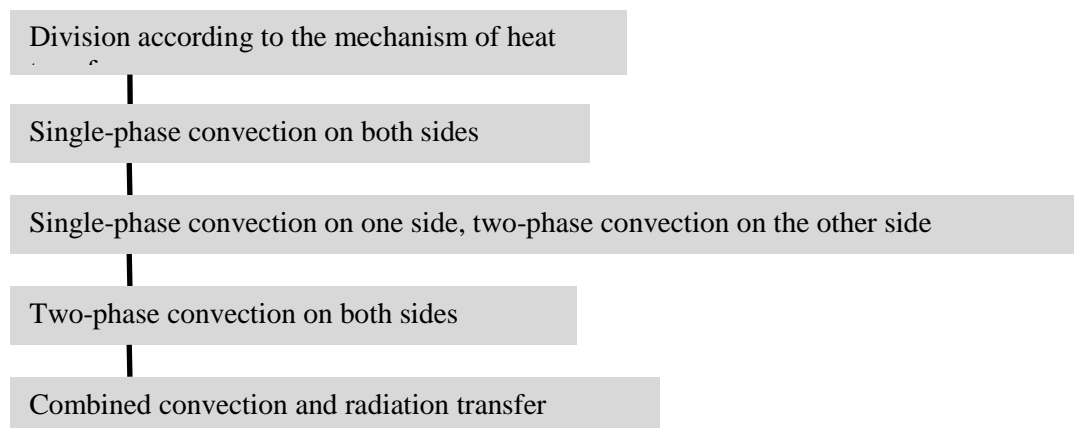


Fig. 8.3 Division of heat exchangers according to the mechanism of heat transfer

8.1. Basic types of heat exchangers and their description

8.1.1. Exchanger of fluid-fluid tubular type

Heat exchangers of fluid- fluid type are most common heat exchangers, in particular the tubular, tubus, spiral heat exchangers, whereby it is used usually the liquid-gas system or one of the fluids can change phase (boiling, evaporation, condensation). Heat exchangers may be paralel-flow, counter-flow and cross type. Examples include heat exchangers in boilers, superheaters, evaporators, economizers, parallel-flow and counter-flow water coolers of stationary hydraulics etc. Their design is very varied and depend on the type and purpose of the installation of the heat exchanger.

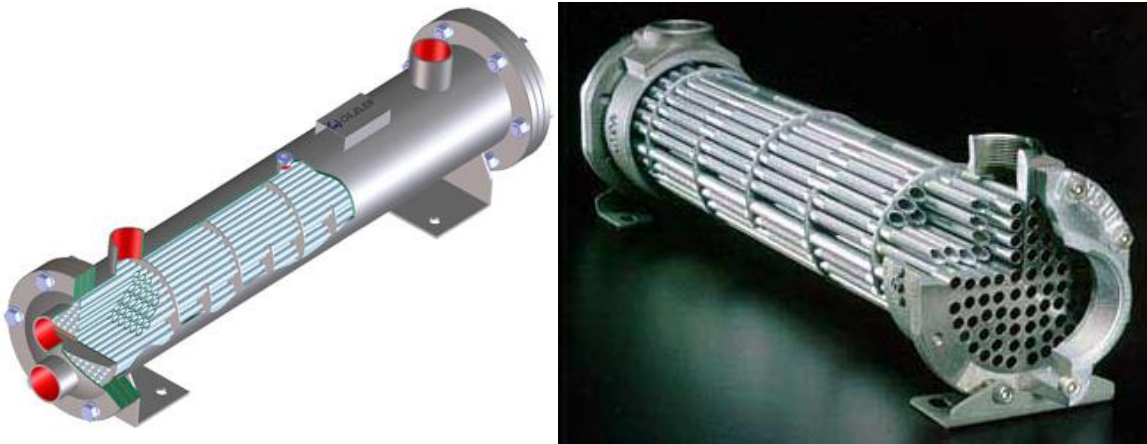
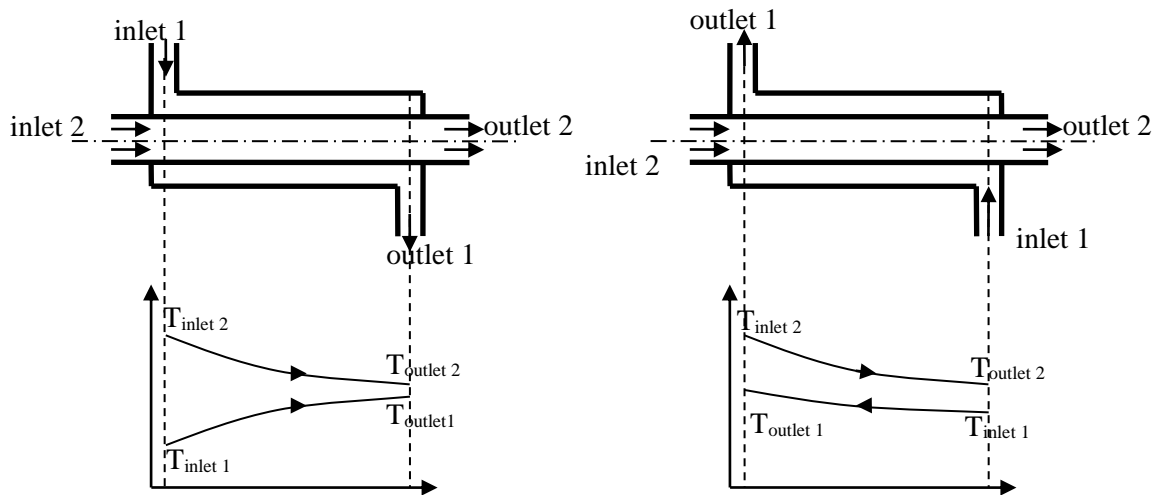


Fig. 8.4 Scheme a tubular paralel-flow and counter-flow heat exchanger with the function of temperature vs length of the heat exchanger and its implementation [3] , [20] , [21]

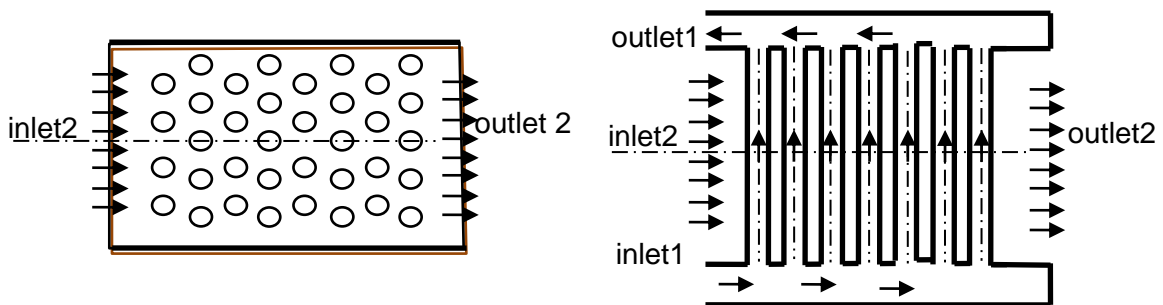


Fig. 8.5 Scheme of cross and tubular heat exchanger [3]

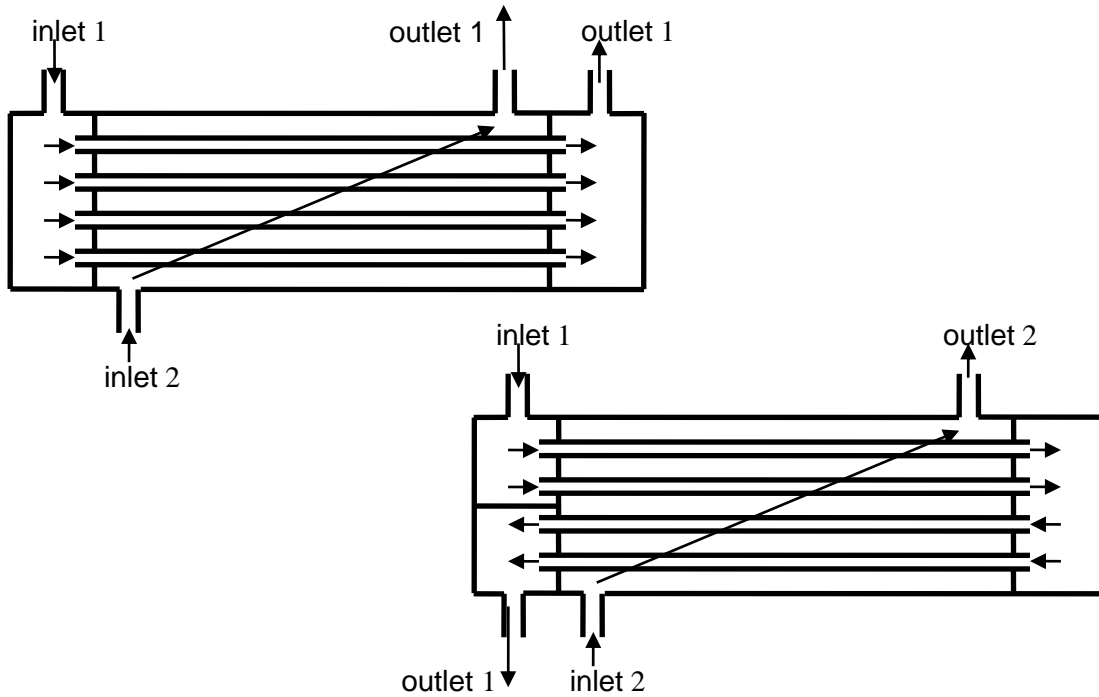
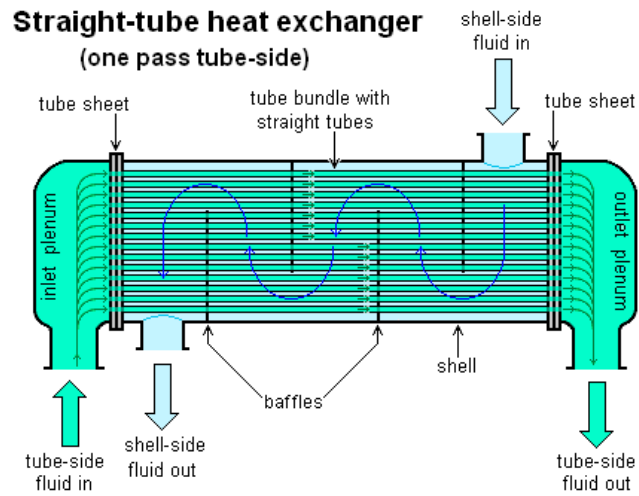


Fig. 8.6 Scheme of shell-tube heat exchanger [3]



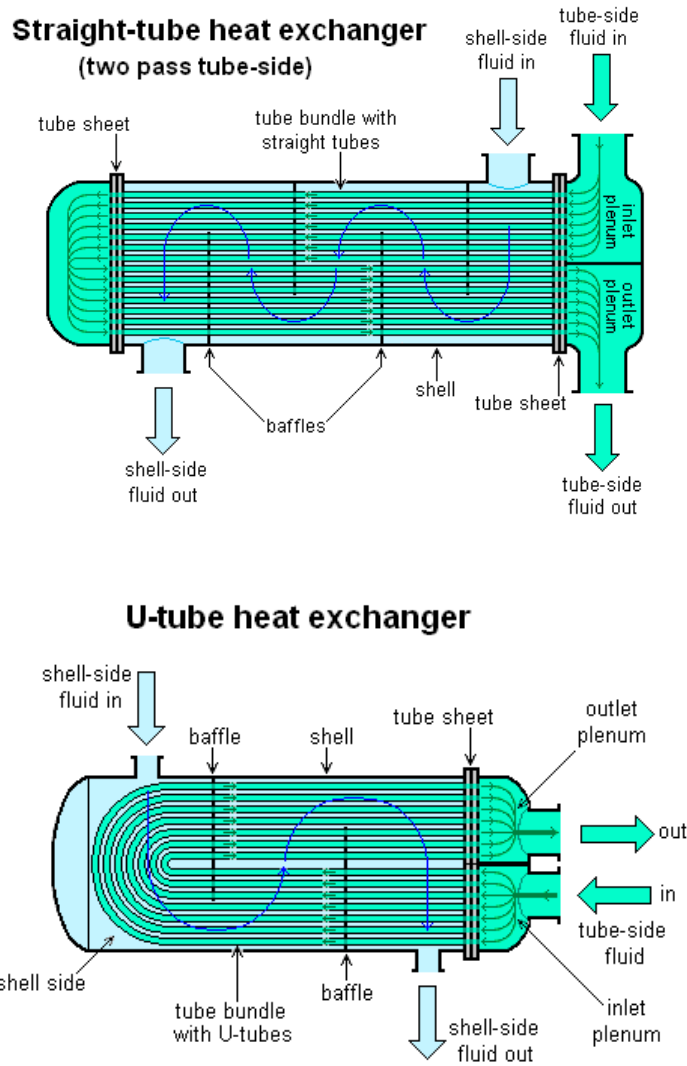


Fig. 8.7 Scheme of the shell-tube heat exchanger with a different location of inputs and outputs and baffles [22]

8.1.2. Finned heat exchangers

This type of heat exchangers consists of a flat plates, provided with ribs made from thin sheet metal. For the gas-liquid exchanger, the heat exchange surface on the gas side is always riveted. If it is a gas-gas heat exchanger, the area may be ribbed on both sides of the exchanger.

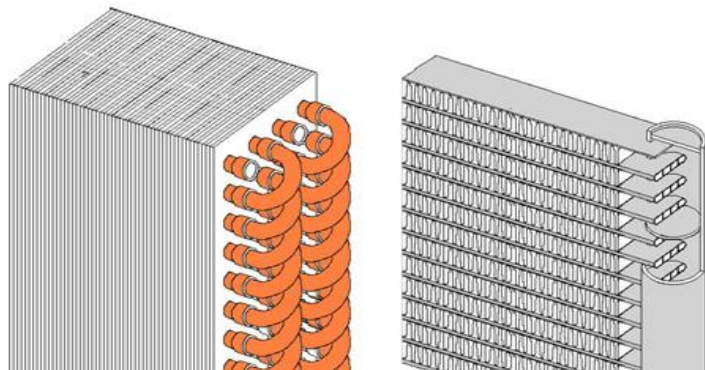


Fig. 8.8 Scheme of finned heat exchanger [23].

For liquids the ribs can not be used because of large pressure force of thin ribs and the possibility of collapse of ribbed structure. The ribs are generally made from a material with high thermal conductivity, eg. copper or aluminum, and are made of thin sheet bending or shearing process, see **Fig. 8.10**. Average number of ribs is then 120-700 fins per 1 m of length. At the heat exchangers with high power, this value may be up to 2100 ribs per 1 m of length. This ensures a high heat transfer surface, which can be up to $1300 \text{ m}^2 \cdot \text{m}^{-3}$. The following figure shows the typical configuration of the heat exchanger of type liquid-gas and gas-gas. Structural arrangement can be very varied and again depends on the type of application for which the exchanger is used.

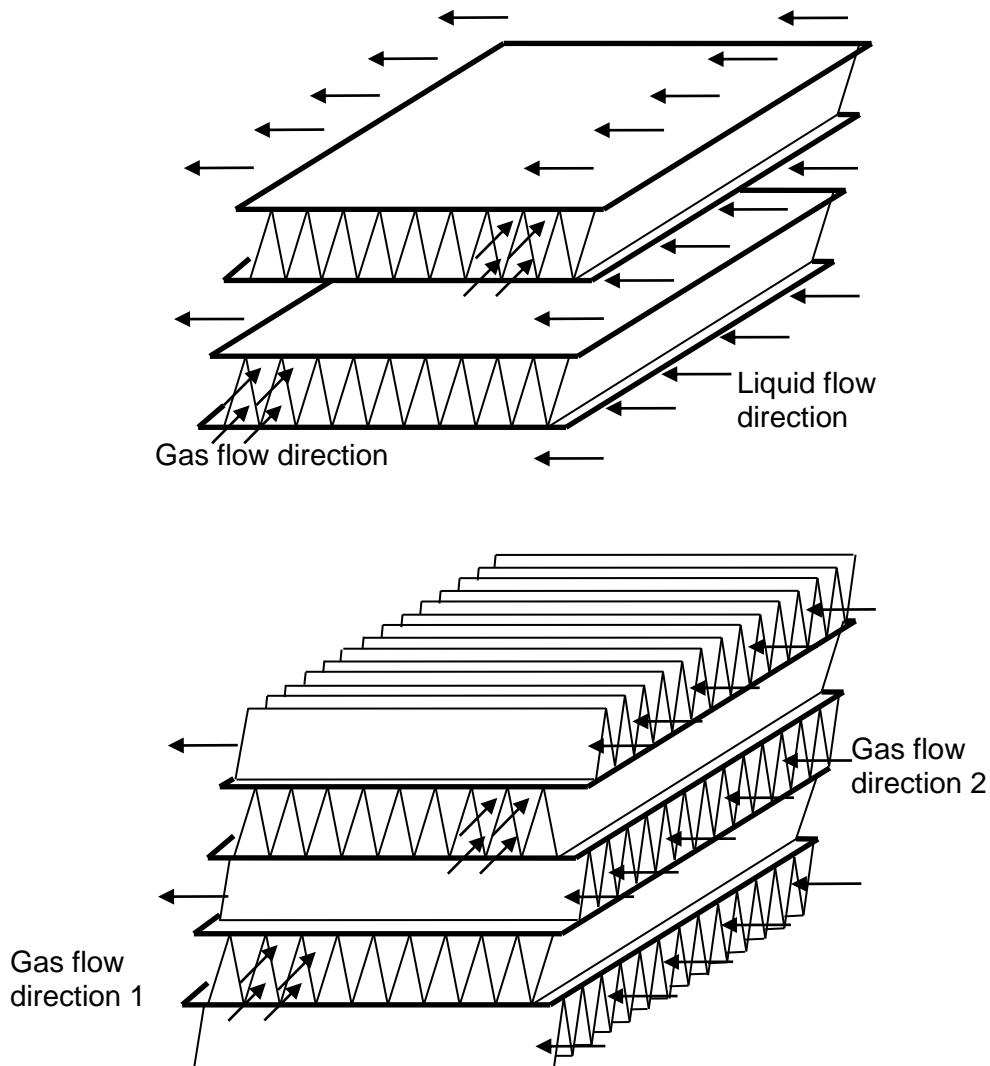


Fig. 8.9 Typical configuration of finned heat exchanger gas-gas and gas-liquid [3] .

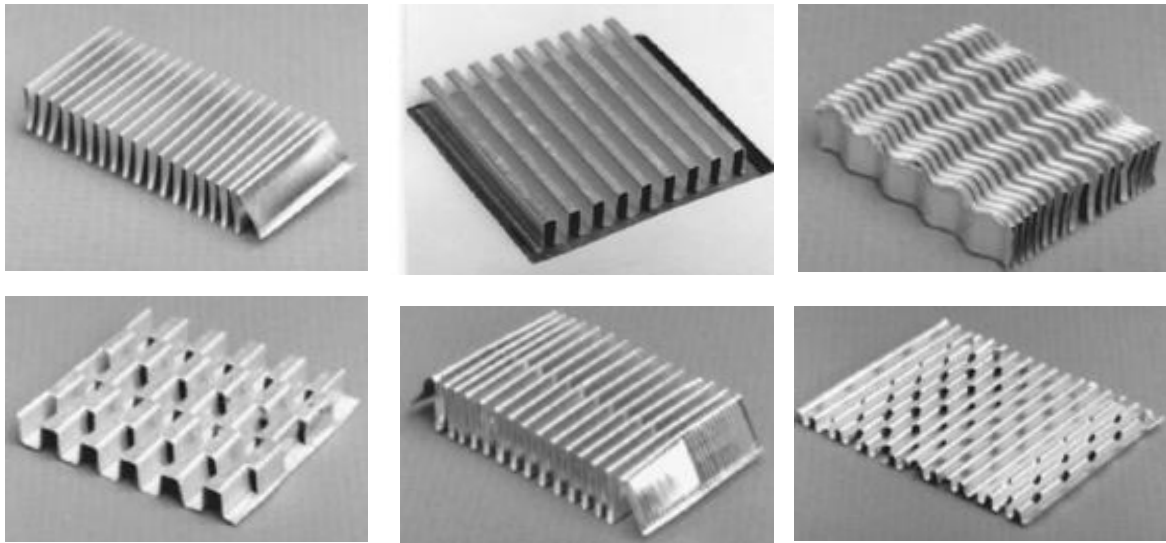


Fig. 8.10 Examples of finn [3] .

Finned exchangers are mainly used in the field of cooling of liquids and gases through the flowing air. It is thus eg. water cooler of an internal combustion engine, various cooler of air-conditioning units, radiators, hydraulic circuits in mobile hydraulics. The cooler is usually fitted with a propeller fan, which provides sufficient air flow. For the car the air flow exists due to moving of vehicle (ie. ram air cooling) and the fan operates only when the vehicle is not moving longer and engine possibly air conditioning unit is still running.

8.1.3. Plate heat exchangers

Plate heat exchangers are constructed from thin plates (sheets), which separate media. This type of heat exchangers has a relatively large heat transfer area, but on the contrary it is not designed for high pressures and temperatures, as well as temperature and pressure difference. Plate heat exchangers are structurally very simple and variable. On the base axis two mirror plates are placed to separate fluid. Depending on the desired power a sufficient number of plates is then placed on a beam and the entire heat exchanger is ended by lid. Each plate is fitted with a seal made of elastomer which ensures the separation of media. The seal on each plates is alternating, ensuring the periodic alternation of media between the plates.

This type of heat exchanger is predominantly compact with a relatively large heat exchange surface (rib) and uses in cases where it is necessary to keep following criteria

- both fluids must be clean and shall not cause corrosion because the heat exchanger has a small hydraulic diameter due to the small flow channels
- heat exchangers are characterized by relatively high pressure drop which is proportional to heat exchanger power
- pressure and temperature of the media are limited by design, thickness of plates and resistance of seal
- this type of heat exchangers is compact and has a large heat transfer area that is to $6000 \text{ m}^2 \cdot \text{m}^{-3}$.

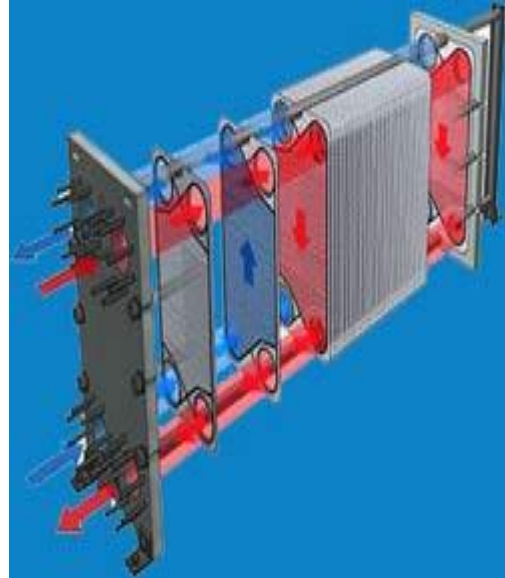


Fig. 8.11 An example of a plate heat exchanger [19]

Plate heat exchangers are used wherever great power is required, with relatively small installation dimensions. But they can not be used in heavy applications, due to their sensitivity to pollution and consequent increase in pressure drop.

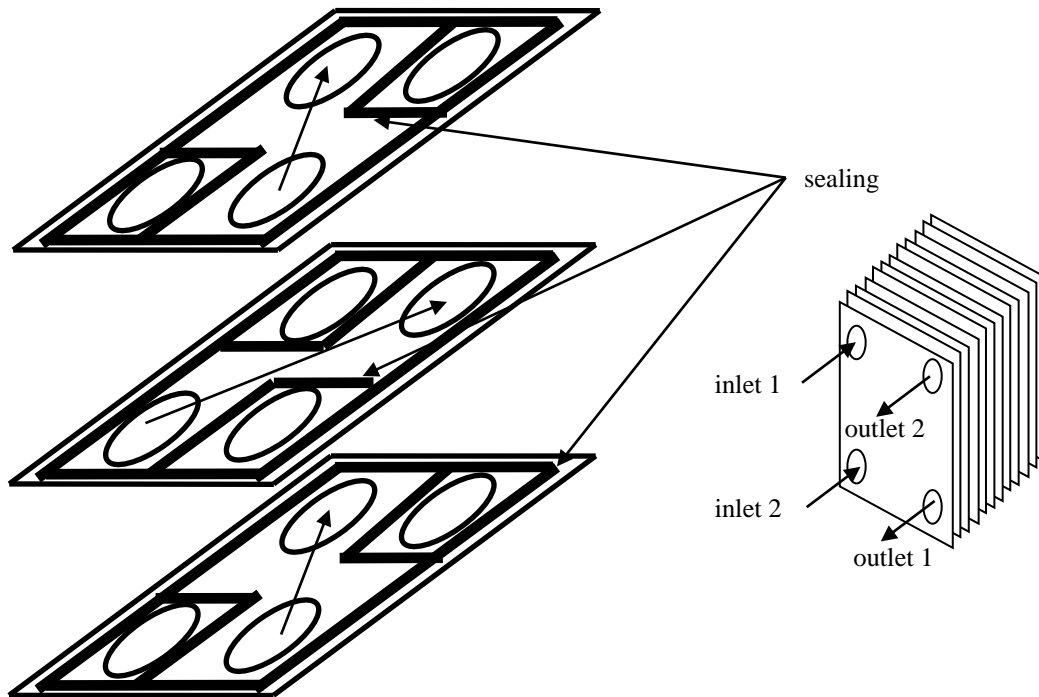


Fig. 8.12 Schema of the plate heat exchanger [3] .

8.2. Thermal power and pressure drop of heat exchanger

8.2.1. Thermal power

The thermal power and pressure drop are two basic design parameters of heat exchangers. For simplicity, the basic calculation formulas are based on a simple exchanger, which will separate two fluids using solid walls of a given thickness, see **Fig. 8.13**, [3] .

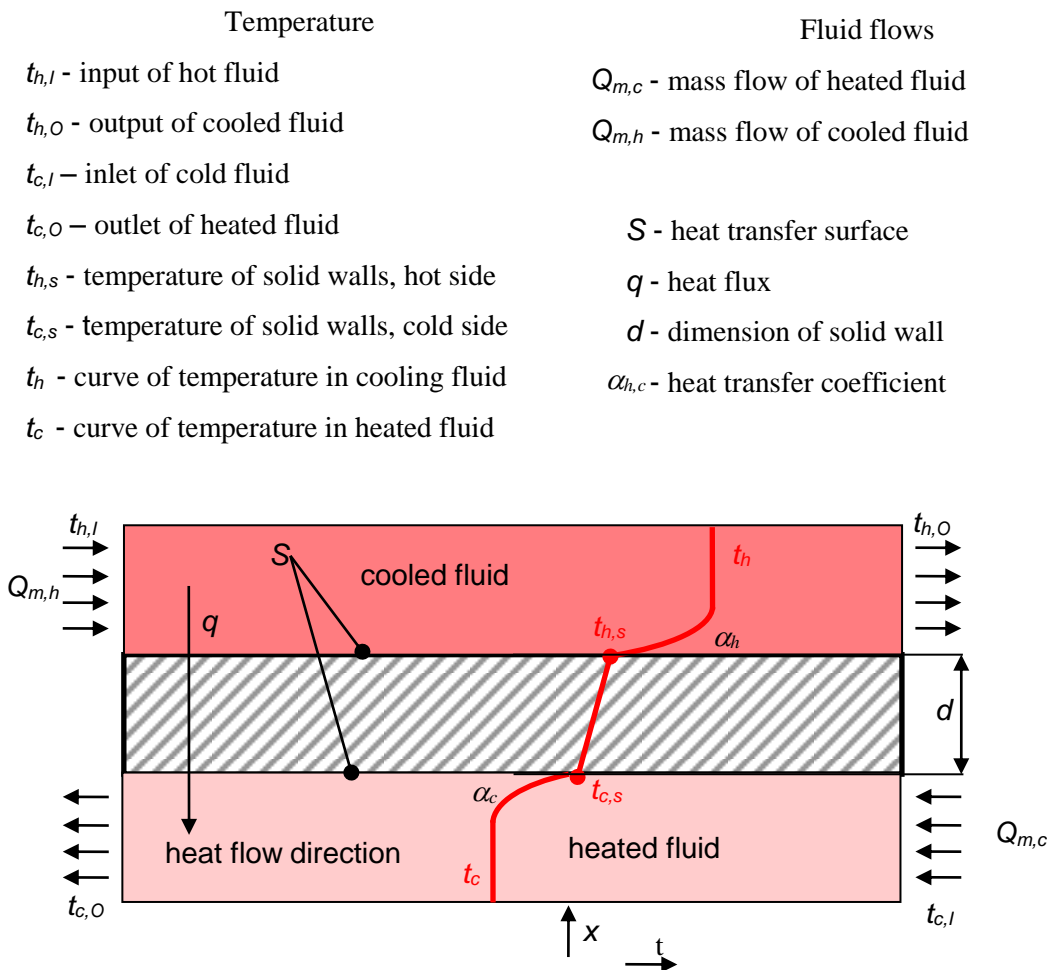


Fig. 8.13 Scheme of fluid and heat flows in heat counterflow exchanger [3] .

Energy analysis is based on **calorimetric equation**, which describes the exchange of heat between two immobile bodies. At substitution of the bodies mass in calorimetric equation by the fluid mass flow we obtain an equation for the power of the heat exchanger. Index c denotes cool fluid, h denotes heat fluid, l denotes input of fluid, O denotes output of fluid. Because the law of energy conservation is valid, at perfectly insulated system the thermal power for cooling and heating fluid is identical. For heated fluid (indexing c) the power P_c is positive because the output temperature of the fluid is higher than the input

temperature $t_{c,O} > t_{c,I}$. In other words heated fluid receives heat, so the power is positive. For cooling fluid (indexing h) on the contrary the outlet temperature of media is lower than the input temperature $t_{h,O} < t_{h,I}$, power P_h is so negative, because the fluid gives the heat. In absolute terms, however, these powers are identical.

$$\begin{aligned} P_c &= Q_{m,c} c_{p,c} (t_{c,O} - t_{c,I}) \quad [\text{W}] \\ P_h &= Q_{m,h} c_{p,h} (t_{h,O} - t_{h,I}) \\ P &= P_c = -P_h \end{aligned} \quad (8.2.1)$$

where $c_{p,c}$ [$\text{J}\cdot\text{kg}^{-1}\cdot\text{K}^{-1}$] is the specific heat capacity of the cooling fluid (heated), and $c_{p,h}$ [$\text{J}\cdot\text{kg}^{-1}\cdot\text{K}^{-1}$] is the specific heat capacity of the cooled fluid. Both thermal capacity are defined at a constant pressure.

Heat also passes through a solid wall of heat exchanger from the hot fluid to the cold fluid. Heat conduction through the solid wall is described by the following equation

$$P = \lambda \frac{t_{h,s} - t_{c,s}}{d} S \quad (8.2.2)$$

This equation solves only heat conduction in the solid wall. Near the wall, however, the velocity and temperature boundary layer is situated. Thermal boundary layer is associated with heat transfer coefficient that defines how strongly the heat is transferred from fluid to solid wall or vice versa. The equation for heat transfer for hot and cold wall is given by the following equation

$$\begin{aligned} P &= \alpha_c (t_{c,s} - t_c) S \\ P &= \alpha_h (t_{h,s} - t_h) S \end{aligned} \quad (8.2.3)$$

Heat transfer coefficient is related to the dimension of the thermal boundary layer. The temperature boundary layer is a thin layer of fluid near the solid walls, in which the temperature varies from temperature of solid walls to temperature very close to temperature of uninfluenced flow. The velocity boundary layer is similarly defined, it is a thin layer near the wall, where the velocity rises from a zero value on the wall to a value very close to the uninfluenced flow. It is important to remember that the thickness of the thermal boundary layer δ_t and the thickness of the velocity boundary layer δ_v are not identical, and their thicknesses are controlled by different physical processes.

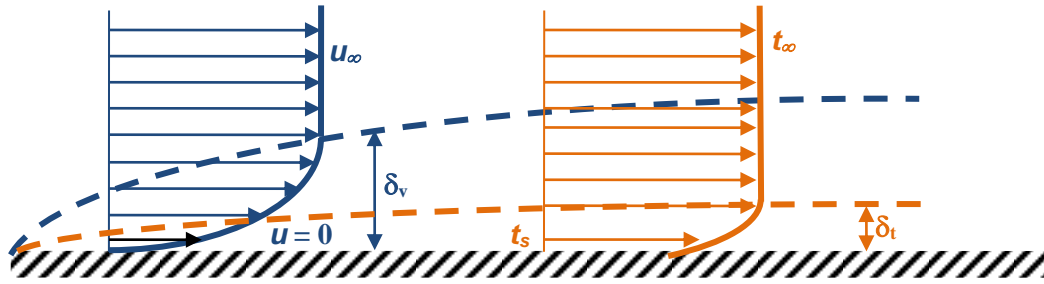


Fig. 8.14 Illustration of velocity and temperature boundary layer

By introducing of heat transfer coefficient in the equation (8.2.2) we obtain the equation for heat transfer.

$$P = \frac{t_h - t_c}{\frac{1}{\alpha_h} + \frac{d}{\lambda} + \frac{1}{\alpha_c}} S \quad (8.2.4)$$

This removes the temperature of the solid wall, which does not interest us in the calculation because it is the inner part of the heat exchanger and only the temperature of the inlet and outlet fluids in the heat exchanger is of interest to us. Furthermore, the new variable is introduced and will be called the heat transfer coefficient.

$$k = \frac{1}{\frac{1}{\alpha_h} + \frac{d}{\lambda} + \frac{1}{\alpha_c}} \quad (8.2.5)$$

After introduction of heat transfer coefficient, then the equation for the power goes into formula

$$P = k(t_h - t_c)S \quad (8.2.6)$$

By analyzing the previous relationship we can thus determine the parameters that affect the power of heat exchanger. If the intention is to maximize the power, then it must be based on the following conditions

- wall thickness should be as small as possible (this is the reason of thin walls in heat exchangers)
- thermal conductivity of the solid walls should be as large as possible (this is the reason why using materials with high thermal conductivity, aluminum, copper, etc.)
- heat transfer surface should be as large as possible (that's why it's large number of ribs, finnes, small tubes etc. in heat exchangers).
- coefficient of heat transfer should be as large as its value can be influenced by fluid velocity, but with velocity increasing, however, the pressure loss increase with the square of velocity.

8.2.2. Pressure drop

Source of pressure and kinetic energy, which ensures the flow of medium through the heat exchanger, is a pump, fan or blower. Pressure loss of the heat exchanger is highly dependent on the physical fluid properties (density, viscosity, etc.). The power, which is necessary to supply the fluid flow through the exchanger in a given amount, can be determined by the pressure drop from the following equation, see [3] , [11] :

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

$$P \approx \frac{1}{2} \frac{\eta}{\rho^2} \frac{4l}{d_h} f(\text{Re}) \quad \text{for laminar flow} \quad (8.2.7)$$

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

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 heat exchanger.

Generally, the pressure loss of the exchanger depends on the following parameters:

- frictional losses associated with friction (viscous) forces of fluid flow around the heat transfer surfaces
- torque effect related to the change of density during the flow in the exchanger
- compression and expansion of the fluid when it flows around bodies (heat transfer surfaces)
- geometric parameters of heat exchanger (for large vertical heat exchanger we must also include static pressure caused by gravity, for gases, this loss is neglected).

Determination of pressure loss is very difficult and in the literature there are numerous empirical and semi-empirical formulas for each type of heat exchanger. Pressure loss in the analytical calculation consists of friction and local losses [3]

$$\Delta p = \left(\lambda \frac{l}{d_h} + \zeta \right) \frac{8}{\pi^2 \rho d_h^4} Q_m^2 \quad (8.2.8)$$

where λ is the friction loss coefficient, ζ the local loss coefficient determined empirically for hydraulic system, l is the length at which there is a heat transfer.

Pressure losses in the exchanger must always be solved at both sides of heat exchanger i.e. for both fluids. In many heat exchangers the determining a pressure drop is based on solution of pressure loss in flow around the body or pressure loss during flow in a closed channel (a tube, a thin gap, etc.). The simplest case occurs in tubular exchangers, where one fluid is flowing inside the tube bundle and a second one flows across the tube

bundle. When flowing around the tube bundle it is possible to determine the pressure loss based on the equation including all losses in the local loss coefficient [3] .

$$\Delta p = \zeta \frac{8}{\pi^2 \rho d_h^4} Q_m^2 \quad (8.2.9)$$

where d_h is the diameter of the tubes.

8.3. Methods for heat calculation of heat exchanger

Heat exchanger calculation can be done by a variety of methods, professional standards, etc. The methods are therefore [3] :

- ε -NTU* method
- P-NTU* method
- MTD** method

*NTU - Number of Transfer Units

**MTD - Mean Transfer Difference

For all methods there are idealized the material properties, it is believed that the specific heat capacity is constant, and if it is a function of temperature, is necessary to calculate the mean temperature of the fluid for which the value of the specific heat capacity will be determined. The same procedure must be applied to other physical properties such as density, thermal conductivity, etc.

8.3.1. Method ε -NTU

In this method, the heat transfer from the hot fluid to the cold fluid in the heat exchanger is represented by the equation [3]

$$P_c = \varepsilon (Q_m c_p)_{\min} (t_{h,i} - t_{c,i}) \quad (8.3.1)$$

$$(Q_m c_p)_{\min} = \text{MIN}(Q_{m,c} c_{p,c}; (Q_{m,h} c_{p,h}))$$

Variable ε represents the effectiveness, which is function of many variables, and may only take values $0 \leq \varepsilon \leq 1$

$$\varepsilon = f(NTU, C^*) \quad (8.3.2)$$

variable $\varepsilon = f(NTU, C^*)$ is expressed by relationship

$$\varepsilon = \frac{Q_{m,h} c_{p,h} (t_{h,i} - t_{h,o})}{(Q_m c_p)_{\min} (t_{h,i} - t_{c,i})} = \frac{Q_{m,c} c_{p,c} (t_{h,i} - t_{h,o})}{(Q_m c_p)_{\min} (t_{h,i} - t_{c,i})} \quad (8.3.3)$$

Variable NTU can get values $0 \leq NTU \leq \infty$ and is defined by

$$NTU = \frac{kS}{(Q_m c_p)_{\min}} = \frac{1}{(Q_m c_p)_{\min}} \int_0^S k dS \quad (8.3.4)$$

Proportional flow rate C^* can take values $0 \leq C^* \leq 1$ and is defined by

$$C^* = \frac{(Q_m c_p)_{\min}}{(Q_m c_p)_{\max}} \quad (8.3.5)$$

8.3.2. Method P-NTU

P-NTU is a variant of the ε -NTU method which removes generality and specifies the calculation due to the different designs of heat exchangers. In this method, the calculation is related to one fluid, because from the relationship there is indicated equality of power in both fluids. In this chapter the index 1 is used for heating liquids and the index 2 for cooling fluid (for simplicity, here is done the procedure of calculation relative to heating liquids) [3].

$$P = P_1 Q_{m,1} c_{p,1} (t_{1,i} - t_{2,i}) \quad (8.3.6)$$

Variable P_1 represents the thermal efficiency, which is a function of variable NTU, temperature resistance R_1 and heat exchanger type (co-flow, counter-flow and cross exchanger)

$$P_1 = f(NTU_1, R_1, \text{type of heat exchanger}) \quad (8.3.7)$$

Temperature resistance R_1 can be determined based on the temperature of both fluids

$$R_1 = \frac{(t_{2,i} - t_{2,o})}{(t_{1,o} - t_{1,i})} \quad (8.3.8)$$

For clarity, we will still find relationships for conversion between fluids 1 and 2

$$\begin{aligned} P &= P_1 Q_{m,1} c_{p,1} |t_{1,i} - t_{2,i}| = P_2 Q_{m,2} c_{p,2} |t_{2,i} - t_{1,i}| \\ P_1 &= P_2 R_2 & P_2 &= P_1 R_1 \\ NTU_1 &= NTU_2 R_2 & NTU_2 &= NTU_1 R_1 \end{aligned} \quad (8.3.9)$$

$$R_1 = \frac{1}{R_2}$$

8.3.3. Method MTD

This method also regulates calculations in method P-NTU using a correction factor F . This method does not include a simple temperature difference but a mean logarithmic temperature difference Δt_{lm} (Log Mean Temperature). The power is then defined by the relationship [3]

$$P = kSF\Delta t_{lm} \quad (8.3.10)$$

Mean logarithmic temperature difference is given by the relationship, which is dependent on the fluid flows in the heat exchanger.

$$\Delta t_{lm} = \frac{\Delta t_1 - \Delta t_2}{\ln \frac{\Delta t_1}{\Delta t_2}} \quad (8.3.11)$$

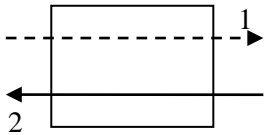
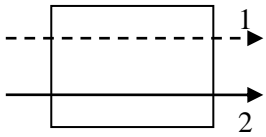
where temperature difference Δt_1 and Δt_2 are defined

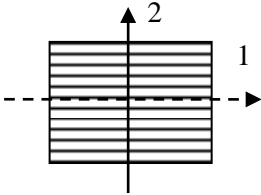
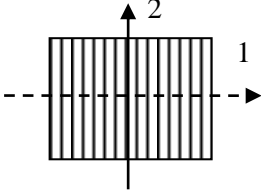
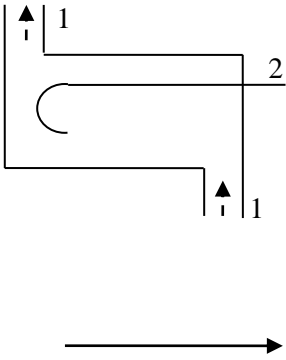
$$\Delta t_1 = t_{h,i} - t_{c,o}, \Delta t_2 = t_{h,o} - t_{c,i} \text{ for all heat exchangers except co-flow one} \quad (8.3.12)$$

$$\Delta t_1 = t_{h,i} - t_{c,i}, \Delta t_2 = t_{h,o} - t_{c,o} \text{ for co-flow heat exchanger}$$

The following table lists the equations for the basic construction types of heat exchangers for the method P-NTU and MTD, see [3].

Tab. 8.1 Basic computational formulas for different types of heat exchangers for method P-NTU and MTD

Type of heat exchanger	formulas
Counter-flow heat exchanger 	$P_1 = \frac{1 - \exp(-NTU_1(1 - R_1))}{1 - R_1 \exp(-NTU_1(1 - R_1))}$ $NTU_1 = \frac{1}{1 - R_1} \ln \left(\frac{1 - R_1 P_1}{1 - P_1} \right)$ $F = 1$
Co-flow heat exchanger 	$P_1 = \frac{1 - \exp(-NTU_1(1 + R_1))}{1 + R_1}$ $NTU_1 = \frac{1}{1 + R_1} \ln \left(\frac{1}{1 - P_1(1 + R_1)} \right)$ $F = \frac{(R_1 + 1) \ln \left(\frac{1 - R_1 P_1}{1 - P_1} \right)}{(R_1 - 1) \ln(1 - P_1(1 + R_1))}$

<p>Cross heat exchanger</p> <p>1 – flows in tubes or finnes</p> <p>2 – flows aroud tubes</p> 	$P_1 = \frac{1 - \exp(-KR_1)}{R_1}$ $K = 1 - \exp(-NTU_1)$ $NTU_1 = \ln \left(\frac{1}{1 + \frac{1}{R_1} \ln(1 - R_1 P_1)} \right)$ $F = \frac{\ln \left(\frac{1 - R_1 P_1}{1 - P_1} \right)}{(R_1 - 1) \ln \left(1 + \frac{1}{R_1} \ln(1 - R_1 P_1) \right)}$
<p>Cross heat exchanger</p> <p>1 – flows aroud tubes</p> <p>2 - flows in tubes or finnes</p> 	$P_1 = 1 - \exp \left(-\frac{K}{R_1} \right)$ $K = 1 - \exp(-R_1 NTU_1)$ $NTU_1 = \frac{1}{R_1} \ln \left(\frac{1}{1 + R_1 \ln(1 - P_1)} \right)$ $F = \frac{\ln \left(\frac{1 - R_1 P_1}{1 - P_1} \right)}{\left(1 - \frac{1}{R_1} \right) \ln(1 + R_1 \ln(1 - P_1))}$
<p>Shell-tubular heat exchanger</p> <p>1 - flows in the tubus</p> <p>2 - flows in the tubes</p> 	$P_1 = \frac{2}{1 + R_1 + E \coth \left(\frac{E \cdot NTU_1}{2} \right)}$ $E = \sqrt{1 + R_1}$ $NTU_1 = \frac{1}{E} \ln \left(\frac{2 - P_1(1 + R_1 - E)}{2 - P_1(1 + R_1 + E)} \right)$ $F = \frac{E \ln \left(\frac{1 - P_1 R_1}{1 - P_1} \right)}{(1 - P_1) \ln \left(\frac{2 - P_1(1 + R_1 - E)}{2 - P_1(1 + R_1 + E)} \right)}$

8.4. Solution of co-flow and counter-flow heat exchanger

To illustrate the flow and temperature field and the definition of the mathematical model, the physical properties of the flowing media and boundary conditions, the simplified scheme of co-flow heat exchanger was chosen. Flowing medium is water and air or possibly again water. The physical properties of the gas are generally highly dependent on temperature, which will be changing. Therefore, in the next the options defining these dependencies will be listed. Physical properties of water may be defined as a function of temperature by methods previously described. In conclusion the graphical evaluation of options will be presented.

8.4.1. Physical properties of gases (kinetic theory)

In chap. 1.3. the basic physical properties were defined. The gas density is given by the ideal equation gas and it is able to take into account the influence of temperature and pressure, ie.

$$\rho V = m \frac{R}{M} T \Rightarrow \rho = \frac{Mp}{RT} = \frac{p}{rT} \quad (8.4.1)$$

Kinetic theory

Other physical quantities can depend on temperature by experimentally determined functions, such as polynomial, table, etc. According to the kinetic energy of an ideal gas [6] the following physical properties and characteristics can be defined:

- viscosity
- thermal conductivity
- specific heat capacity
- mass diffusion coefficients (for the special multi-species mixture)

Definition of the dynamic viscosity μ using kinetic theory is as follows:

$$\mu = 2.67 \cdot 10^{-6} \frac{\sqrt{MT}}{\sigma^2 \Omega_\mu} \quad (8.4.2)$$

where

$$\Omega_\mu = \Omega_\mu(T^*) \quad \text{a} \quad T^* = \frac{T}{(\varepsilon / k_B)} \quad (8.4.3)$$

Function Ω_μ is experimentally determined dimensionless dependence on temperature for air, e.g. [8] :

$$\Omega_{\mu} = \frac{1.16145}{(T^*)^{0.14874}} + \frac{0.52487}{\exp(0.77320T^*)} + \frac{2.16178}{\exp(2.43787T^*)}$$

The formula for the specific heat capacity c_p using kinetic theory is:

$$c_p = \frac{R}{M}(f + 2) \quad (8.4.4)$$

where f is the number of modes of energy (number of degrees of freedom). Thermal conductivity λ using kinetic theory is expressed as follows:

$$\lambda = \frac{15}{4} \frac{R}{M} \mu \left[\frac{4}{15} \frac{c_p M}{R} + \frac{1}{3} \right] \quad (8.4.5)$$

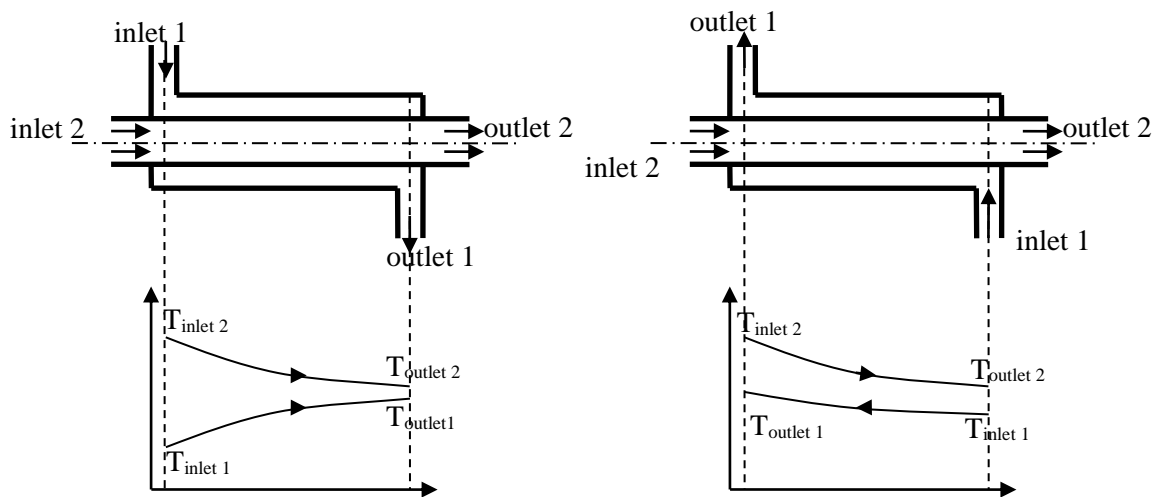
Parameters of air, vapor or other gases for the kinetic theory are given in the database Fluent [9]:

Tab. 8.2 – Parameters of air and vapor (for kinetic theory)

mass	Molecular mass	Lennard-Jones parameters	
	M [kg·kmol ⁻¹]	σ (Å)	ε/k_B (°K)
air	29	3.617	97
vapor H ₂ O	18	2.605	572.4

8.4.2. Co-flow heat exchanger water-water

The flow in the co-flow heat exchanger has been tested to confirm the temperature distributions presented in chap. 8.1. The flow must be assumed very slow in order to transfer the heat in given geometry and to represent a typical temperature drop, or rise of temperature. The optimal flow will be theoretically laminar, unfortunately this type of small liquid flow rate source does not exist. Therefore, the examples in next chapters will be devoted to real flow (i. e. turbulent).



The schema of region with an inlet and outlet faces and a mesh are displayed na Fig. 8.15. Fluid (water) is assumed to be an incompressible liquid with constant physical properties (for higher temperature the properties as a function of temperature can be defined).

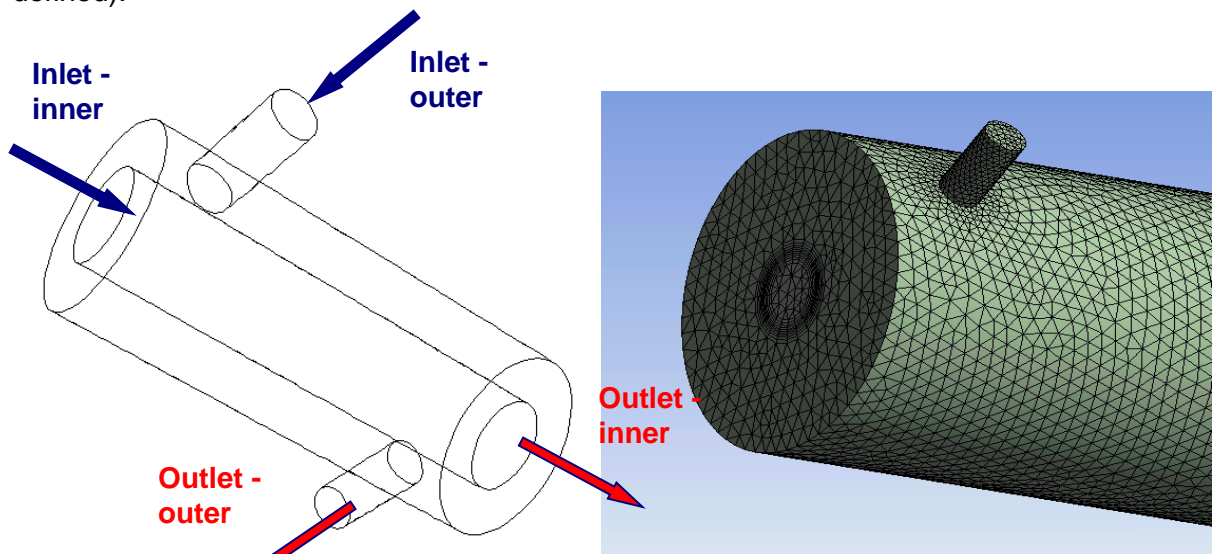


Fig. 8.15 Scheme of region and mesh

Dimensions of region, input and output are given in Tab. 8.3.

Tab. 8.3

Region	$x=$ 0.5	m
	$y=$ 0.12	m
	$z=$ 0.08	m

Inlet – outer –water	$S= 7.65E-05$	m^2	$d= 0.01$	m
Inlet – inner - water	$S= 0.00031214$	m^2	$d= 0.02$	m

For simplicity, a mesh from tetrahedral elements with 131,133 cells was created.

Physical properties and mathematical model

The water flows in both parts of area representing the co-flow cooler. The walls are made from steel tubes of various diameters. Liquid is assumed to be an incompressible liquid with constant physical properties (basic gas parameters can be found and copied from the Fluent database, including the Lennard - Jones parameters).

Tab. 8.4

Physical properties	Units	Label	Water
Density	kgm^{-3}	ρ	998
Specific heat	$J.kg^{-1}K^{-1}$	c_p	4182
Viscosity kinematic	m^2s^{-1}	ν	0.000001
Viscosity dynamic	Pa.s	μ	9.98E-04
Thermal conductivity	$Wm^{-1}K^{-1}$	λ	0.6
Temperature conductivity	m^2s^{-1}	a	1.44E-07

Reynolds number was determined from the mass flow estimation. Its value is low, it is a laminar flow, see Tab. 8.5.

Tab. 8.5

		Tube inner water	Tube outer water
Mass flow rate	$kg s^{-1}$	$Q_m= 0.0003$	0.0003
Velocity	ms^{-1}	$u= 0.00096$	0.0038
Reynolds number		$Re= 19$	38

Boundary conditions

Because this is an illustrative example where the flow trajectories, temperature, density will be monitored, the boundary conditions were pre-defined and modified so that these characteristic variables were clearly visible. The input variable is mass flow rate.

Tab. 8.6

	Mass flow rate [kg ³ ·s ⁻¹]	Static pressure [Pa]	Total temperature [°C]
Inlet - outer	0.0003		20
Inlet - inner	0.0003		80
Outlet - outer		0	
Outlet - inner		0	

Numerical calculation is very stable and converges well. The complexity of the flow can be evaluated by flow trajectories of liquid elements colored by temperature, see **Fig. 8.16**. This phenomena also applies to other variables such as pressure, velocity and temperature, see Fig. 8.17 till Fig. 8.19. Typical temperature distribution along the axis, i.e. decreasing curve in the "outer" area, and increasing curve in the „inner“ area can be evaluated using a graph of temperature in areas parallel to the axis of the tubes is evaluated, see **Fig. 8.20**. A curve typical for the co-flow exchanger could be obtained by converting a graph into Excel and by passing a trend line.

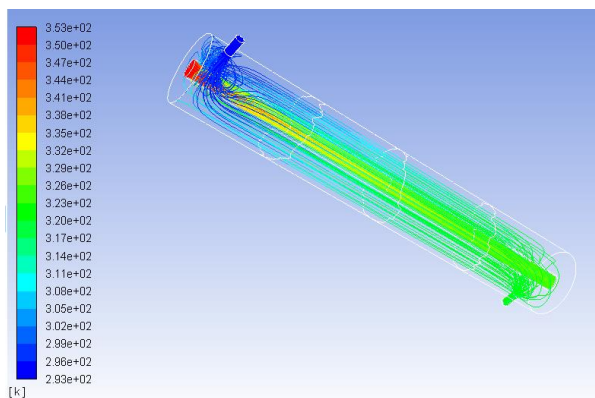


Fig. 8.16 Trajectories of particles colored by temperature

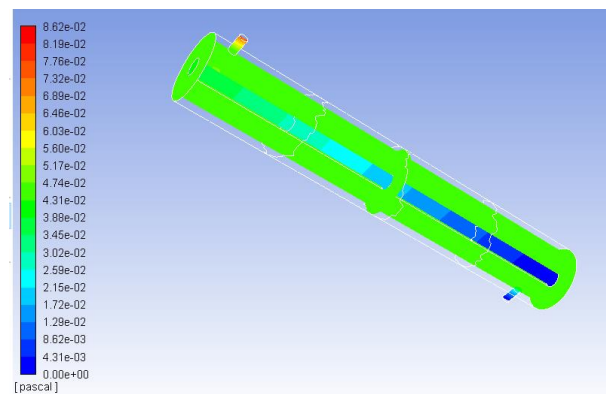


Fig. 8.17 Static pressure in axial direction and in three cross sections

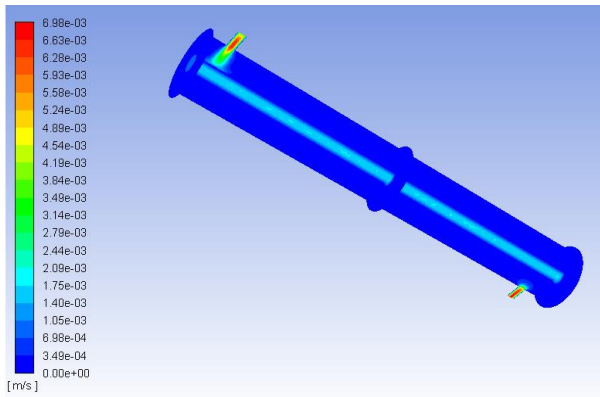


Fig. 8.18 Velocity magnitude in axial direction and in three cross sections

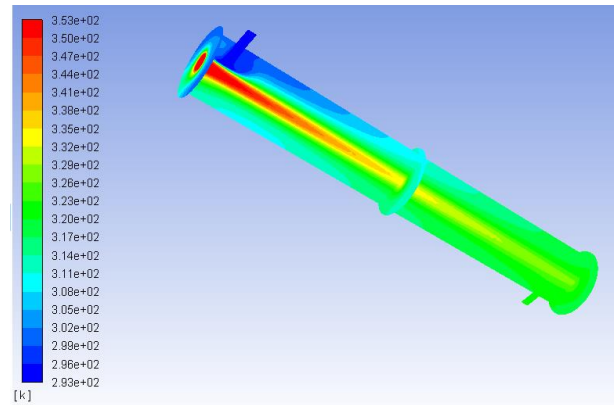


Fig. 8.19 Static temperature in axial direction and in three cross sections

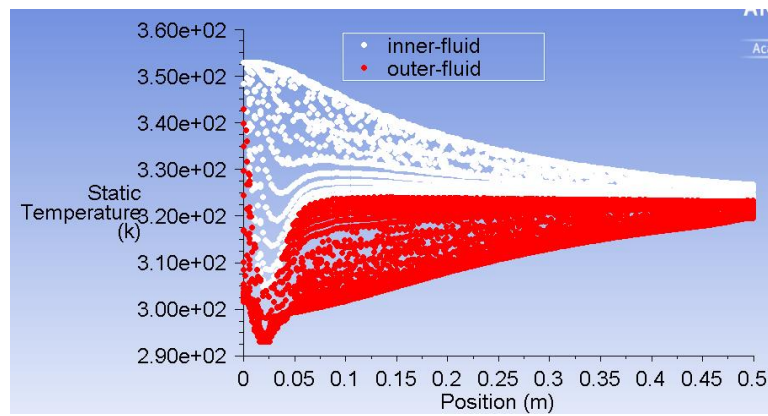


Fig. 8.20 Static temperature in interior - inner, interior – outer

Surface heat flux is a significant parameter specifying the heat transfer and is evaluated on the entire surface of the inner tube or by graph **Fig. 8.21**.

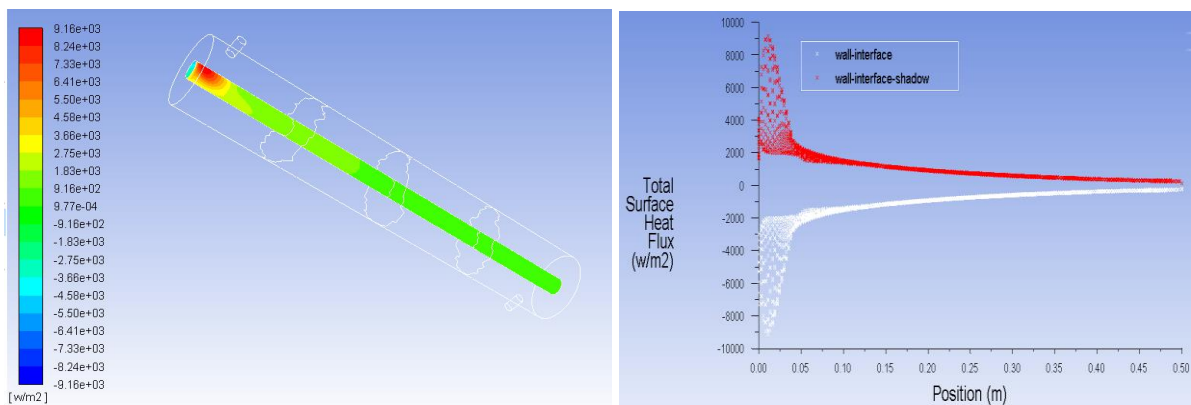


Fig. 8.21 Surface heat flux

The heat transfer coefficient and the Nusselt number are evaluated on the inner wall. The values are based on the heat flux towards the inner or outer flowing fluid. Because the wall is a double-sided wall, it is necessary to determine which wall belongs to the wall-interface and the wall-interface-shadow using Boundary condition command. Surface heat transfer coefficient is determined again by equation $\alpha = \frac{q}{T_w - T_{ref}}$ and subsequently Nusselt number

$$Nu = \frac{\alpha d_{ref}}{\lambda}$$
. It is clear from the definition that the reference values of the temperature and the hydraulic diameter are related to the direction in which heat transfer is determined whether in the direction of the tube inner (wall-interface) or the tube outer (wall-interface-shadow), see Fig. 8.22 and Fig. 8.23.

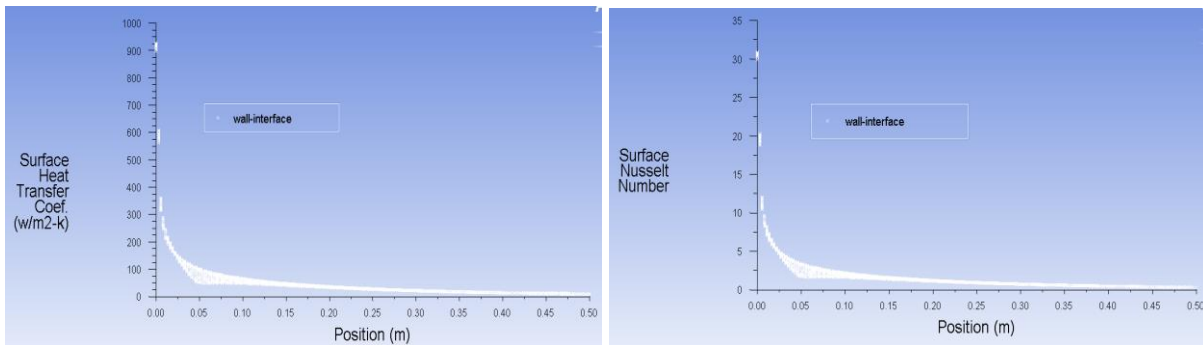


Fig. 8.22 Surface heat transfer coefficient and Nusselt number on the wall interface $T_{ref}=353$ K, $d_h=0.02$ m

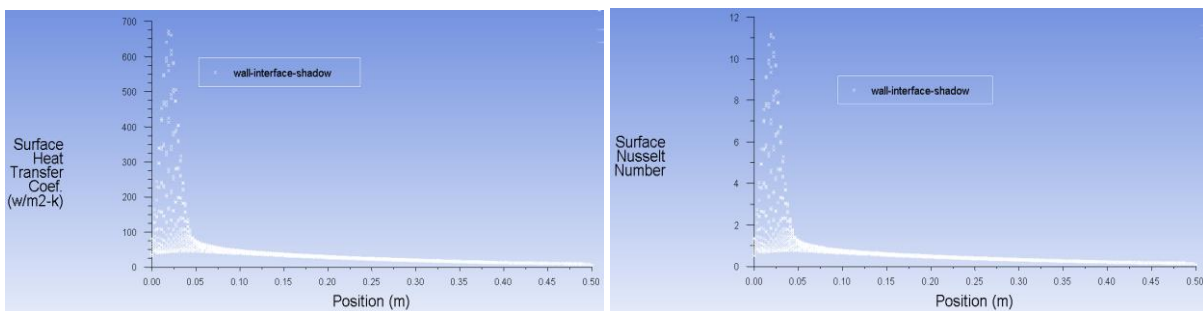
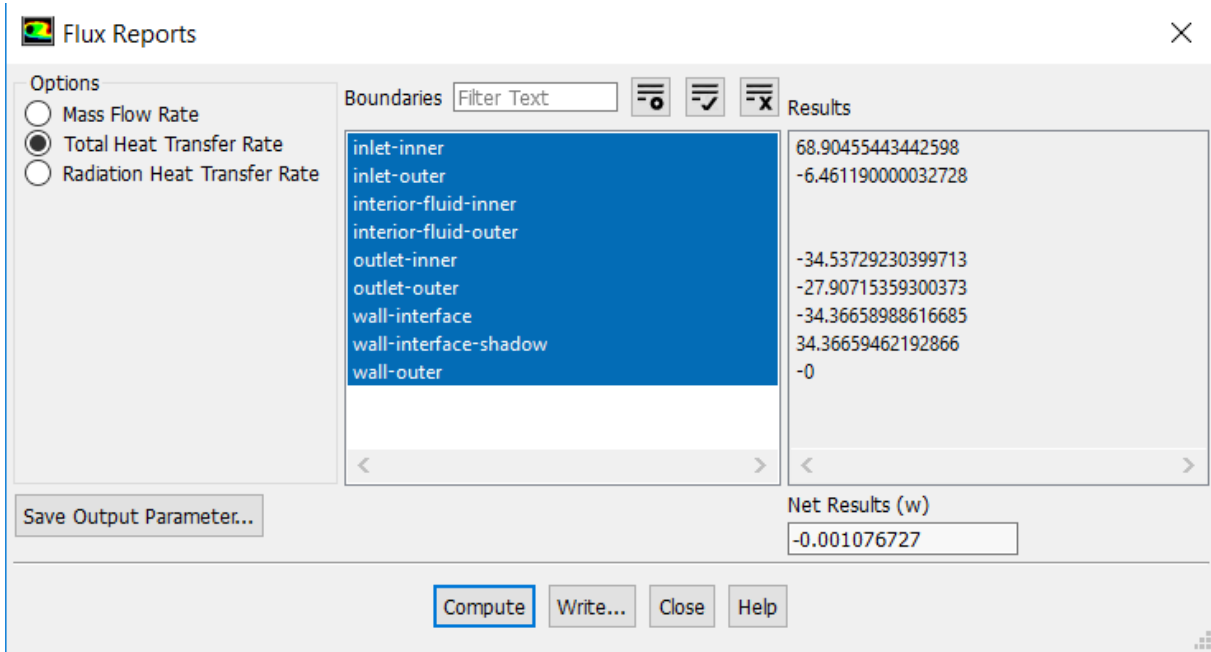


Fig. 8.23 Surface heat transfer coefficient and Nusselt number on the wall interface shadow $T_{ref}=293$ K, $d_h=0.01$ m

The heat power of this heat exchanger across all defined areas can also be determined:



After a detailed calculation of the heat power between the inputs and outputs we see, that the value is the same as the heat power through the wall interface and is equal to 34 W.

From the results its possible to determine average values of outlet temperatures, inlet pressures, heat transfer coefficients, Nusselt numbers, etc., see comparison of this results with results from solution of counter-flow heat exchanger in next chapter.

8.4.3. Counter-flow heat exchanger water-water

Geometry for the counter-flow heat exchanger is the same as that defined in chap. 8.4.2 as a co-flow exchanger. Boundary conditions are the same, only the input for output for the outer area are changed. On Fig. 8.24 till Fig. 8.27 the trajectories of particles colored by temperature, pressure, velocity and temperature in the selected sections are evaluated.

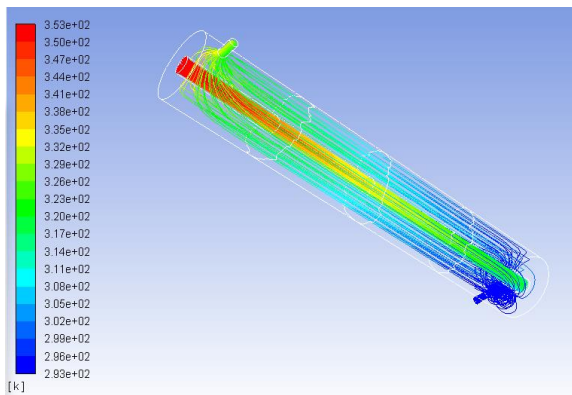


Fig. 8.24 Trajectories of particles colored by temperature

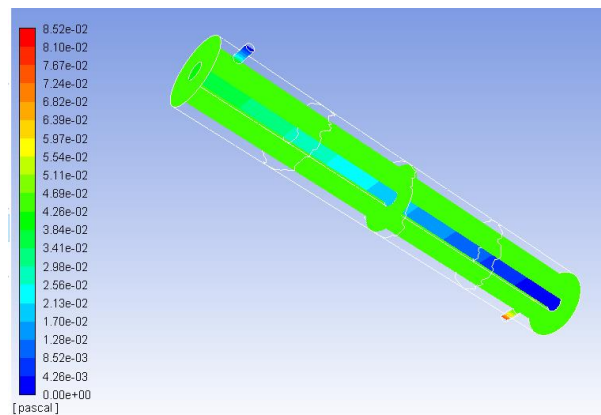


Fig. 8.25 Static pressure in axial direction and in three transverse planes

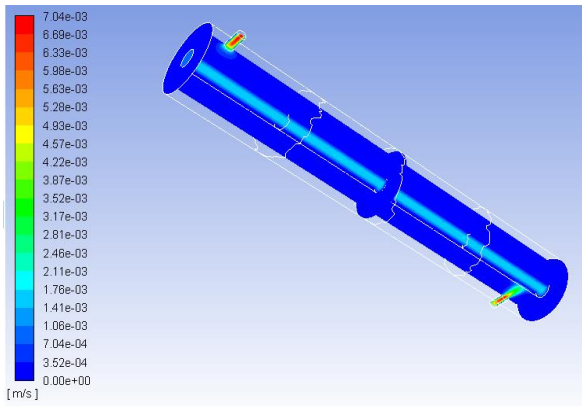


Fig. 8.26 Velocity magnitude in axial direction and in three transverse planes

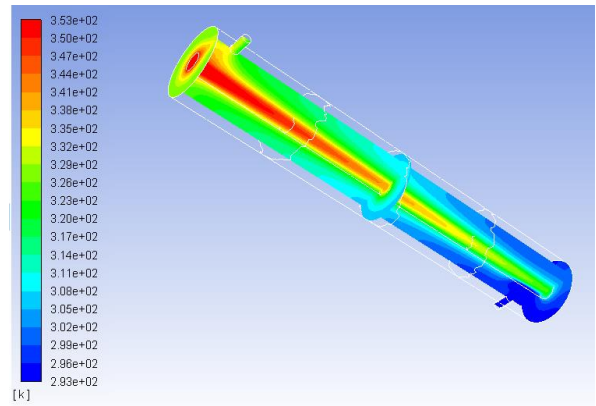


Fig. 8.27 Static temperature in axial direction and in three transverse planes

In Fig. 8.28 it is shown a typical temperature drop in both flow areas, more likely it would be to convert to Excel and use trend line.

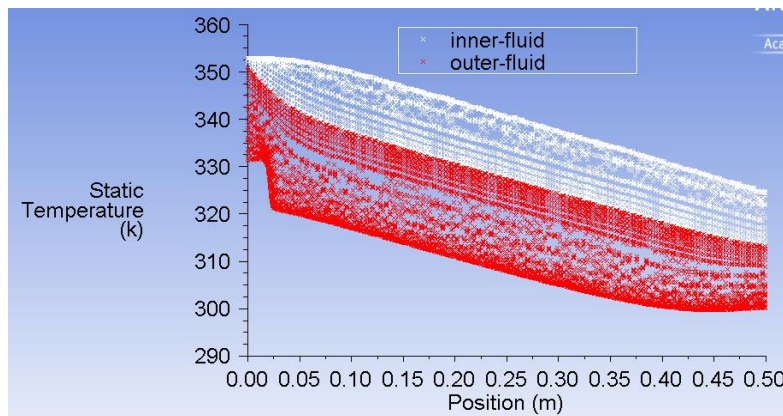


Fig. 8.28 Static temperature in interior - inner, interior – outer

The heat flux through the wall interface and the wall shadow interface, ie on the inner wall is on Fig. 8.29.

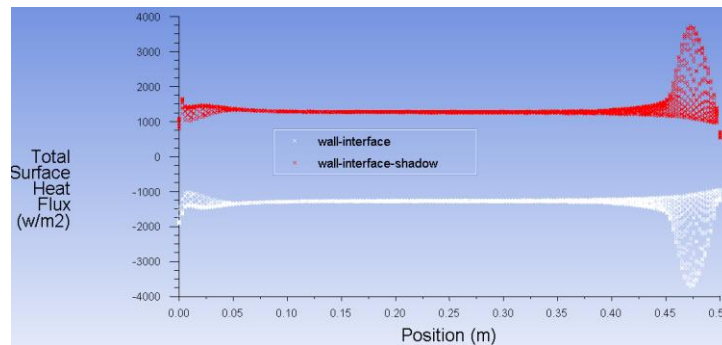


Fig. 8.29 Total surface heat flux

The heat transfer coefficient and Nusselt number are evaluated on the wall based on the

heat flux towards the inner or outer flowing fluid using the formula $\alpha = \frac{q}{T_s - T_{ref}}$ and

$$Nu = \frac{\alpha d_{ref}}{\lambda}$$

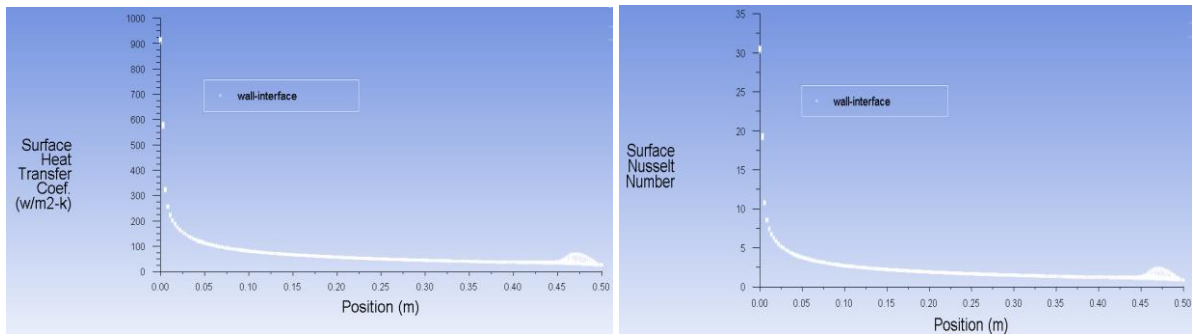


Fig. 8.30 Surface heat transfer coefficient and Nusselt number on the wall interface towards the inner area $T_{ref}=353$ K, $d_h=0.02$ m

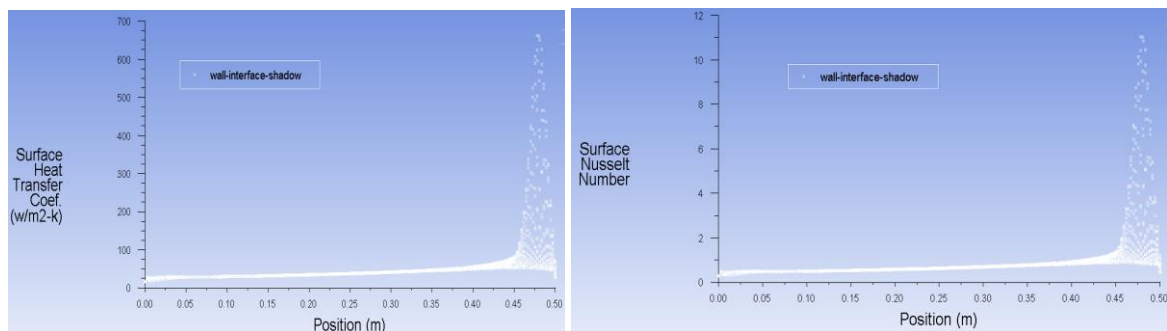
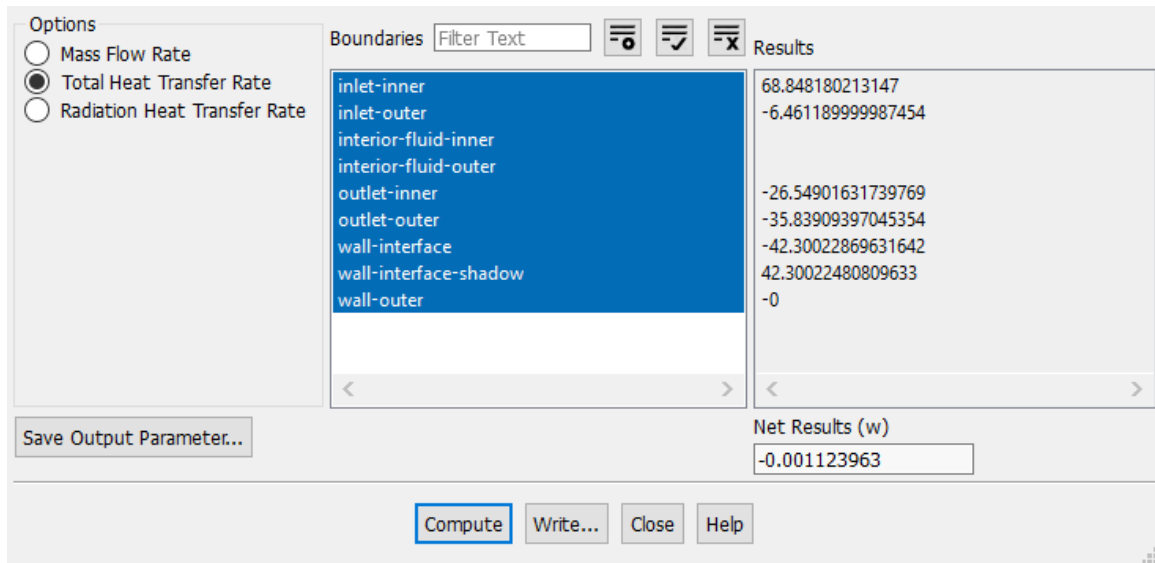


Fig. 8.31 Surface heat transfer coefficient and Nusselt number on the wall interface shadow towards the outer area of the area $T_{ref}=293$ K, $d_h=0.01$ m

The heat power of this heat exchanger across all defined areas can also be determined and heat power accross the tube wall is 42 W.



Tab. 8.7 compares averaged pressure and temperature on inlets and outlets, averaged value of Nusselt number and Surface heat coefficient for co-flow and counter-flow heat exchanger.

Tab. 8.7

	Tube - inner		Tube - outer	
	co-flow Fluent	counter- flow Fluent	co-flow Fluent	counter- flow Fluent
T_{ref}	353	353	293	293
d_{ref}	0.02	0.02	0.01	0.01
Nu-lam	1.52	2.29	0.63	0.81
α -lam	46.12	68.45	36	49
T_{outlet}	325.13	353	320.29	326.29
p_{inlet}	0.039	0.02	0.078	0.078

8.4.4. Co-flow heat exchanger water-air

The geometry of this co-flow water and air heat exchanger was the same as in chap. 8.4.2. Therefore, the same mesh was used. The flow medium in inner part of the exchanger will be high temperature air (substituent of flue gas). The flow will be turbulent and the boundary conditions will be set as the real conditions typical of the stove heat exchanger.



Approaches to solution

Solution of the hot gas flow with heat transfer is a complex problem and complete mathematical model may often lead to divergence. Therefore, so-called step-by-step method (from the simplest to the more complex model) was used to obtain a stable and convergent solution. Subsequently, turbulent models, mesh, wall functions, or boundary conditions are repaired. In our case, this means that the following variants have been solved:

- solution with constant physical properties, turbulent RNG $k-\varepsilon$ model
- solution with physical properties depending on temperature or pressure
- if there are the convergence problems in stable regime, the task must be solved as time-dependent and the statistically averaged values will be evaluated
- solution with a better $k-\omega$ sst turbulent model, which is suitable for low Reynolds numbers
- correction of mass flow rate to get corresponding temperature gradient (evaluation using average values of velocities and temperatures at inlets and outlets)

Physical properties, mathematical model, boundary conditions

The constant physical properties for water and air are taken from the Fluent database. To determine the Reynolds number $Re = \frac{ud_h}{\nu}$, however, air density at 1100 °C was used.

Tab. 8.8

Tube outer	Tube inner
---------------	---------------

pressure	Pa	$\rho_{ref} =$	101325	101325
temperature	°C	$T_{ref} =$	55	1100
temperature absolut	K	$T_{ref} =$	328	1373
density	kgm ⁻³	$\rho =$	998	0.255
mass flow rate	kg s ⁻¹	$Q_m =$	0.03	0.0003
velocity	ms ⁻¹	$u =$	0.38274	3.74482
Reynolds number		$Re =$	3827	4713

For the calculation, water with constant properties and air with physical properties defined by kinetic theory is considered. The mathematical model is specified as turbulent. The task had to be solved as time-dependent and the results are the mean values of all evaluated quantities, including turbulent ones. Following these considerations, the boundary conditions can be defined as follows:

Tab. 8.9

Boundary conditions	Mass flow rate [kg ³ ·s ⁻¹]	Static pressure [Pa]	Total temperature [°C]	Turbulent intensity [%]	Hydraulic diameter [m]
Inlet - outer	0.0003		55	1	0.01
Inlet - inner	0.03		1100	1	0.02
Outlet - outer		0		1	0.01
Outlet - inner		0		1	0.02

Results

The mean value of the basic hydraulic quantities is again evaluated.

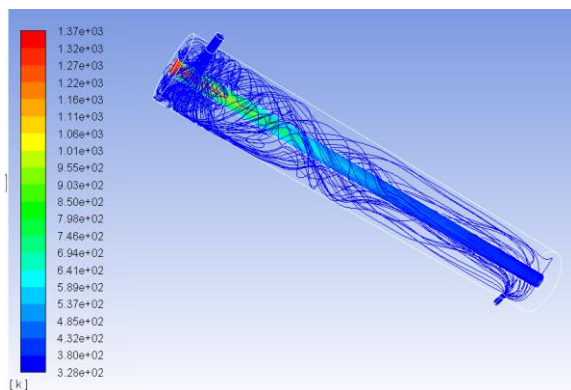


Fig. 8.32 The trajectories of particles colored by temperature

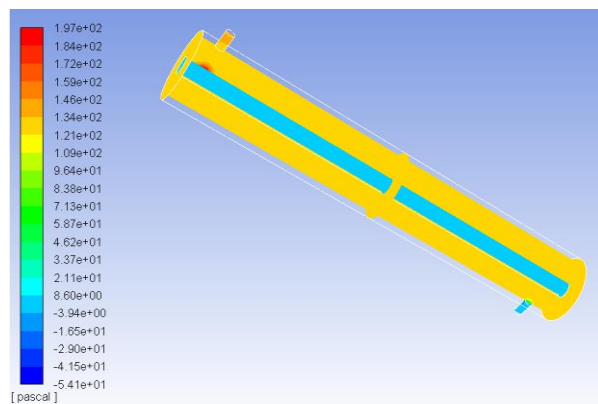


Fig. 8.33 Mean static pressure in axial direction and in three transverse planes

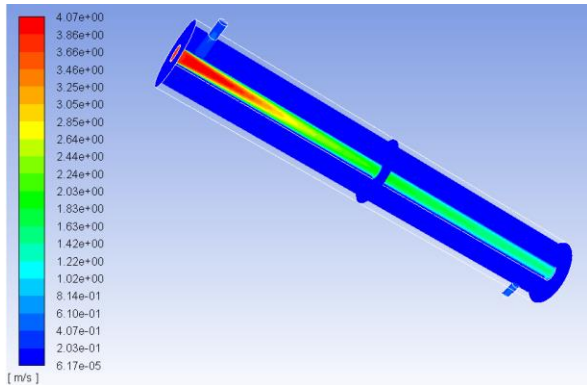


Fig. 8.34 Mean velocity magnitude in axial direction and in three transverse planes

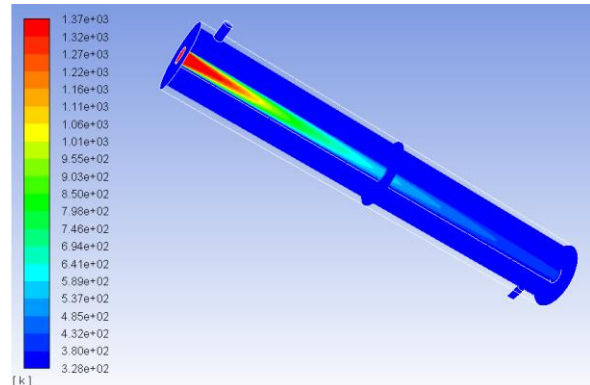


Fig. 8.35 Mean static temperature in axial direction and in three transverse planes

The effective viscosity (sum of turbulent and molecular viscosity) is very low, at least surpassing turbulent viscosity, see **Fig. 8.36**. This means that the flow is assumed in the transition between the laminar and the turbulent model, as evidenced by the value of the Re numbers in the initial estimation. The dimensionless dimension of the first cell near the wall of the inner tube is up to 10 (Fig. 8.37), meaning that the mesh is very fine, but it could be more coarsen at the wall. Enhanced Wall Treatment (two-layer wall function, y^+ should be up to 5) is better than the logarithmic wall function (y^+ should be between 30 and 60).

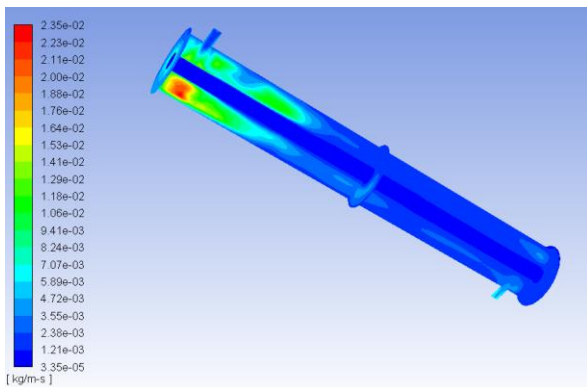


Fig. 8.36 Effective viscosity in axial direction and in three transverse planes

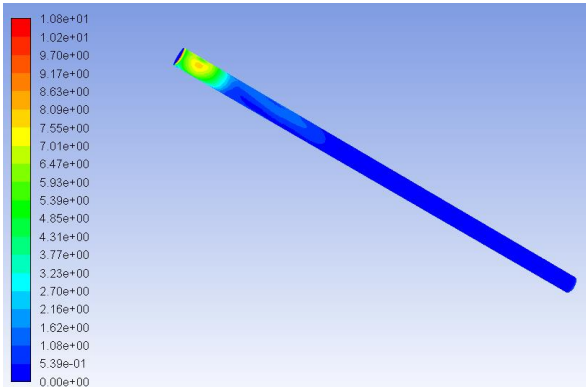


Fig. 8.37 y^+ on the tube wall

Figure 8.38 shows a typical gas temperature drop and a slight increase in water temperature, which is characteristic of the co-flow exchanger. The heat flux density is again symmetrical on the inner wall of the tube on the liquid side and on the gas side, see Figure 8.39.

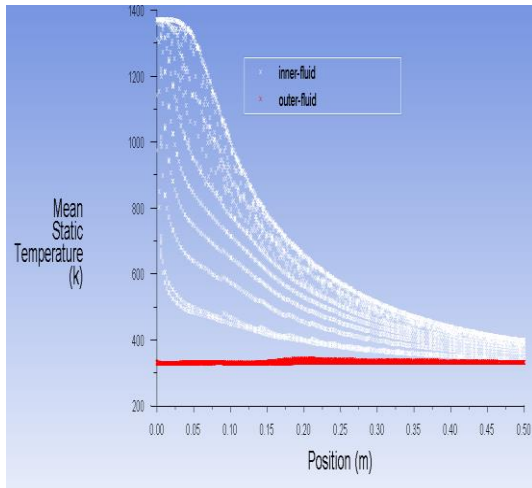


Fig. 8.38 Mean temperature of liquid and gas in axial direction

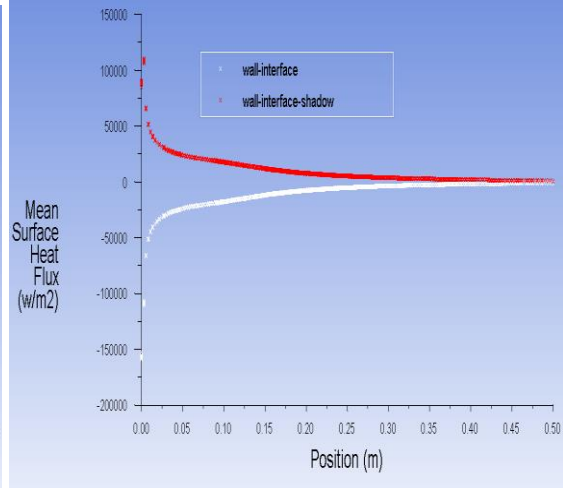
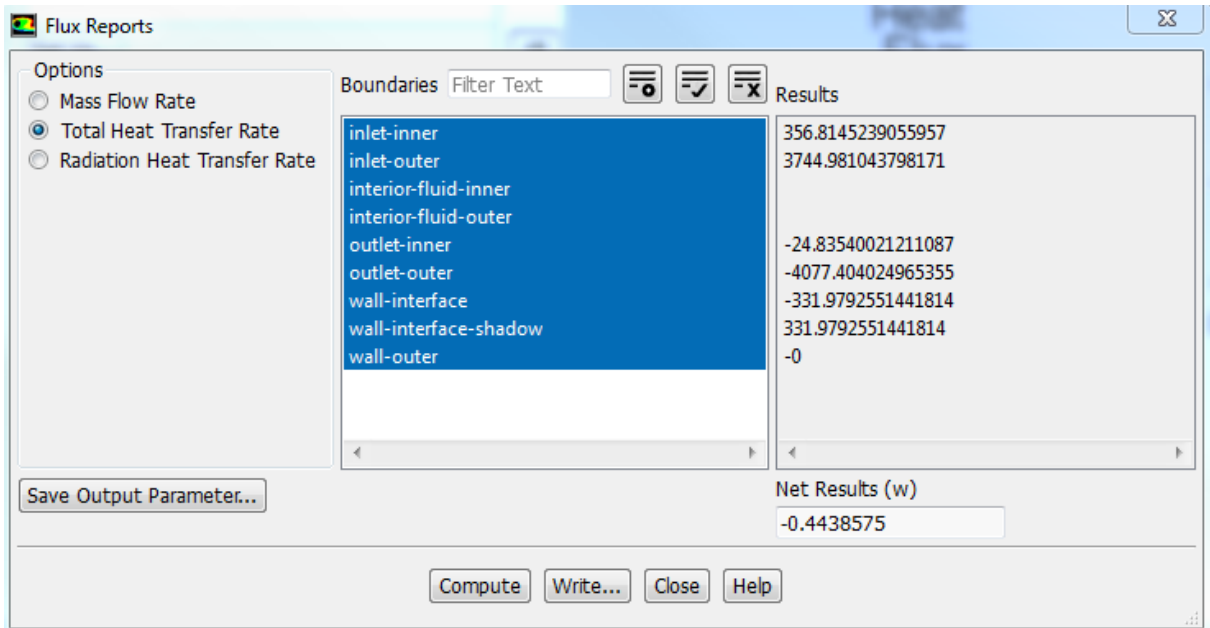


Fig. 8.39 Mean surface heat flux na the wall of inner tube

Heat power through the walls is:



An interesting result is comparison of the flow parameters in inner tube and outer tube, see Tab. 8.10

Tab. 8.10

	Tube inner Fluent	Tube outer Fluent
T_{ref}	1373	328

d_{ref}	0.02	0.01
Nu-turb	5.86	14.31
α	10.37	858.47
T_{outlet}	374.56	330.60
ρ_{inlet}	2.32	140.96

8.4.5. Co-flow heat exchanger air-water-air

Higher cooling and heating of the flowing media can be achieved by increasing the heat exchange surface. For example, an extension area and inserting another tube with the flowing air into inside area causes a significant increase in heat transfer surface, see Fig. 8.40. Inserting the tube into inner area is a simplified tube system in the axial direction.

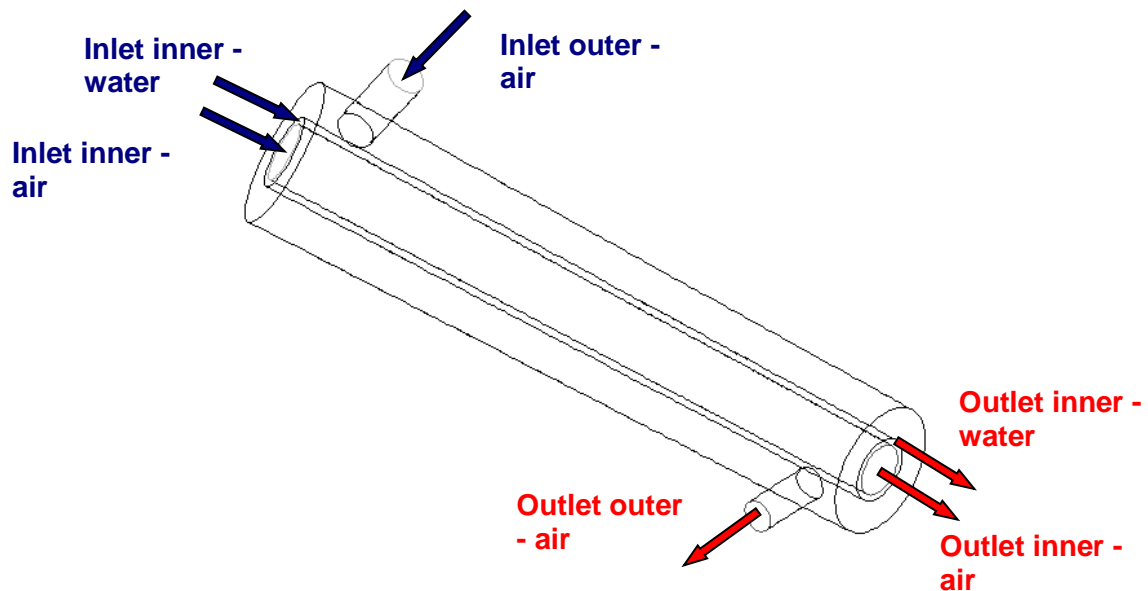


Fig. 8.40 Scheme of region

Dimensions of region, inlets of air and inlet of water are defined in Tab. 8.11.

Tab. 8.11

Region	$x=$ 0.2	m		
	$y=$ 0.09	m		
	$z=$ 0.04	m		
Inlet outer – air	$S=$ 7.65E-05	m ²	$d=$ 0.01	m
Inlet inner - water	$S=$ 5.7676E-05	m ²	$d=$ 0.02	m
Inlet inner - air	$S=$ 0.00025447	m ²	$d=$ 0.018	m

The **physical properties** of the streaming media coincide with the previous task.

Boundary conditions

The boundary conditions have been defined so that the flow of hydraulic quantities and temperatures can be monitored.

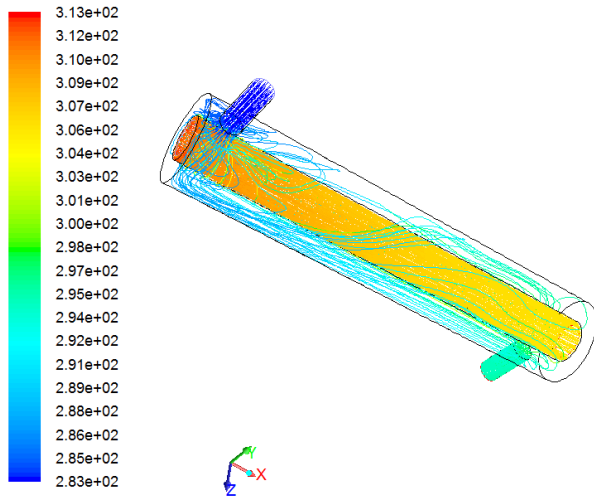


Fig. 8.41 Trajectories of particles colored by temperature

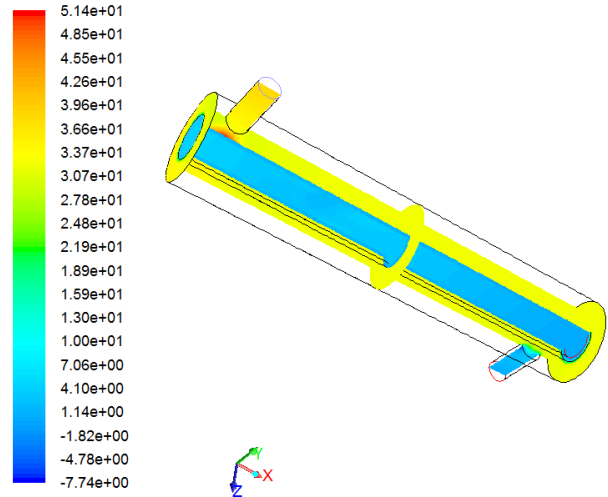


Fig. 8.42 Static pressure in axial direction and in three transverse planes

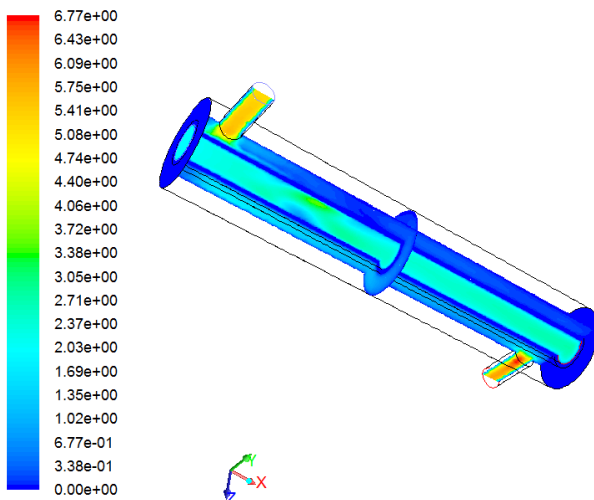


Fig. 8.43 Velocity magnitude in axial direction and in three transverse planes

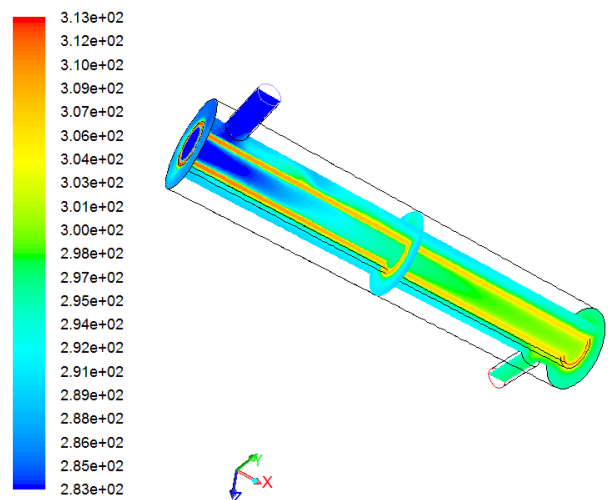


Fig. 8.44 Static temperature in axial direction and in three transverse planes

In the previous figures it is possible to evaluate the difference in co-flow water-air heat exchanger. Fig. 8.45 shows a more significant change in flow direction.

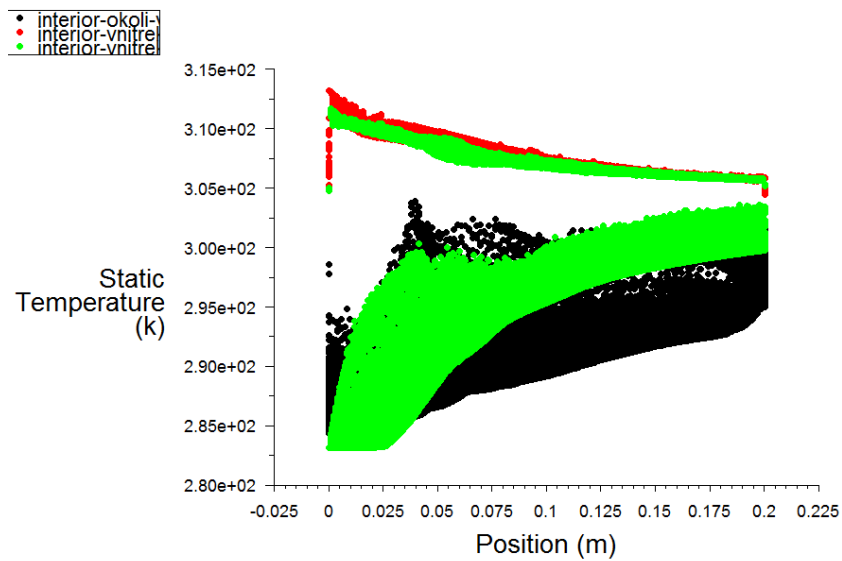


Fig. 8.45 Static temperature in interior - inner, interior – outer and on interface

Heat pover of wall outer is 6.25 W and wall inner is 9.07 W.

Tab. 8.12 Co-flow heat exchanger

air – water - air		
	static temperature [K]	static pressure [Pa]
Inlet-outer-air	283.136	38.237
Inlet-inner-water	313.150	1.869
Inlet-inner-air	283.148	0.4817
Outlet-outer-air	295.591	0
Outlet-inner-water	305.820	0
Outlet-inner-air	301.552	0

9. Time dependent solution

Generally the flow around the body is typical by separating flow and vortex shedding, which are time-dependent structures. Flow is time-dependent. Numerical solution of such flow in both laminar and turbulent regime is time-dependent too, see Fig. 9.1. Response to time-dependent boundary conditions is time-dependent too, see Fig. 9.2. Then you must undergo a complicated and time-consuming solution. It is a function of time.

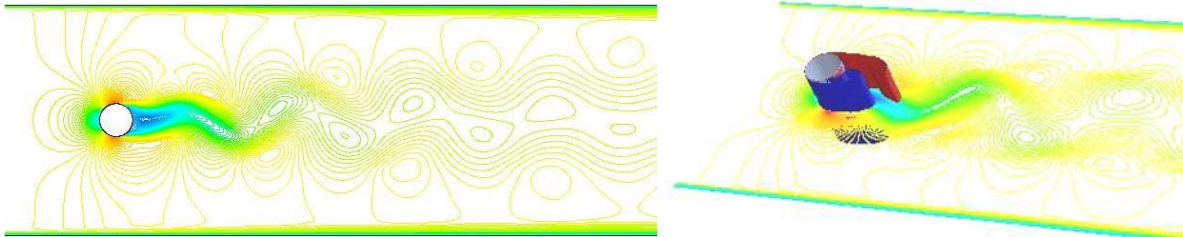


Fig. 9.1 The formation of vortex shedding when flowing around cylinder [18]

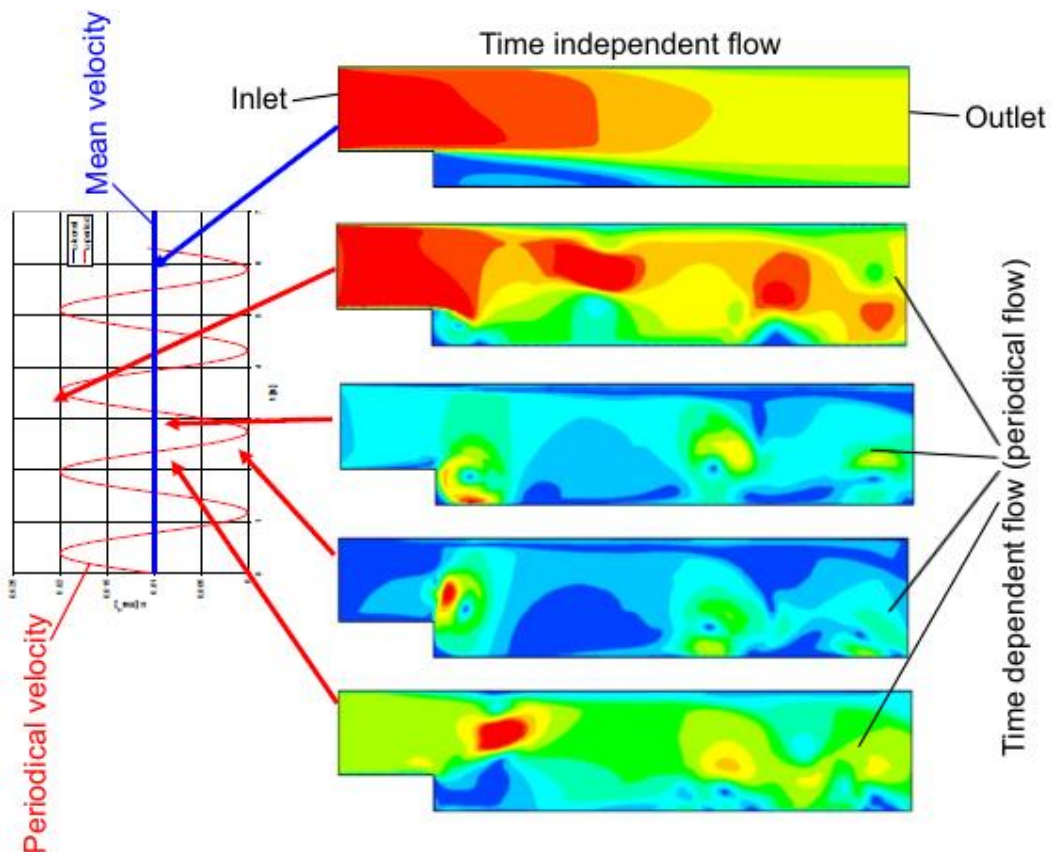


Fig. 9.2 Progress of velocity as a response to a constant and sinusoidal velocity at the input on the left side (periodic boundary condition produces periodic flow in the area)

9.1. Discretization of time-dependent equation

In case of time dependent flow it is assumed default balance equation (for simplicity, one-dimensional shape) for general variable in the form

$$\frac{\partial \zeta}{\partial t} + \frac{\partial}{\partial x}(u\zeta) = \frac{\partial}{\partial x} \left[\alpha_\zeta \frac{\partial \zeta}{\partial x} \right] + S_\zeta \quad (9.1.1)$$

In integral form it is

$$\int_V \frac{\partial \zeta}{\partial t} dV + \int_A (u\zeta) dA = \int_A \left[\alpha_\zeta \frac{\partial \zeta}{\partial x} \right] dA + \int_V S_\zeta dV \quad (9.1.2)$$

The default equation must be discretized in time and space. Spatial discretization for time-dependent equation is identical to the stationary task. Time discretization includes the integration of each member of differential equations with a time step Δt . Integration of time expression is simple, as will be described below.

The above equation is written in general form

$$\frac{\partial \zeta}{\partial t} = F(\zeta) \quad (9.1.3)$$

where the function F contains spatial discretization. On the time derivative it is applied differential approximation of the first-order forward, then discretized equation is given as

$$\frac{\partial \Phi}{\partial t} \cong \frac{\Phi^{n+1} - \Phi^n}{\Delta t} = F(\Phi) \quad (9.1.4)$$

and eventually discretization of second order accuracy is

$$\frac{3\zeta^{n+1} - 4\zeta^n + \zeta^{n-1}}{\Delta t} = F(\zeta) \quad (9.1.5)$$

where ζ general scalar variable

$n+1$ value in following time $t+\Delta t$

n value in time t

$n-1$ value in previous time $t-\Delta t$

Time discretization of default equation (9.1.2) assumes an implicit approach, ie convection, diffusion and source term are evaluated in time $t+\Delta t$

$$\int_V \frac{\partial \zeta}{\partial t} dV + \int_A \zeta^{n+1} u^{n+1} dA = \int_A \alpha_\zeta \frac{\partial \zeta^{n+1}}{\partial x} dA + \int_V S_\zeta^{n+1} dV \quad (9.1.6)$$

In the iterative scheme, all equations are solved iteratively for a given time step until convergence is reached. Thus, the solution in each time step requires a certain number of external iterations until it converges within each time step (corresponding converging stationary tasks in each time step).

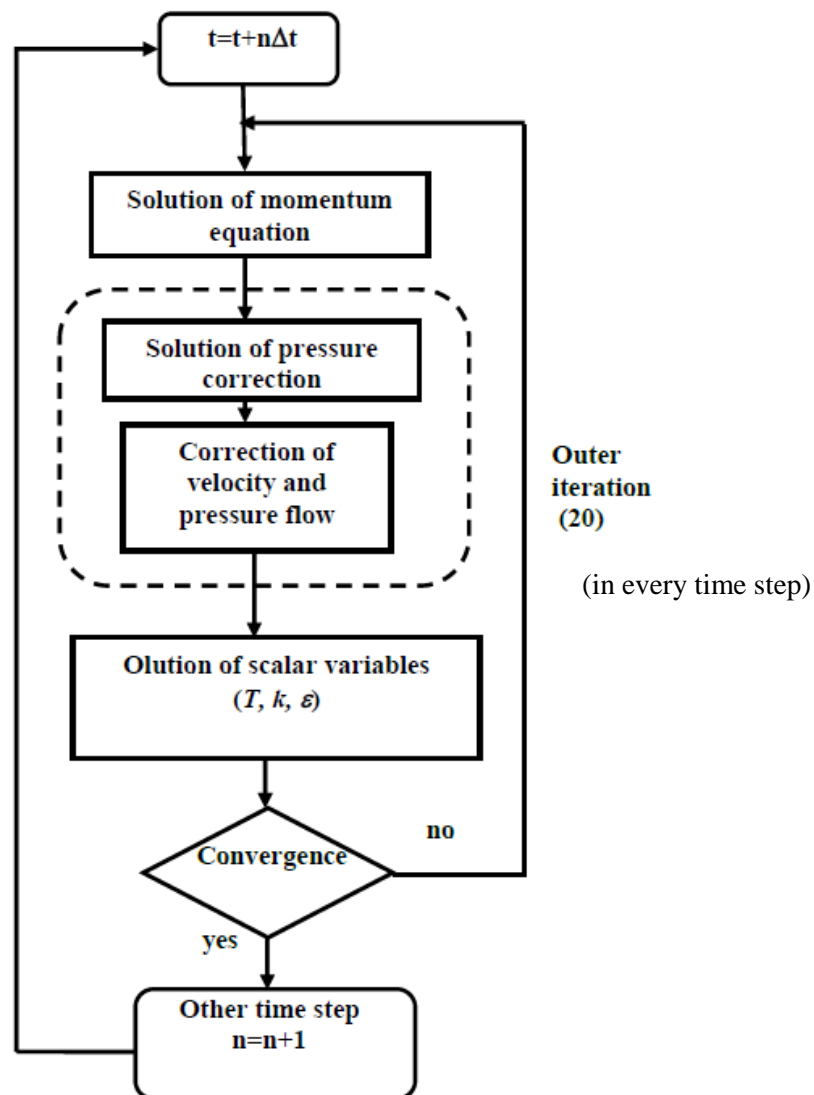


Fig. 9.3 Diagram of solution using segregation of solver.

The choice of the time step is problematic. If the time dependence is caused by the known boundary condition, then it is possible to approximately estimate the time step. Otherwise, the time dependence due to eg. vortex shedding behind the sharp edge, it is necessary to test the size of time step at the beginning of the calculation, and meet the following requirements

- **ideal recommended number of outer iterations at each time step is 10-20**
- **a bigger number of iterations means a big time step**
- **a smaller number of iterations means a small time step**
- **beginning of calculation will be realized for relatively small time step Δt and during the calculation gradually it can be increased**

Evaluation of calculation with time-dependent step is possible with automatically storing data files using the FILE-WRITE-AUTO-SAVE. It is a regular saving of solution results after a certain number of time steps during calculation. Another option is to store the values of selected variables in a specific location of the area during the time-dependent solutions, to monitor their changes over time and to assess whether eg. the solution approaches steady state while monitoring the run-up system. First, it creates so-called monitoring points in the menu SURFACE-POINT by entering the exact coordinates of the point or by mouse estimation. In the command SOLVE-MONITORS-SURFACE INTEGRALS it is then possible to select the point and variable being evaluated. Record versus time can be recorded in a file and in a graph on a monitor. Of course the optimal evaluation is by animations created directly by software during the calculation.

9.2. Boundary conditions

Time-dependent boundary conditions can be entered in two ways:

- using the file (table) to define a profile
- UDF (User Defined Function) – to define by the C language, store, compile, assign the boundary conditions using the file (table)

9.2.1. Table for time-dependent boundary condition

Table is created by text editor with extension TXT. Format of such table is following:

```
profile-name n_field n_data periodic?  
field-name-1 field-name-2 field-name-3 .... field-name-n_field  
v-1-1 v-2-1 ... .. v-n_field-1  
v-1-2 v-2-2 ... .. v-n_field-2  
...  
v-1-n_data v-2-n_data ... .. v-n_field-n_data
```

where profile-name	name of all variables
n-field	number of variables
n-data	number of data characterizing the functional dependence (number rows in the table)
periodic?	equals 1 for periodic condition,

	equals 0 for aperiodic condition
field-name-1	is exclusively used for a time vector whose values must increase
field-name-i	vectors of other time dependent variables
v-1-1 ... v-n_field-n_data	items in a matrix whose columns correspond to the time dependencies of the vectors of time

File of dependence of velocity vs time

Table of input values		Table of input values for Fluent
<i>time</i>	<i>u</i>	sampletabprofile 2 3 0
1	10	time u
2	20	1 10
3	30	2 20
		3 30

File of periodical dependence of velocity vs time

Table of input values		Table of input values for Fluent
<i>time</i>	<i>u</i>	sampletabprofile 2 4 1
0	10	time u
1	20	0 10
2	30	1 20
3	10	2 30
		3 10

All variables must be entered in SI units (no conversion of data is required when reading a profile, and only lowercase characters are used to label the variables). The profile is read from the text menu by the following commands:

```
FILE-READ TRANSIENT TABLE
```

It is possible to use abbreviations (f-rtt). The file name is also given with the extension, the file reading information appears on the screen. The profile then enters by commands in the boundary condition

```
DEFINE-BOUNDARY CONDITIONS
```

9.2.2. UDF for boundary condition

Time-dependent boundary conditions can be defined by the C-language procedure. Variables have a precisely defined by designation that must be found in the manual, there are also simple examples.

Define x-coordinate of velocity at inlet using sine function of time

$$u_x(t) = u_0 + A\sin(\omega t):$$

```
/******  
unsteady.c  
UDF for specifying a transient velocity profile boundary condition  
*****/  
  
#include "udf.h"  
  
DEFINE_PROFILE(unsteady_velocity, thread, position)  
{  
    face_t f;  
    real t = CURRENT_TIME;  
  
    begin_f_loop(f, thread)  
    {  
        F_PROFILE(f, thread, position) = 10. + sin(7.*t);  
    }  
    end_f_loop(f, thread)  
}
```

The file will be created as a *.txt file and will be saved with the extension C. It will be compiled interactively using the DEFINE-UDF-ITERPRETED-COMPILE commands. Then it joins the boundary conditions for the given entry boundary.

9.3. Time-dependent tasks, evaluation

Time-dependent problem in comparison with the time-independent (stationary task) is much more complex because at each time there is observed a change of flow field and thus all monitored values.

The most perfect image of solution enables animation, for example the velocity vector, pressure and other variables. But this is very demanding in terms of hardware. In addition, the presentation of the results requires a computer that does not fit into text reports. If it is really necessary to present time dependence of variables in text report, it is possible to create a series of images so that the data files are stored in predefined time steps, then create a graphic presentation and insert a picture into a text file. Because of the time and hardware requirements the simpler means are used for evaluation, as are the graphs of a variable vs. time at a predefined point, or evaluation of the mean value on the surface.

Example of evaluation of flow behind the step

Solve the flow in the area where at the input the velocity is varied periodically according to the functional velocity versus time. Evaluate the velocity and pressure at selected points.

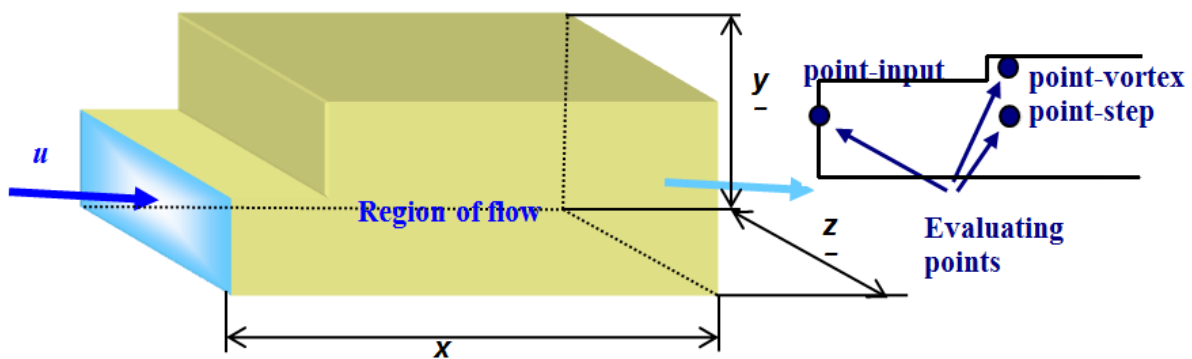


Fig. 9.4 Scheme of resolved area

Geometry of area:	x [m]	3.5
	y [m]	0.5
	z [m]	1.5
Physical properties of air:	density ρ [$\text{kg}\cdot\text{m}^{-3}$]	1.225
	dynamic viscosity μ [$\text{Pa}\cdot\text{s}$]	1.7894e-05
Boundary conditions at inlet:	velocity u [$\text{m}\cdot\text{s}^{-1}$]	$2 + \sin(7t)$
	mean velocity u_{mean} [$\text{m}\cdot\text{s}^{-1}$]	2
	intensity of turbulence I [%]	2
	hydraulic diameter d_h [m]	0.4
Mathematical model:	Reynolds number $Re = 235\,294$	turbulent flow

Results:

It is necessary to define the points at which the graphic and text record of velocity and pressure will be performed.

Commands:

Point definition:

SURFACE-POINT coordinates and name: POINT-INPUT, POINT-STEP, POINT-VORTEX

Definiton of file:

SOLVE-MONITOR-SURFACE name, plot, write time step, define

DEFINE-AREA WEIGHTED AVERAGE- point choice POINT-INPUT

FLOW TIME-PRESSURE

SOLVE-MONITOR-SURFACE name, plot, write time step, define

DEFINE-AREA WEIGHTED AVERAGE- point choice POINT-STEP

FLOW TIME-VELOCITY

SOLVE-MONITOR-SURFACE name, plot, write time step, define

DEFINE-AREA WEIGHTED AVERAGE- point choice POINT-VORTEX

FLOW TIME-VELOCITY

Determined period is depending on the velocoity input

$$\omega = 2\pi f = 7 \rightarrow f = \frac{\omega}{2\pi} = \frac{7}{2 \cdot 3,14} = 1,115 \rightarrow T = 0,897$$

Computation starts with time-dependent calculation, the estimated time step (less than one tenth of period)

$$\Delta t = 0,02s.$$

When calculating it is checked whether the number of internal iterations is less than 20, otherwise the time step will be corrected. The calculation result is written into the file- POINT-INPUT.OUT, POINT-STEP.OUT and POINT-VORTEX.OUT. The files are text files and can be read into EXCEL to prepare graphs. The recording residuals shown the periodicity happens except for the first few iterations, which are affected by the calculation starting from an initial approximation, which is given by the zero-values of variables, see Fig. 9.5.

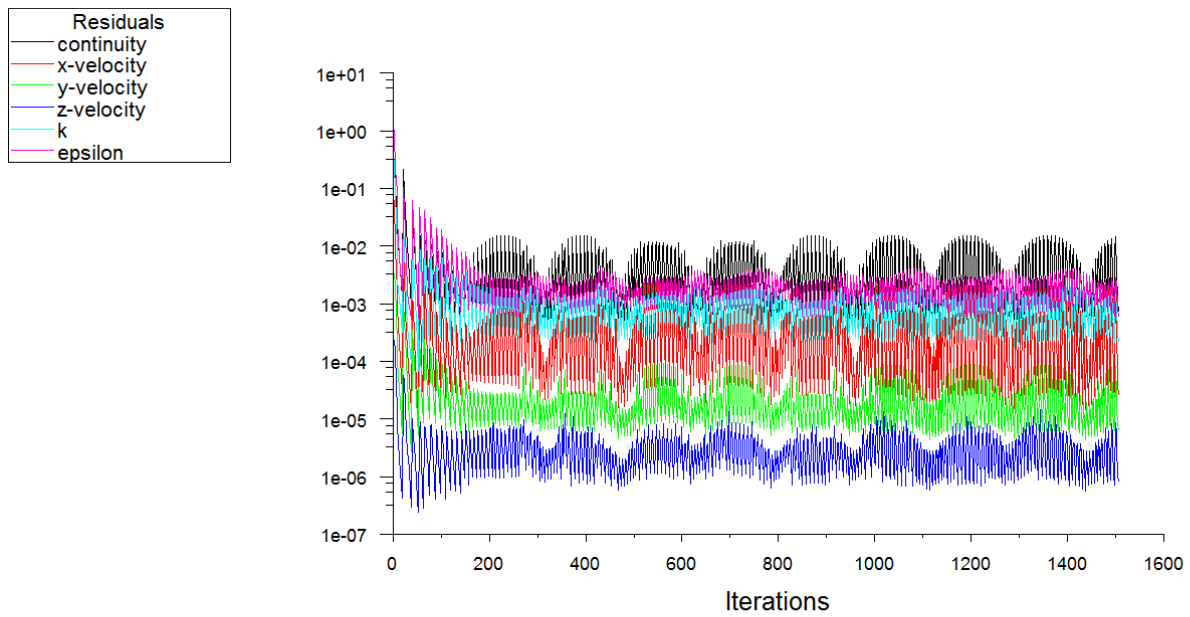


Fig. 9.5 Residuals of periodical solution

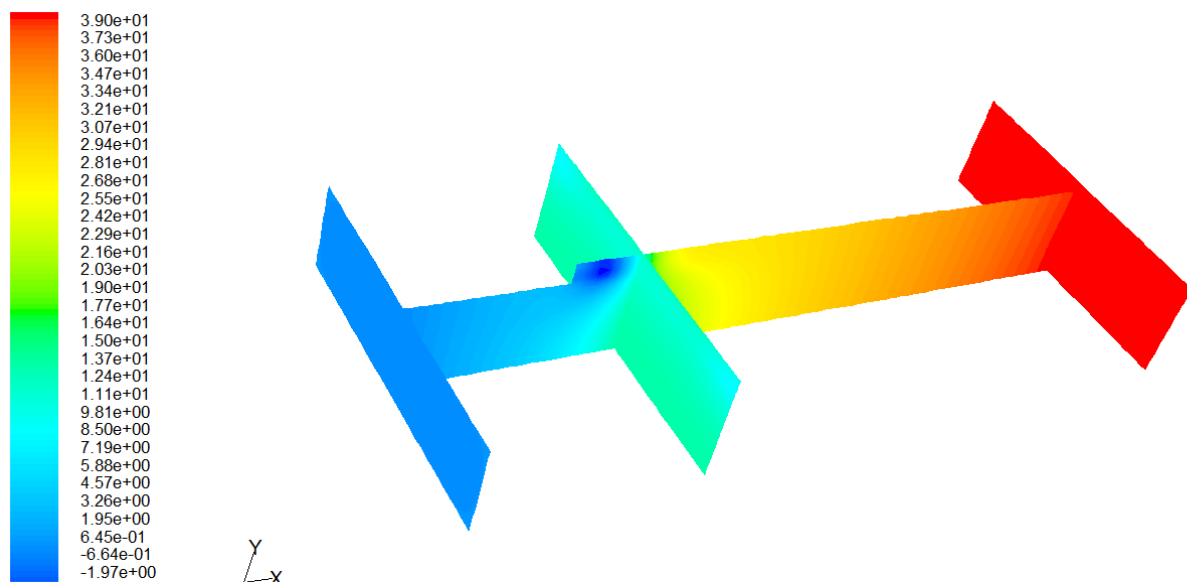


Fig. 9.6 Static pressures

Velocity value at POINT-STEP, POINT-VORTEX and pressure at POINT-INPUT are evaluated and displayed in Excel, see Fig. 9.7.

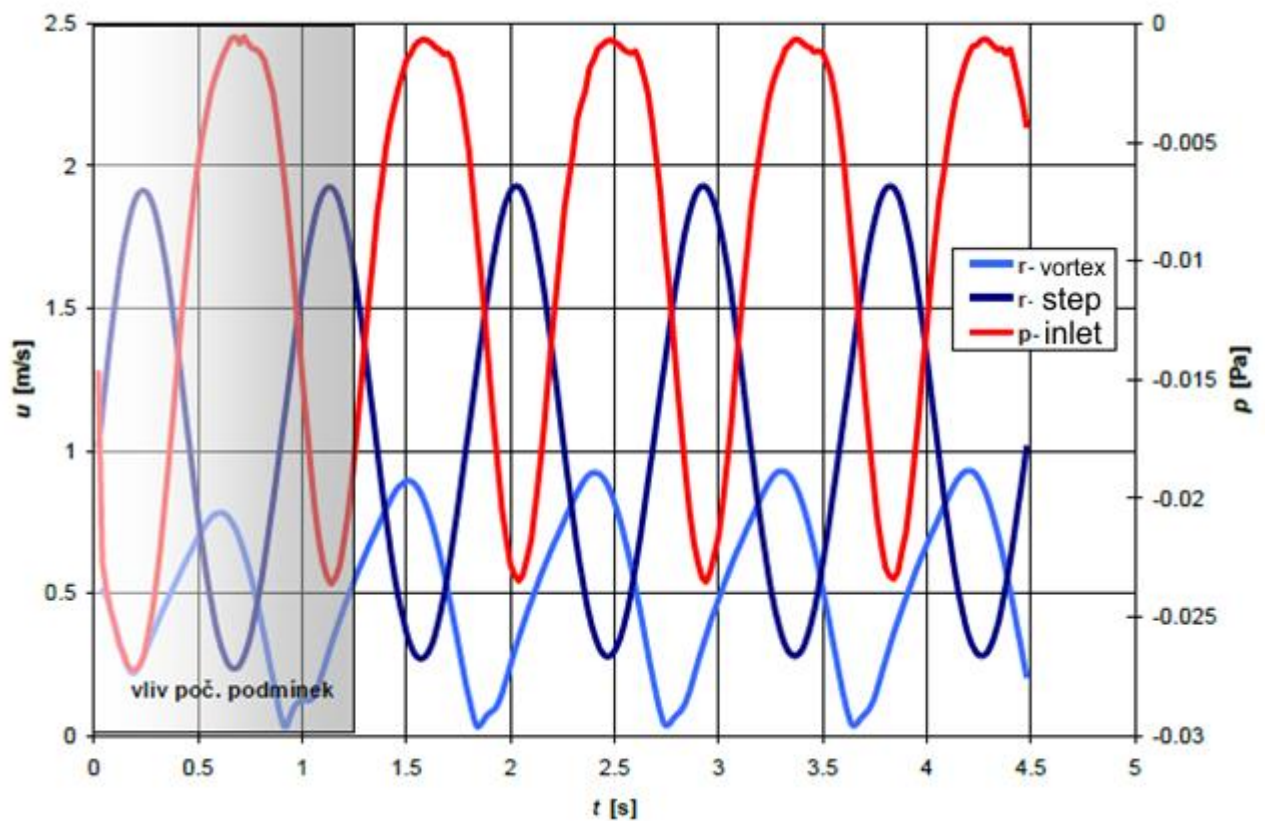


Fig. 9.7 Evaluation of velocity and pressure in a point in relation to time

On Fig. 9.7 it is evident periodical curve of velocity and pressure, whose period is equal to the period of input velocity determined previously. Period is initially deformed by calculation starting with zero initial conditions inside the flow field. After about 1 s the amplitude of the displayed functions is constant.

10. Optimization (Adjoint solver)

The idea of the adjoint is everywhere in modern and classical mathematics. Dates back to the 18th century. More recently it has emerged as a powerful technique for expanding engineering CFD analysis. An adjoint solver provides specific information about a fluid system that is very difficult to gather otherwise. An adjoint solver can be used to compute the derivatives of an engineering quantity with respect to all of the inputs for the system. The examples are:

- Derivative of drag with respect to the shape of a vehicle.
- Derivative of total pressure drop with respect the shape of the flow path.

An adjoint solver is a specialized tool that extends the scope of the analysis provided by a conventional flow solver by providing detailed sensitivity data for the performance of a fluid system. In order to perform a simulation using the ANSYS Fluent standard flow solvers, a user supplies the geometry in the form of a computational mesh, specifies material properties and physics models, and configures boundary conditions of various types. The conventional flow solver, once converged, provides a detailed data set that describes the flow state governed by the flow physics that are being modeled. If a change is made to any of the data that defines the problem, then the results of the calculation can change. The degree to which the solution changes depends on how sensitive the flow is to the particular parameter that is being adjusted. Indeed, the derivative of the solution data with respect to that parameter quantifies this sensitivity to first order. Determining these derivatives is the domain of sensitivity analysis.

The process of computing an adjoint solution resembles that for a standard flow calculation in many respects. The adjoint solver solution advancement method is specified, residual monitors configured, and the solver is initialized and run through a sequence of iterations to convergence. One notable difference is that a scalar-valued observation is selected as being of interest prior to starting the adjoint calculation. Once the adjoint solution is converged the derivative of the observable with respect to the position of each and every point on the surface of the geometry is available, and the sensitivity of the observation to specific boundary condition settings can be found. This remarkable feature of adjoint solutions has been known for hundreds of years, but only in the last 25 years has the significance for computational physics analysis been recognized widely.

10.1. Using the adjoint solver

This chapter describes the process for working with the adjoint solver module, as well as setting up, running, and postprocessing the adjoint solutions. Also, this chapter demonstrates the shape modification process that is guided by the adjoint solution. The typical use of the adjoint solver involves the following steps:

- Load or compute a conventional flow solution.
- Load the adjoint solver module.
- Specify the observable of interest.
- Set the adjoint solver controls.
- Set the adjoint solver monitors and convergence criteria.
- Initialize the adjoint solution and iterate to convergence.
- Post-process the adjoint solution to extract the sensitivity of the observable with respect to boundary condition settings.
- Post-process the adjoint solution to extract the sensitivity of the observable with respect to shape of the geometry.
- Modify boundary shapes based on shape-sensitivity data and recompute the flow solution.

10.2. Limitations of adjoint solver

Adjoint solver is a method that has some limitations and is implemented on the following basis:

- The flow state is specified as a steady incompressible single-phase flow in an inertial reference frame that is either laminar or turbulent.
- For turbulent flows a frozen turbulence assumption is made, in which the effect of changes to the state of the turbulence is not taken into account when computing sensitivities.
- For turbulent flows standard wall functions are employed on all walls.
- The adjoint solver uses methods that are first order accurate in space by default. If desired, you can select second order accurate methods
- The boundary conditions are only of the following types:
 - Wall
 - Velocity inlet
 - Pressure outlet
 - Symmetry

– Rotational and translation periodic

It is important to note that these requirements are not strict limitations for the conventional flow solver, but they are limitations for the adjoint solver. For hydraulic and pneumatic tasks (ie flow in closed areas), it is appropriate to use the part of the solver for the optimization of the pressure loss. For flow around the body, resistance and pressure forces are evaluated. Also, the combination of the monitored parameters is very illustrative.

Notes:

For tasks that converge well, it is possible to prepare computational automation to make it easier to work with the adjoint solver. In tasks with complicated geometry that converge poorly in the basic solver, it is possible to assume problematic convergence even when using the adjacent solver. The reason is that linearized Navier's Stokes equations (which are in essence a non-linear system of partial differential equations) are dealt with.

10.3. Application

10.3.1. Minimizing the pressure drop in 180° elbow

The goal of the task is to reduce the pressure loss in the 180° tube elbow using the Adjoint solver and to achieve a higher uniformity of the output profile. The problem is solved as spatial, see Fig. 10.1. The flow is assumed to be stationary, isothermal, turbulent ($k-\varepsilon$ model, scalable wall functions). The flowing medium is air. The physical properties of the air are given in Tab. 10.1.

Tab. 10.1 Dimensions and physical properties of air

Diameter of the tube d (mm)	20
Viscosity μ (kg/m.s)	$1,789 \cdot 10^{-5}$
Density ρ (kg/m ³)	1,225

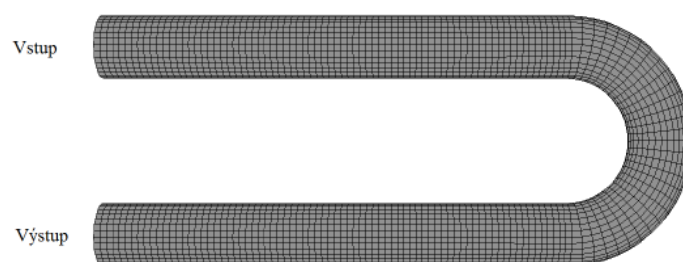


Fig. 10.1 Elbow model and mesh

Computational mesh consists from 43680 hexahedral cells, Boundary conditions are defined as very simple conditions Wall, Velocity inlet – 15 m/s, Pressure outlet – 0 Pa.

The resolver settings are as follows:

- Solver Coupled solver.
- Gradient scheme Green-Gauss Cell based.
- Discretization Second Order Upwind.
- Method Pseudo Transient
- Under-relaxation.
 - o Length scale 0,02 m.
 - o Timescale factor 5.
- High Order Term Relaxation (all variables 0,75) is used.
- Calculation till 100 iteration.

Evaluation

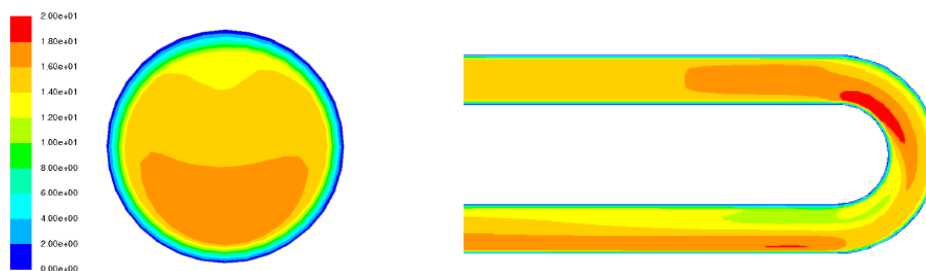


Fig. 10.2 Evaluate output speed and axial section through contours

For further consideration, mean pressure drop (ie inlet pressure) and output velocity were evaluated:

Pressure drop	107 Pa
The mean value of the output speed	15 m/s
Standard velocity deviation at output	1.14 m/s

Solving the Adjusted Solver:

- In the adjoint solver, it is possible to minimize the pressure drop between the control points (areas) in the area (between input and output - $\Delta p_0 = 90$ Pa)

- To achieve a uniform output velocity profile, the target function is complemented by a standard deviation of the output velocity of the domain and is determined from the total output pressure values of the region
- Definition of target function is a combination of pressure drop and standard deviation of the velocity at the output in one equation:

$$f_{\text{unkce}} = |dp_{\text{vstup}} - dp_{\text{výstup}}| + \sqrt{\text{var}(p_{\text{tot-výstup}})}$$

- Setting the adjoint solver:
Turn off convergence control - Check convergence,
- Make the following settings in the menu
Adjoint Solution Controls:

Courant Number	2,
Artificial Compressibility	0,05,
Flow Rate Courant Scaling	2,
Under-Relaxation Factors	0,6,
Number of iteration	300

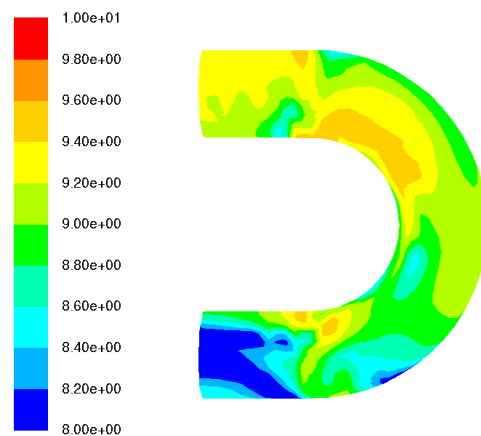


Fig. 10.3 Sensitivity map of geometry

Sensitivity evaluation and morphing:

The sensitivity map is in **Fig. 10.3**. Displays geometry locations where the pressure drop and the size of the standard deviation are sensitive to the shape of the tube. We display sensitivity map with contours $\log_{10}(\text{Shape Sensitivity Magnitude})$.

Morphing:

- Only the tube elbow is closed in the morphing area - Fig. 10.4.
- To change geometry, select Scale Factor 2.5.
- Expected change of the monitored function is done by pressing Expected Change. Optimal Displacement contours are shown in **Fig. 10.5**
- To modify the geometry, press Modify Mesh.
- Before accepting the edit (Accept), the modified geometry can be viewed and reversed if necessary (Revert).

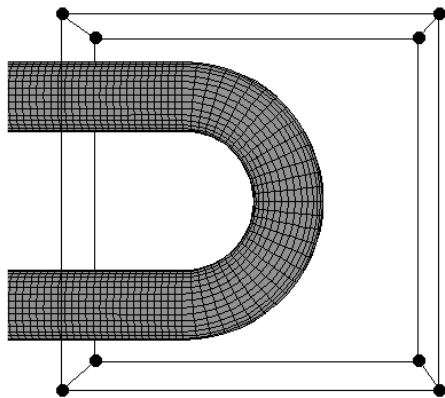


Fig. 10.4 Region of morphing

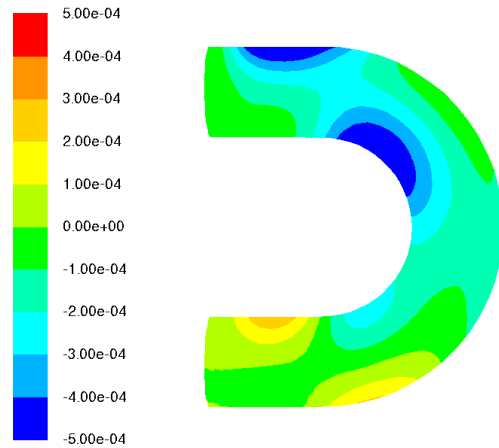


Fig. 10.5 Contours of Normal Optimal Displacement

Evaluation of the calculation:

The effect of the shape change is checked by calculation of the flow field. The table shows the change of the monitored quantities against the initial flow field. Repeating the calculation with the adjoining solver and adjusting the geometry will iteratively approximate the optimal shape of the ube elbow. The results after six iterative loops are shown in Tab. 10.2

Tab. 10.2

	Initial values	Values after 6. iteration
Pressure drop	107 Pa	89 Pa
The mean value of the output speed	15 m/s	15 m/s
Standard velocity deviation at output	1.14 m/s	1.03 m/s

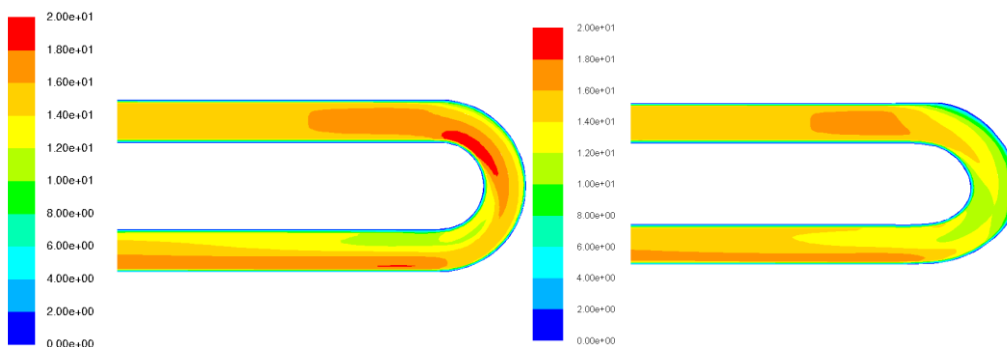


Fig. 10.6 Velocity in the domain [m/s] (left-default, right-optimized).

11. Multiphase flow

11.1. Specification of multiphase materials

A large number of flows encountered in nature and technology are the flows of phase mixture. Physical phases of matter are gas, liquid, and solid, but the concept of phase in a multiphase flow system is applied in a broader sense. In multiphase flow, a phase can be defined as an identifiable class of material that has a particular inertial response to and interaction with the flow and the potential field in which it is immersed. For example, different-sized solid particles of the same material can be treated as different phases because each collection of particles with the same size will have a similar dynamical response to the flow field.

Multiphase Flow Regimes

Multiphase flow regimes can be grouped into four categories:

- gas-liquid or liquid-liquid flows - bubbly flow, droplet flow, slug flow, stratified/free-surface flow,
- gas-solid flows - particle-laden flow, pneumatic transport, fluidized bed
- liquid-solid flows – slurry flow, hydrotransport, sedimentation
- three-phase flows - three-phase flows are combinations of the other flow regimes listed in the previous sections.

For multiphase modeling two approaches are used:

Euler – Lagrange approach (discrete phase) - the fluid phase is treated as a continuum by solving the Navier-Stokes equations, while the dispersed phase is solved by tracking a large number of particles, bubbles, or droplets through the calculated flow field. The dispersed phase can exchange momentum, mass, and energy with the fluid phase. This requires that the dispersed second phase occupies a low volume fraction, even though high mass loading is acceptable. The particle or droplet trajectories are computed individually at specified intervals during the fluid phase calculation. This makes the model appropriate for the modeling of spray dryers, coal and liquid fuel combustion, and some particle-laden flows, but inappropriate for the modeling of liquid-liquid mixtures, fluidized beds, or any application where the volume fraction of the second phase cannot be neglected.

Euler – Euler approach - in the Euler-Euler approach, the different phases are treated mathematically as interpenetrating continua. Since the volume of a phase cannot be occupied by the other phases, the concept of phasic volume fraction is introduced. These volume fractions are assumed to be continuous functions of space and time and their sum is equal to one. Conservation equations for each phase are derived to obtain a set of

equations, which have similar structure for all phases. These equations are closed by providing constitutive relations that are obtained from empirical information, or, in the case of granular flows, by application of kinetic theory.

In ANSYS Fluent, three different Euler-Euler multiphase models are available: the volume of fluid (VOF) model, the mixture model, and the Eulerian model.

VOF Model

The VOF model is a surface-tracking technique applied to a fixed Eulerian mesh. It is designed for two or more immiscible fluids where the position of the interface between the fluids is of interest. In the VOF model, a single set of momentum equations is shared by the fluids, and the volume fraction of each of the fluids in each computational cell is tracked throughout the domain. Applications of the VOF model include stratified flows, free-surface flows, filling, sloshing, the motion of large bubbles in a liquid, the motion of liquid after a dam break, the prediction of jet breakup (surface tension), and the steady or transient tracking of any liquid-gas interface.

Mixture Model

The mixture model is designed for two or more phases (fluid or particulate). As in the Eulerian model, the phases are treated as interpenetrating continua. The mixture model solves the mixture momentum equation and prescribes relative velocities to describe the dispersed phases. Applications of the mixture model include particle-laden flows with low loading, bubbly flows, sedimentation, and cyclone separators. The mixture model can also be used without relative velocities for the dispersed phases to model homogeneous multiphase flow.

Eulerian Model

The Eulerian model is the most complex of the multiphase models in ANSYS Fluent. It solves a set of momentum and continuity equations for each phase. Coupling is achieved through the pressure and interphase exchange coefficients. The manner in which this coupling is handled depends upon the type of phases involved; granular (fluid-solid) flows are handled differently than nongranular (fluid-fluid) flows. For granular flows, the properties are obtained from application of kinetic theory. Momentum exchange between the phases is also dependent upon the type of mixture being modeled. ANSYS Fluent's user-defined functions allow you to customize the calculation of the momentum exchange. Applications of the Eulerian multiphase model include bubble columns, risers, particle suspension, and fluidized beds.

To choose between the mixture model and the Eulerian model, you should consider the following guidelines:

- If there is a wide distribution of the dispersed phases (that is, if the particles vary in size and the largest particles do not separate from the primary flow field), the mixture model may be preferable (that is, less computationally expensive). If the dispersed phases are concentrated just in portions of the domain, you should use the Eulerian model instead.
- If interphase drag laws that are applicable to your system are available (either within ANSYS Fluent or through a user-defined function), the Eulerian model can usually provide more accurate results than the mixture model. Even though you can apply the same drag laws to the mixture model, as you can for a non-granular Eulerian simulation, if the interphase drag laws are unknown or their applicability to your system is questionable, the mixture model may be a better choice. For most cases with spherical particles, the Schiller-Naumann law is more than adequate. For cases with non-spherical particles, a user-defined function can be used.
- If you want to solve a simpler problem, which requires less computational effort, the mixture model may be a better option, since it solves a smaller number of equations than the Eulerian model. If accuracy is more important than computational effort, the Eulerian model is a better choice. Keep in mind, however, that the complexity of the Eulerian model can make it less computationally stable than the mixture model.

11.2. Mixture model

11.2.1. Continuity equation for mixture

The continuity equation for the mixture is

$$\frac{\partial}{\partial t}(\rho_m) + \nabla \cdot (\rho_m \vec{V}_m) = 0 \quad (11.2.1)$$

where $V_{m,j}$ are the components of the mass-averaged velocity:

$$V_{m,j} = \frac{\sum_{k=1}^n \alpha_k \rho_k V_{k,j}}{\rho_m} \quad (11.2.2)$$

and ρ_m is the mixture density

$$\rho_m = \sum_{k=1}^n \alpha_k \rho_k \quad (11.2.3)$$

where α_k is the volume fraction of phase k .

11.2.2. Momentum equation

The momentum equation for the mixture can be obtained by summing the individual momentum equations for all phases. It can be expressed as

$$\frac{\partial}{\partial t}(\rho_m \bar{v}_m) + \nabla \cdot (\rho_m \bar{v}_m \bar{v}_m) = -\nabla p + \nabla \cdot [\eta_m (\nabla \bar{v}_m + \nabla \bar{v}_m^T)] + \rho_m \bar{g} + \bar{F} + \nabla \cdot \left[\sum_{k=1}^n \alpha_k \rho_k \bar{v}_{dr,k} \bar{v}_{dr,k} \right] \quad (11.2.4)$$

where n is the number of phases, f_i are the components of body force, and μ_m is the viscosity of the mixture

$$\mu_m = \sum_{k=1}^n \alpha_k \mu_k \quad (11.2.5)$$

and $v_{dr,k,i}$ is the component of drift velocity for secondary phase

$$\bar{v}_{dr,k} = \bar{v}_k - \bar{v}_m \quad (11.2.6)$$

Equation of volume fraction of second phase is given by equation:

$$\frac{\partial}{\partial t}(\alpha_q \rho_q) + \nabla \cdot (\alpha_q \rho_q \bar{v}_m) \frac{1}{\rho_q} = -\nabla \cdot (\alpha_q \rho_q \bar{v}_{dr,q}) + \sum_{p=1}^n (\dot{m}_{pq} - \dot{m}_{qp}) \quad (11.2.7)$$

11.2.3. Mixture flow behind the step

Compute and display the volume fraction of water and air in flowing mixture.

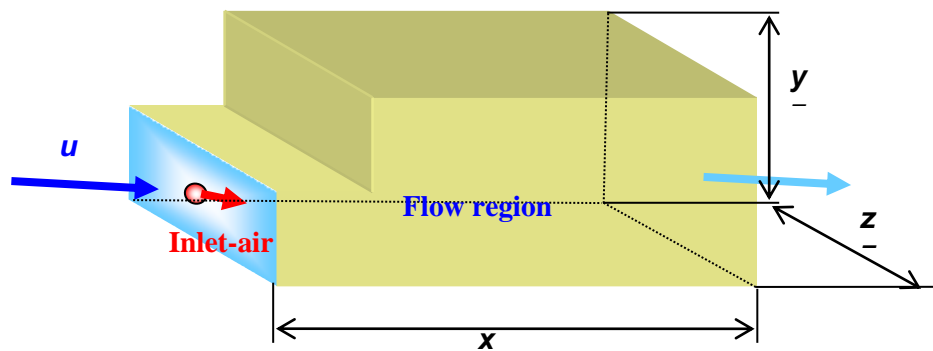


Fig. 11.1 Scheme of region

Geometry of region

Length of region x [m]	3.5
height of region y [m]	0.5
width of region Z [m]	1.5

Physical properties:

	Water	Air source
Density ρ [kgm ⁻³]	998	1.225
Dynamic viscosity η [kg(ms) ⁻¹]	0.001003	1.7894e-05

Boundary conditions:

Inlet-water	turbulent intensity I [%]	mixture	2
	hydraulic diameter d_h [m]	mixture	0.4
	velocity u [ms ⁻¹]	water	1
		air	1
volume fraction α [1]	water	1	
	air	0	
Inlet-air	turbulent intensity I [%]	mixture	5
	hydraulic diameter d_h [m]	mixture	0.0794
	velocity u [ms ⁻¹]	water	0.8
		air	0.8
volume fraction α [1]	water	0	
	air	1	

It is also possible to monitor the effect of the buoyancy force on air propagation.

Mathematical model:

Reynolds number $Re=400\ 000 \Rightarrow$ flow is turbulent

Results:

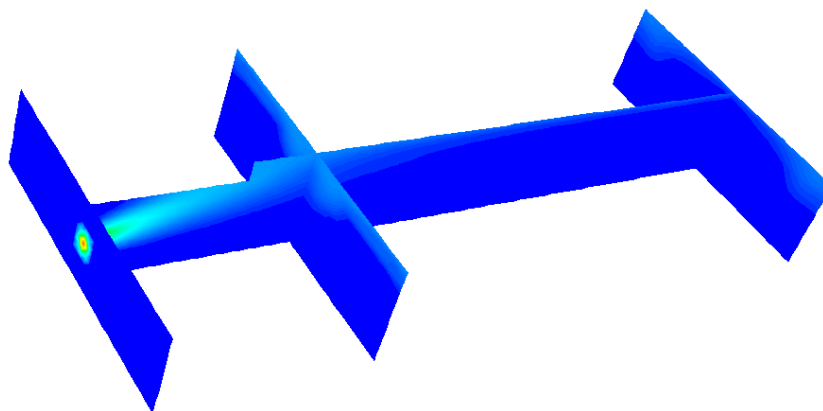


Fig. 11.2 Contours of the volume fraction of air in longitudinal and three cross sections, the buoyancy force is considered

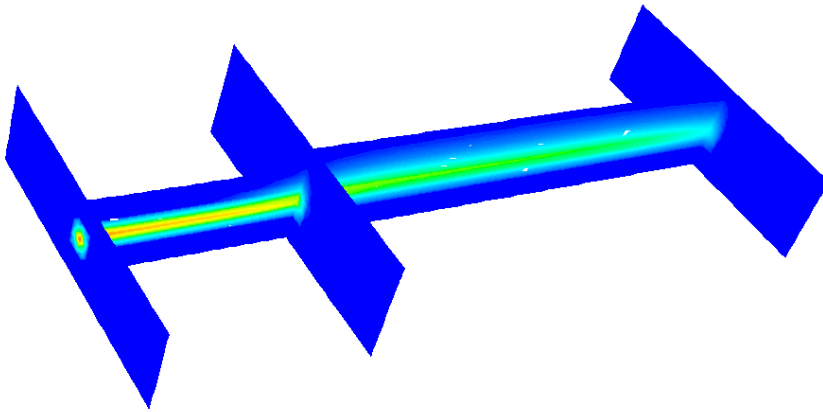


Fig. 11.3 Contours of the volume fraction of air in longitudinal and three transversal sections, no buoyancy force is considered

12. Cavitation

The flow of liquids in hydraulic systems is accompanied by cavitation and cavitation destruction, noise and other undesirable phenomena, which are currently at the forefront of interest in the areas of mathematical and experimental research on cavitation in hydraulic components and systems of technical practice, as these systems are still working in increasing operating conditions (pressures and flow rates) and, therefore, this phenomenon occurs in many technical applications.

The cavitation fluid is a mixture of liquid, vapor and air, undissolved in the form of bubbles. The gas content significantly influences the density of the liquid, resulting in a change in the dynamic properties of the liquid. Thus, cavitation occurs when certain physical states of the liquid, especially temperature and pressure, are reached. If the pressure drops to the so-called cavitation pressure, or the saturated vapor pressure, which is the function of the temperature of the liquid, the liquid continuity is disturbed and the cavitation cavity is formed by evaporation of the liquid. On the other hand, cavitation can also arise in case of separation of flow or as the result of oscillation and subsequent induced pressure waves.

Cavitation flow modeling is possible as a modeling of the multiphase flow of a mixture of liquids and gases when bubble dynamics according to the Rayleigh - Plesset equation is considered. This model is time-consuming, but due to the development of computational options it is possible to use it.

12.1. Rayleigh -Plesset thory

All cavitation models are based on the linearized Rayleigho-Plesset equation describing the growth of individual gas bubbles in the liquid:

$$\frac{p_B(t) - p_\infty(t)}{\rho_L} = R \frac{d^2 R}{dt^2} + \frac{3}{2} \left(\frac{dR}{dt} \right)^2 + \frac{4\nu_L}{R} \frac{dR}{dt} + \frac{2S_B}{\rho_L R} \quad (12.1.1)$$

where $p_B(t)$ is the bubble pressure, $p_\infty(t)$ is the pressure around the bubble, ρ_L is the density of the liquid, ν_L is the kinematic viscosity of the liquid, R is the radius of the bubble, S_B is the area of the bubble. This equation was solved by a number of scientists, such as Rayleigh and Plesset, in a simplified form (a member of surface tension, a viscous member and members with higher derivatives were neglected). The simplified differential equation in multiphase flow application was used in the form:

$$\frac{dR_b}{dt} = \sqrt{\frac{2}{3} \frac{p_{vap}(t) - p_\infty(t)}{\rho_L}} \quad (12.1.2)$$

There are several cavitation models that are different in solution approach and entering of input parameters. The tested system must contain a liquid and vapor phase. In FLUENT program, Singhal, Schnerr-Sauer and Zwart-Gerber-Belamri cavitation models are available. The Schnerr-Sauer and Zwart-Gerber-Belamri models are more stable and thus faster conversion can be expected.

12.2. Schnerr and Sauer cavitation model

This model is a possible approach to deriving a phase change from liquid to vapor. The equation for the vapor fraction is in shape

$$R = \frac{\partial}{\partial t}(\rho_{\text{vap}}\alpha) + \frac{\partial}{\partial x_j}(\rho_{\text{vap}}\alpha \bar{u}_j) \quad (12.2.1)$$

where R it presents steam generation or evaporation rate

$$R = \frac{\rho_{\text{vap}}\rho_1}{\rho} \left(\frac{\partial \alpha}{\partial t} + \frac{\partial (u_j \alpha)}{\partial x_j} \right) \quad (12.2.2)$$

When this equation is implemented in the previous equation, then next shape is obtained

$$R = \frac{\rho_{\text{vap}}\rho_1}{\rho} \alpha(1-\alpha) \frac{3}{R_b} \sqrt{\frac{2(p_{\text{vap}} - p)}{3\rho_1}} \quad (12.2.3)$$

Radius of bubble is

$$R_b = \left(\frac{\alpha}{1-\alpha} \frac{3}{4\pi n_b} \right)^{\frac{1}{3}} \quad (12.2.4)$$

Thus, in this model, the only parameter that must be defined is the number of spherical bubbles in the volume of liquid n_b . If we assume that no bubbles are formed or lost, then the density of the bubbles will be constant.

The final form of equations is:

$$\text{if } p \leq p_{\text{vap}} \quad R_e = \frac{\rho_v \rho_1}{\rho} \alpha(1-\alpha) \frac{3}{R_b} \sqrt{\frac{2(p_{\text{vap}} - p)}{3\rho_1}} \quad (12.2.5)$$

$$\text{if } p \geq p_{\text{vap}} \quad R_c = \frac{\rho_v \rho_1}{\rho} \alpha(1-\alpha) \frac{3}{R_b} \sqrt{\frac{2(p - p_{\text{vap}})}{3\rho_1}} \quad (12.2.6)$$

12.3. Cavitation in the nozzle

12.3.1. Description of experiment

Experimental equipment consisted of hydraulic circuit with circular converging-diverging nozzle, frequency converter, flowmeter, pressure and noise sensors and dissolved oxygen transmitter. The flow in the circuit was forced by the pump connected to the tank, the working liquid was water. Water passed through the flowmeter and reached the nozzle. With increasing pump motor's frequency more air was released in the nozzle. The change of oxygen content in the water was measured by the dissolved oxygen transmitter located in the tank. In the beginning and end of the nozzle the pressure sensors were installed. The nozzle is made of transparent plastic. The flow was regulated by changing the frequency of pump's motor by means of frequency converter. The photo of the circuit is shown in

Fig. 12.1. The measurements were performed at twelve different pump's motor frequencies, from 10 Hz to 32 Hz with step equal to 2 Hz, see Tab. 12.1 and Fig. 12.2 .



Fig. 12.1 Photo of circuit

Tab. 12.1 Results of measurements

f	Q_v	p_{in}	p_{out}	L
Hz	l/s	Pa	Pa	cm
18	2.61	138982	96476	1.6
20	2.79	155301	96725	2.2
22	2.95	173085	97048	4.1
24	3.15	192279	97329	5.3
26	3.31	213556	97775	7.0
28	3.49	236471	98202	9.4
30	3.68	260885	98493	11.0
32	3.88	286609	97937	11.5

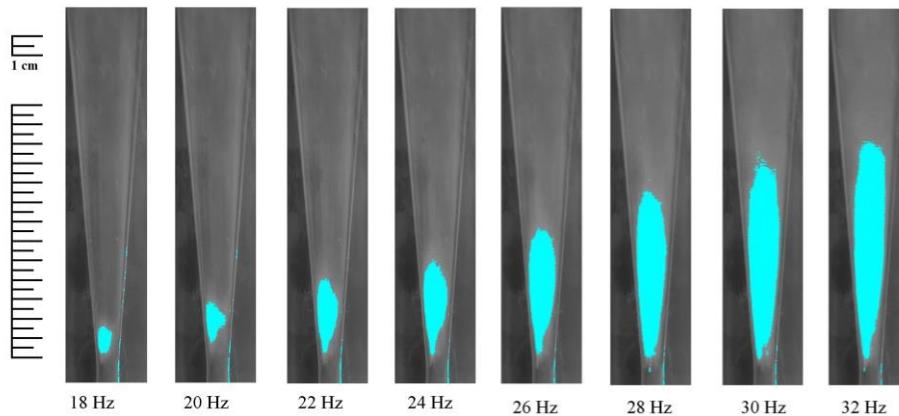


Fig. 12.2 Cavitation clouds development

12.3.2. Results

To choose the most suitable turbulent model to the further calculation the series of tests were performed. At one frequency of pump’s motor equal to 32 Hz (maximum flow) four different turbulence models were tested: $k-\epsilon$ RNG, $k-\epsilon$ realizable, $k-\omega$, SAS. Finally based on comparison of cavitation area size, the $k-\epsilon$ RNG model was chosen for further calculation and the results are in Fig. 12.3.

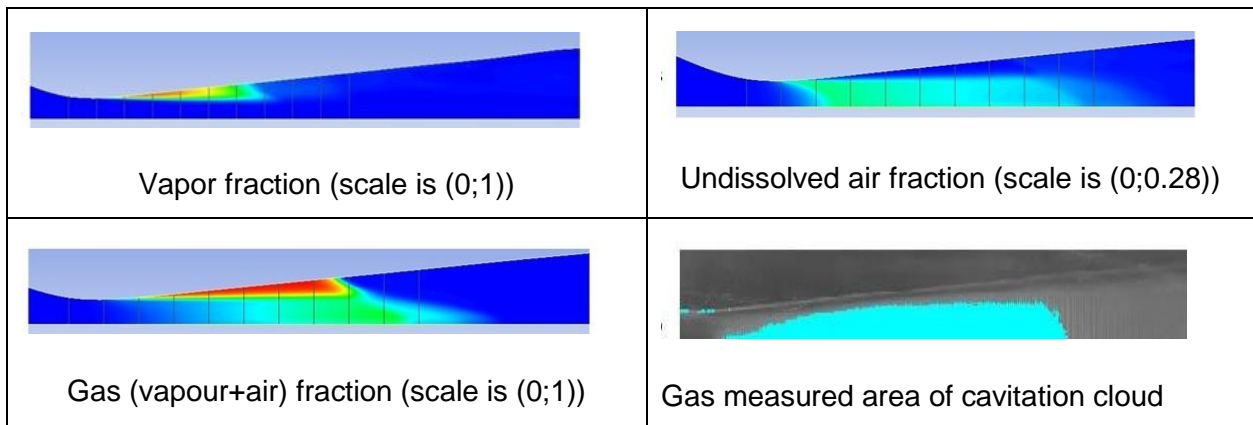
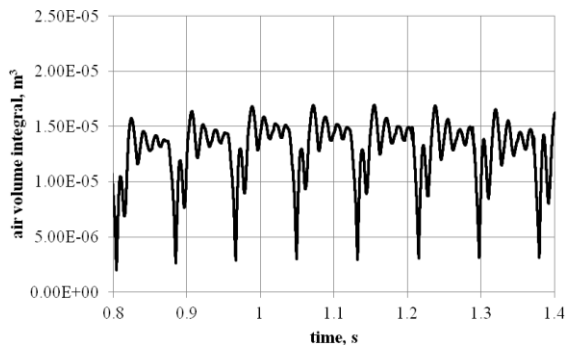
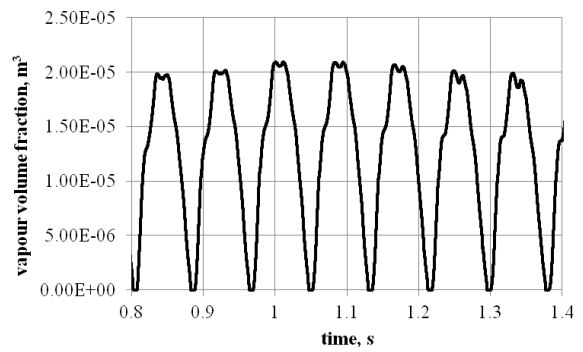


Fig. 12.3 Comparison of vapor, air, vapor+air volume fraction with experimental evaluation

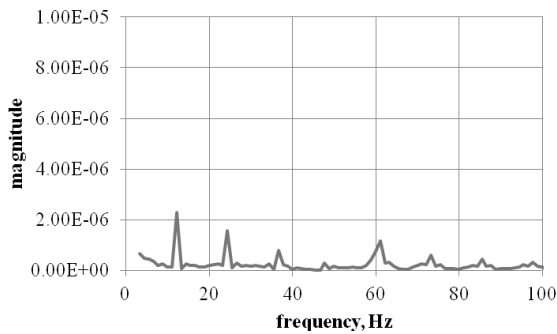
The changes of volume of air and of vapour in the domain in time are shown in Fig. 12.4. It can be observed that for vapour the one frequency dominates the others, in case of air volumes a few frequencies can be seen. The results of FFT analysis are shown in the same figures in second row.. The frequencies obtained by FFT analysis were 12.2 Hz, 24.4 Hz and 36.6 Hz.



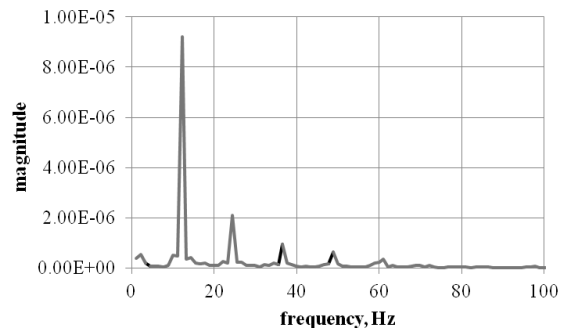
Volume integral of air phase in time, 32 Hz motor's frequency



Volume integral of vapour phase in time, 32 Hz motor's frequency



FFT analysis of volume integral of air phase, 32 Hz motor's frequency



FFT analysis of volume integral of vapour phase, 32 Hz motor's frequency

Fig. 12.4 Time series and FFT analysis of volume integral of vapor and air

After simulation performance it can be stated that in case of vapour phase there is one main growing and shrinking frequency. Meanwhile the air phase is changing with number of frequencies with no such clear domination. It also was observed that for vapour type of cavitation (which occurs at flow rate 2.61 l/s – 18 Hz motor's frequency) higher frequencies of phase changing appeared. When more air is observed in cavitation cloud, the frequency of phase growing and shrinking is decreasing (which occurs at flow rate 3.88 l/s – 32 Hz motor's frequency). That indicates that air is responsible for lower frequencies in noise analysis.

13. Appendix

13.1. Vectors and scalars

Quantities that can be determined by a single number when a unit of measure is chosen are called **scalars**. The vector is a variable that provides different data. One is arithmetic (its size), the other is geometric. The **vector** is an oriented line.

Assume a rectangular coordinate system and let the point be given by three coordinates $\vec{x}=(x, y, z)$. Let $\vec{a}_x, \vec{a}_y, \vec{a}_z$ are the projections of the vector \vec{a} in coordinate axes. These vectors are called **components** of vector \vec{a} and are valid

$$\vec{a} = \vec{a}_x + \vec{a}_y + \vec{a}_z \quad (13.1.1)$$

If \vec{i} is unit vector of axis x, je $\vec{a}_x = \vec{i} a_x$, where a_x is number, expressing the vector size \vec{a}_x and is called x-coordinate of vector, see Fig. 13.1. Similarly, $\vec{a}_y = \vec{j} a_y$ a $\vec{a}_z = \vec{k} a_z$. It is also possible to write

$$\vec{a} = \vec{i} a_x + \vec{j} a_y + \vec{k} a_z \quad (13.1.2)$$

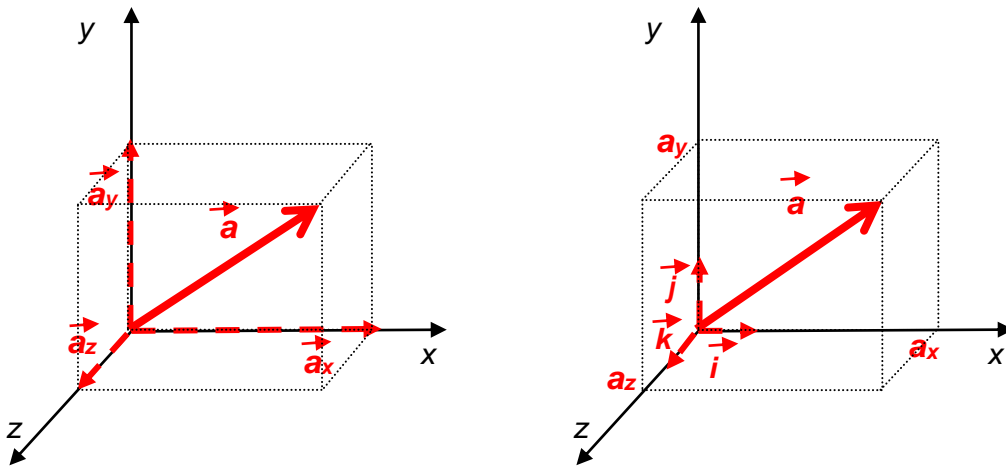


Fig. 13.1 Vector components, vector coordinates, unit vectors

The vector in the coordinate system is defined as an ordered triple of numbers and is recorded

$$\vec{a} = (a_x, a_y, a_z) \text{ resp. } \vec{a}(a_x, a_y, a_z) \quad (13.1.3)$$

The **scalar product** of vectors \vec{a} and \vec{b} with coordinates $\vec{a} = (a_x, a_y, a_z)$ and $\vec{b} = (b_x, b_y, b_z)$ is scalar

$$\vec{a} \cdot \vec{b} = a_x b_x + a_y b_y + a_z b_z, \text{ resp. } \vec{a} \cdot \vec{a} = a_x a_x + a_y a_y + a_z a_z = |\vec{a}|^2 \quad (13.1.4)$$

Dyadic vector product is tenzor

$$\vec{a} \vec{b} = \begin{pmatrix} a_x b_x & a_x b_y & a_x b_z \\ a_y b_x & a_y b_y & a_y b_z \\ a_z b_x & a_z b_y & a_z b_z \end{pmatrix} \quad (13.1.5)$$

If scalar function $f(x, y, z)$ is given, than **gradient of scalar function** is vector with coordinates $\frac{\partial f}{\partial x}$, $\frac{\partial f}{\partial y}$ and $\frac{\partial f}{\partial z}$, so

$$\text{grad} f = \nabla f = \left(\frac{\partial f}{\partial x}, \frac{\partial f}{\partial y}, \frac{\partial f}{\partial z} \right) \quad (13.1.6)$$

and a symbolic designation is often used $\nabla = \left(\frac{\partial}{\partial x}, \frac{\partial}{\partial y}, \frac{\partial}{\partial z} \right)$

If a vector \vec{a} with coordinates a_x, a_y, a_z is given, then a **gradient of vector function** is tenzor, so

$$\text{grad} \vec{a} = \nabla \vec{a} = \begin{pmatrix} \frac{\partial a_x}{\partial x} & \frac{\partial a_x}{\partial y} & \frac{\partial a_x}{\partial z} \\ \frac{\partial a_y}{\partial x} & \frac{\partial a_y}{\partial y} & \frac{\partial a_y}{\partial z} \\ \frac{\partial a_z}{\partial x} & \frac{\partial a_z}{\partial y} & \frac{\partial a_z}{\partial z} \end{pmatrix} \quad (13.1.7)$$

Let \vec{a} is a vector with coordinates a_x, a_y, a_z . Then **divergence of vector** \vec{a} is scalar

$$\text{div} \vec{a} = \nabla \cdot \vec{a} = \left(\frac{\partial}{\partial x}, \frac{\partial}{\partial y}, \frac{\partial}{\partial z} \right) \cdot (a_x, a_y, a_z) = \frac{\partial a_x}{\partial x} + \frac{\partial a_y}{\partial y} + \frac{\partial a_z}{\partial z} \quad (13.1.8)$$

Derivative of vector \vec{a} **by vector** \vec{b} is signed $(\vec{b} \cdot \text{grad}) \vec{a}$ and is defined as follows

$$\begin{aligned}
(\vec{b} \cdot \text{grad}) \vec{a} &= (\vec{b} \cdot \nabla) \vec{a} = \left(b_x \frac{\partial}{\partial x} + b_y \frac{\partial}{\partial y} + b_z \frac{\partial}{\partial z} \right) \vec{a} = \\
&= \begin{pmatrix} b_x \frac{\partial a_x}{\partial x} + b_y \frac{\partial a_x}{\partial y} + b_z \frac{\partial a_x}{\partial z} \\ b_x \frac{\partial a_y}{\partial x} + b_y \frac{\partial a_y}{\partial y} + b_z \frac{\partial a_y}{\partial z} \\ b_x \frac{\partial a_z}{\partial x} + b_y \frac{\partial a_z}{\partial y} + b_z \frac{\partial a_z}{\partial z} \end{pmatrix} = \begin{pmatrix} \frac{\partial a_x}{\partial x} b_x + \frac{\partial a_x}{\partial y} b_y + \frac{\partial a_x}{\partial z} b_z \\ \frac{\partial a_y}{\partial x} b_x + \frac{\partial a_y}{\partial y} b_y + \frac{\partial a_y}{\partial z} b_z \\ \frac{\partial a_z}{\partial x} b_x + \frac{\partial a_z}{\partial y} b_y + \frac{\partial a_z}{\partial z} b_z \end{pmatrix} \quad (13.1.9)
\end{aligned}$$

Divergence of dyadic product of vectors (resp. tensors) is :

$$\nabla \cdot (\vec{a} \vec{a}) = \begin{pmatrix} \frac{\partial(a_x a_x)}{\partial x} + \frac{\partial(a_y a_x)}{\partial y} + \frac{\partial(a_z a_x)}{\partial z} \\ \frac{\partial(a_x a_y)}{\partial x} + \frac{\partial(a_y a_y)}{\partial y} + \frac{\partial(a_z a_y)}{\partial z} \\ \frac{\partial(a_x a_z)}{\partial x} + \frac{\partial(a_y a_z)}{\partial y} + \frac{\partial(a_z a_z)}{\partial z} \end{pmatrix} \quad (13.1.10)$$

13.2. Coordinate systems

Imagine we stand on the bridge and see how the concentration of fish just in place changes with time. So we will see how the concentration varies with time in a fixed place firmly connected to the surface of the earth. This space is called **absolute space** and is the basic space. The quantity $\frac{\partial c}{\partial t}$ is **partial derivative of concentration** c with respect to t at constant coordinates x, y, z .

Now, instead of standing on a bridge, we'll get to the motorboat and ride along the river, sometimes upstream, sometimes across the river and sometimes downstream. Changing the concentration of fish with time will depend somehow on the movement of the boat. Then the **total derivative** of concentration by time is given by the relationship.

$$\frac{dc}{dt} = \frac{\partial c}{\partial t} + \frac{\partial c}{\partial x} \frac{dx}{dt} + \frac{\partial c}{\partial y} \frac{dy}{dt} + \frac{\partial c}{\partial z} \frac{dz}{dt} \quad (13.2.1)$$

where $\frac{dx}{dt}$, $\frac{dy}{dt}$ and $\frac{dz}{dt}$ are the components of boat velocity.

Now we'll get to the boat, let go with the river stream and count the fish. The observer's velocity is now the same as the flow velocity. Specifying the change in fish concentration over time depends on the local velocity of the flow. This derivative is a special kind of total derivative and is called a **substantial derivative** or a "derivative following motion". Its relation to the partial derivative by time is

$$\frac{Dc}{Dt} = \frac{\partial c}{\partial t} + \frac{\partial c}{\partial x} u_x + \frac{\partial c}{\partial y} u_y + \frac{\partial c}{\partial z} u_z \quad (13.2.2)$$

where u_x , u_y a u_z are components of local water velocity. Space is relative, ie it is small spaceje, which can move according to absolut space.

13.3. Field of velocity and acceleration

When the field fluid flows, the **velocity field** given by the vector function will be considered [1]

$$\vec{u} = \vec{u}(t, \vec{x}) \quad (13.3.1)$$

Velocity is defined at a point \vec{x} whose components depend on the selected coordinate system. In the most general case, the velocity is a three-dimensional time-dependent vector.

Fluid acceleration is prescribed in the usual way

$$\vec{a} = \frac{D\vec{u}}{Dt} \quad (13.3.2)$$

The designation of the derivative by the letter D represents a substantial derivative. Substantial derivative of the scalar (temperature, concentration) can be expressed by vector

$$\frac{D(\zeta)}{Dt} = \frac{\partial \zeta}{\partial t} + \frac{\partial \zeta}{\partial x} u_x + \frac{\partial \zeta}{\partial y} u_y + \frac{\partial \zeta}{\partial z} u_z = \frac{\partial \zeta}{\partial t} + \nabla \zeta \cdot \vec{u} \quad (13.3.3)$$

First part $\frac{\partial \zeta}{\partial t}$ is called **local time derivative** and second part $\nabla \zeta \cdot \vec{u}$ is so called **convective derivative**.

Substantial derivative of vector is more complicated and is valid (for clarity the vector is written in columns)

$$\frac{D\vec{u}}{Dt} = \begin{pmatrix} \frac{Du_x}{Dt} \\ \frac{Du_y}{Dt} \\ \frac{Du_z}{Dt} \end{pmatrix} = \begin{pmatrix} \frac{\partial u_x}{\partial t} + \frac{\partial u_x}{\partial x} u_x + \frac{\partial u_x}{\partial y} u_y + \frac{\partial u_x}{\partial z} u_z \\ \frac{\partial u_y}{\partial t} + \frac{\partial u_y}{\partial x} u_x + \frac{\partial u_y}{\partial y} u_y + \frac{\partial u_y}{\partial z} u_z \\ \frac{\partial u_z}{\partial t} + \frac{\partial u_z}{\partial x} u_x + \frac{\partial u_z}{\partial y} u_y + \frac{\partial u_z}{\partial z} u_z \end{pmatrix} = \frac{\partial \vec{u}}{\partial t} + \nabla \cdot (\vec{u} \vec{u}) \quad (13.3.4)$$

14. Literature

- [1] FLUENT: FLUENT 13 - *User's guide*. Fluent Inc. 2009 [online]. Dostupné z <<http://spc.vsb.cz/portal/cz/documentation/manual/index.php>>.
- [2] INCROPERA, F., P. ET AL.. *Fundamentals of heat and mass transfer*. 6th ed.. Hoboken : Wiley, c2007 – xxv. 997 s. ISBN 0-471-45728-0 (váz.)978-0-471-45728-2 (dotisk : váz.)
- [3] KREITH, F., RATONA, B. *Mechanical Engineering Handbook - Heat and Mass Transfer*. CRC Press LLC, 1999. CD-ROM: 2624 pages (Heat Transfer: 288 pages). ISBN-10: 0849397510. ISBN-13: 978-0849397516
- [4] SHAUGHNESSY, E.,J., . KATZ. I., M., SCHAFFER, J., P. *Introduction to fluid mechanics*. New York: Oxford University Press, 2005 - xiv, 1018, [24] s. : il. + 1 CD-ROM ISBN 0-19-515451-7
- [5] KOZUBKOVÁ, M. *Matematické modely kavitace a hydraulického rázu*. Monografie. 130. 1.vydání, Ostrava: VŠB-TU Ostrava, 2009, 130 stran. ISBN 978-80-248-2043-9.
- [6] STULL, R.B.: *An Introduction to Boundary Layer Meteorology*, Kluwer Academic Publishers, 1994, s. 251-289.
- [7] BIRD, R. B., STEWART, W. E., LIGHTFOOT, N. N. *Přenosové jevy*. Praha: Academia, 1968, 800 s.
- [8] BIRD, R. B., STEWART, W. E., LIGHTFOOT, N. N. *Transport Phenomena*. John Wiley & Sons, Inc.. New York. 914 p., ISBN 0-471-41077-2
- [9] ROSHKO, A. *On the Development of Turbulent Wakes from Vortex Streets - Technical Report* [online]. Washington, D. C.: National Advisory Committee for Aeronautics, 1954. 27 s. Dostupné z <URL: <http://authors.library.caltech.edu/428/01/ROSnacarp1191.pdf>> [cit. 2006-01-06].
- [10] DRÁBKOVÁ, S., KOZUBKOVÁ, M.: *Numerical modelling of the unsteady vortex structures due to the round Jet-cross flow interaction*. In Sborník XIV. mezinárodní vědecká konference „Aplikácia experimentálnych a numerických metód v mechanike tekutín“. Rajcecké Teplice: Žilinská univerzita Žilina, 2004, pp. 85-90, ISBN 80-8070-234-9, 29.4.-30.4.2004
- [11] DRÁBKOVÁ, S. a kol. *Mechanika tekutin*. Ostrava: VŠB-TU Ostrava, 2007. 248 s. (Elearningová učebnice). ISBN 978-80-248-1508-4.
- [12] Kolat Přenos
- [13] KOZUBKOVÁ, M.: *Modelování proudění tekutin FLUENT, CFX*. Ostrava: VŠB-TU, 2008, 154 s., ISBN 978-80-248-1913-6, (Elektronická publikace na CD ROM)
- [14] BOJKO, M.: *Návody do cvičení "Modelování Proudění" - FLUENT*, VŠB-Technická Univerzita Ostrava, 2008, 141 s., ISBN 978-80-248-1909-9.
- [15] BOJKO, M.: *3D Proudění – ANSYS Fluent*, e-learningová skripta, VŠB-Technická Univerzita Ostrava, 2010, 226s.
- [16] BLEJCHAŘ, T: *Návody do cvičení „Modelování proudění“ – CFX*, VŠB-Technická Univerzita Ostrava, 2008, 133 s., ISBN 978-80-248-2050-7.
- [17] BLEJCHAŘ, T: *Turbulence Modelování proudění – CFX*, e-learningová skripta VŠB-Technická

Literatura

Univerzita Ostrava, 2010, 259 s.Bojko

- [18] FABIÁN, P. Metody matematického a fyzikálního experimentu v proudění tekutin. Disertační práce. VŠB-TU Ostrava. Ostrava 2007. 104 s.
- [19] http://www.tenez.cz/app/clanek/161/vymeniky_tepla_vsech_druhu_a_typu
- [20] <http://www.olaer.cz/cz-produkty-prehled/cz-produkty-chladice-3/cz-prod-kuehl-rohrbund.htm>
- [21] <http://www.spiraxsarco.com/resources/steam-engineering-tutorials/steam-engineering-principles-and-heat-transfer/steam-consumption-of-heat-exchangers.asp>
- [22] http://en.wikipedia.org/wiki/Shell_and_tube_heat_exchanger
- [23] <http://www.hydro.com/en/Subsites/Hydro-Aluminium-Precision-Tubing/HVACR/Why-Aluminium-in-HVACR/Brazed-heat-exchanger/>
- [24] https://cs.wikipedia.org/wiki/Tepelný_výměník#/media/File:Tubular_heat_exchanger.png